Opbouw and test manual for the ZX81+38

For those planning to build the ZX81+38, note this kit is NOT suitable for people who are not proficient in soldering. The ZX81+38 is not easy to solder, the design is dense and the solder pads are smaller than in most other designs.

Also you will need to have the right tools, and probably a loupe or soldering microscope.

You also need to own an EPROM programmer. I suggest using the minipro TL866CS to program a 27C256 with four copies of the ZX81.ROM file.

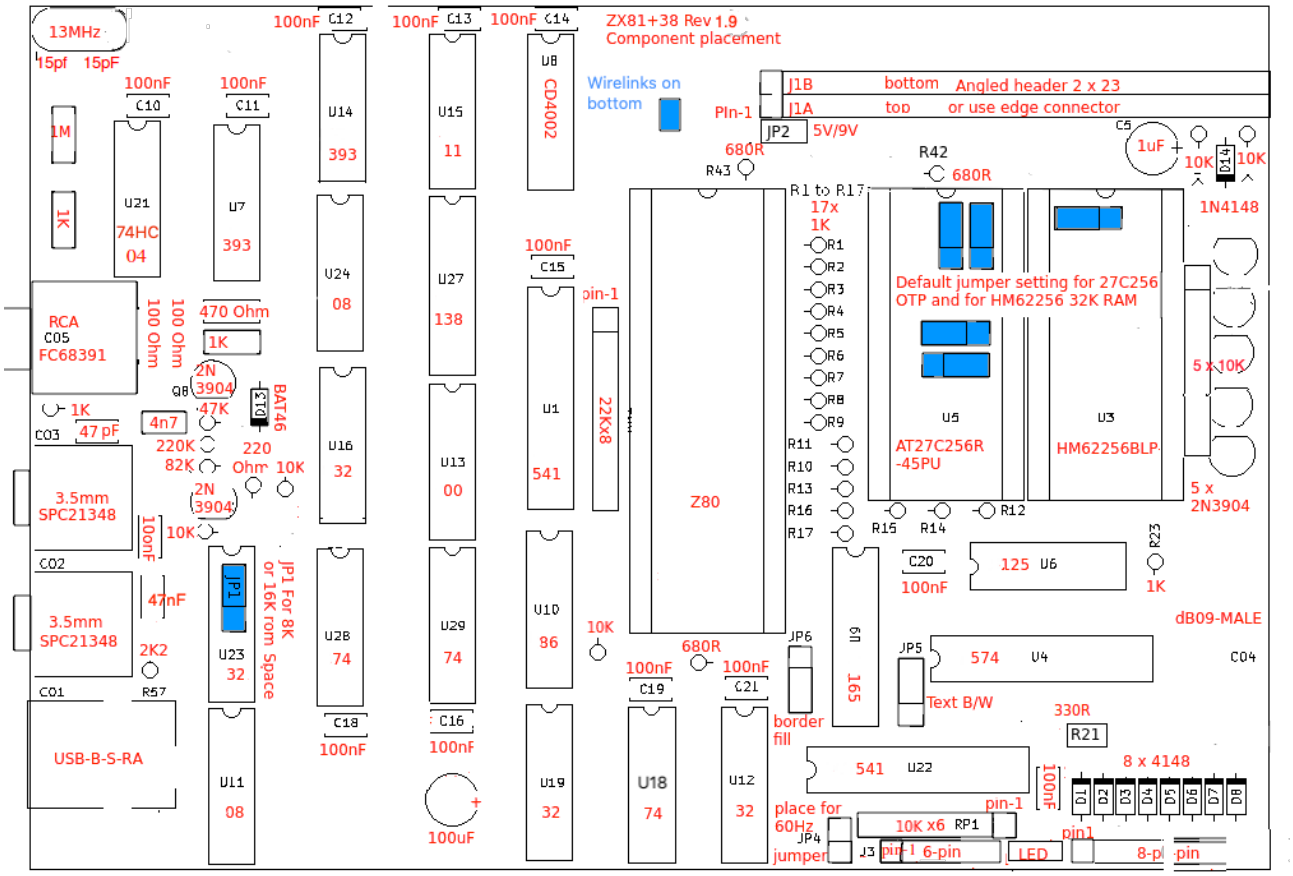
And obviously you need experience like knowing resistor color codes etc etc, and owning a multimeter and know how to use it.

All files you need are on my GitHub page here: https://github.com/mahjongg2/ZX81plus38

You will need the Bill of materials file ZX81plus38_BOM_revision 1.9.ods,

.ods files are libreoffice spreadsheet files. You can open them with the free libreoffice software calc function. Or with an online .osd viewer. Such as this one https://odfviewer.nsspot.net/

you also need a printout of the ZX81+38_rev_1.9_component_placement.png see below

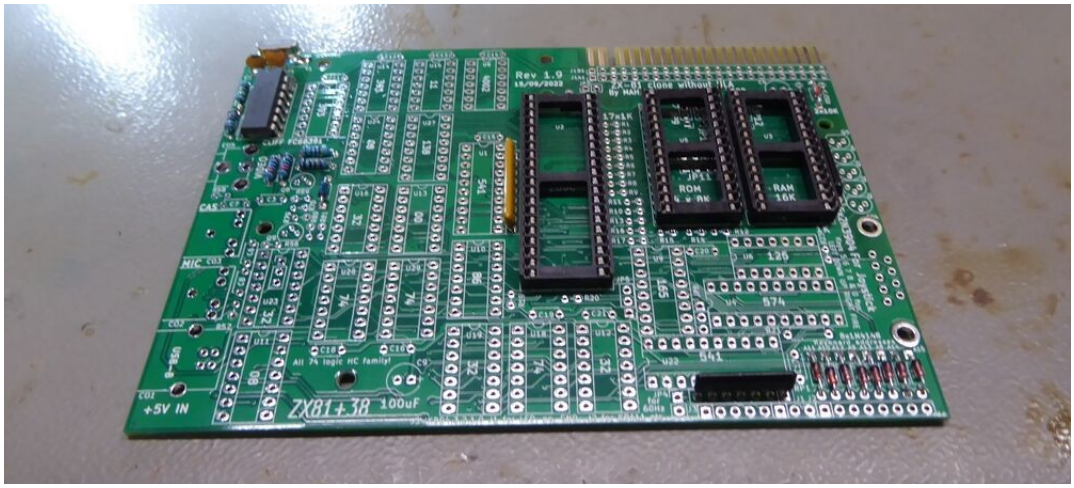


Start soldering the lowest parts

Make sure you use the latest files for the PCB, BOM and component overview drawing, and schematic, at this moment these are revision 1.9.

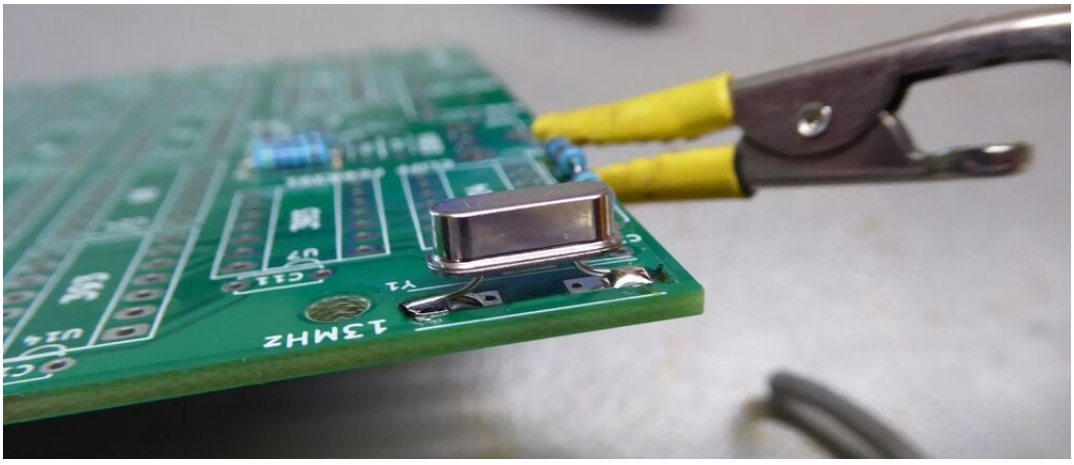
Start by soldering the lowest parts, that is, the flat horizontal resistors, and the resistor packs, and the diodes (the nine 1N4148 and the single BAT46 shottky diode), note their orientation! Also now is the time to place the three main sockets. see this picture: Note that the notch in the socket should point away from you, (toward the back of the PCB) see the component placement file.

Afterward it should look like this (minus U21 and the crystal and 15pF caps).



Place the 13MHz crystal

The PCB is designed for a surface mount crystal, but if you could not find a SMT type of this crystal, (the kind that comes with a plastic carrier and flat pins sticking out besides the metal can) then convert your pin though hole crystal to a SMT solderable one by sticking the pin wires through a piece of paper, fold the pins flat sideways, and cut the wires off just outside the paper isolation. then solder the pins to the solder pads beneath. If you are carefull not to cause a short between the pins and metal can you might leave out the paper isolation, see below:



Now continue with the IC’s

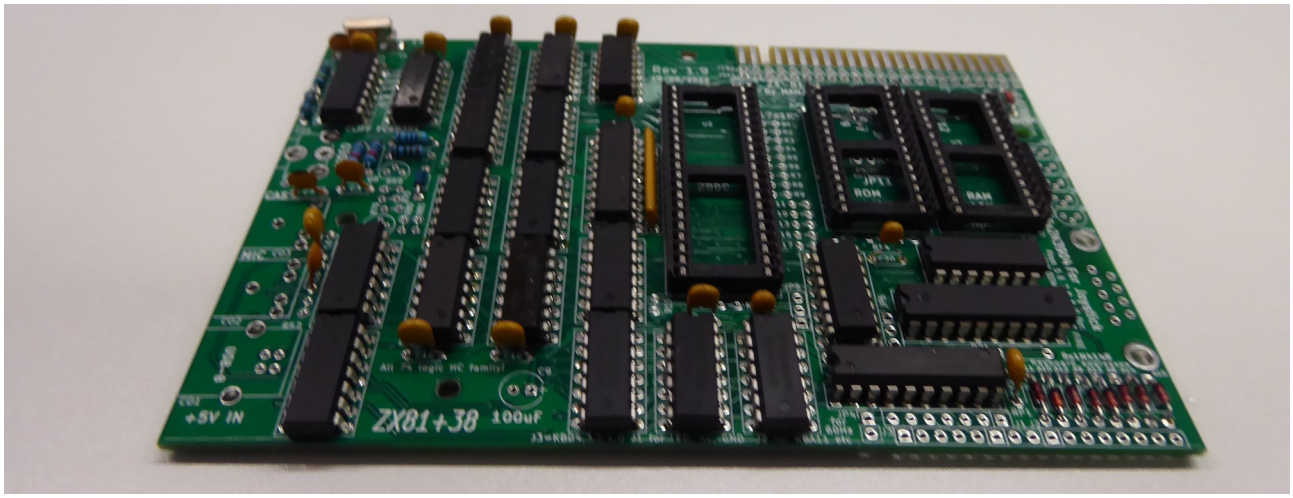
Now add the IC's. Be careful to place them in the right orientation, with their notches either up, or to the left, and in the right spot, reference the component buildup drawing. Use the latest version of revision 1.9 of the component overview drawing, that is the drawing where the 15pF capacitors are BELOW the crystal. And use the latest BOM to order components, specifically make sure you buy the right connectors, as sometimes there are components that are just like them, but still different.

Note that I'm not using sockets for the other IC's, but you can if you want, if you do, use dual wipe sockets, NOT round machined sockets, as those are a disaster to put IC's into, and they are not as reliable as they seem to be. sockets that contact the IC pins from both the inside and outside are more reliable, cheaper, and easier to use.

If you see a number like "08" in the middele of in IC in the component placement drawing this means you need the HC08 version of the IC, so "08" means you need to place a 74HC08 there. All logic IC's are HC type with a possible exception of the CD4002 which you can buy as 74HC4002 or as CD4002. And obviously the Z80 and the ROM and RAM chips.

Now place the small ceramic capacitors.

Now start with the small ceramic capacitors, use the component overview drawing and a highlighter marking which capacitor you have done. Start with all the special capacitors (all non 100nF caps) like the 15pF caps, and all the others, then add the 100nF decoupling caps. as in this picture:



Add the two electrolytic capacitors and the vertical resistors

there are only two elco's in this design, a 100uF bulk capacitor, and a 1uF for reset, solder them in and notice their polarity. There are lots of upright resistors, especially the seventeen 1K resistors to the right of the Z80, but also others, note on the component drawing where they are and which value they need to have and their orientation (chosen to make shorting two resistors together unlikely). use a highlighter to keep track of which you did. After sticking the wires through bent them out slightly so they won't fall out easily.

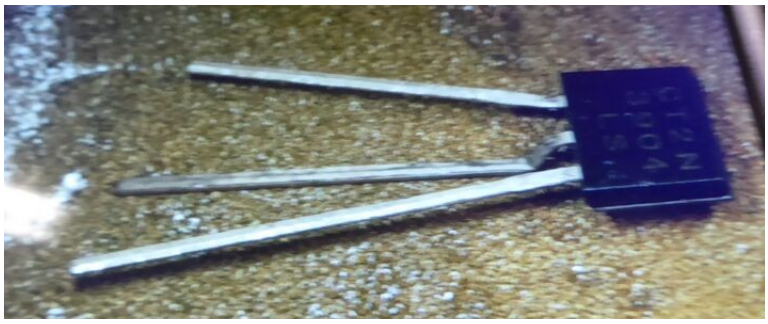
Add the connectors

After that you can start with the connectors, For the Cinch, (RCA composite video out plug) i recommend the FC68391, do not use the reichelt CBP RCA connector unless you have to, as its a very bad quality component. The fit is so tight that you have to wrangle the plug into it which promptly breaks the inner contact. Not recommended, unfortunately Reichelt doesn't seem to sell any other suitable (pin compatible) connector.

Then solder the two 3.5mm audio jacks, models without in-built switches, make sure to buy the correct types with just three pins, there are others with completely different pinout, so be careful what you buy. lastly solder in the USB type-B (square) socket, I know its old and ugly, but it is the only suitable (USB) socket that is pin through hole.

Add the seven 2N3904 Transistors

if you bought the versions that have all three pins in a single line you might want to prepare the transistors, as otherwise it almost impossible to enter then in the PCB, so bend the middle pin towards the rounded part of the plastic, and then bend the wire up again a few mm (1/10 of an inch) so the pins are now in a triangle. The middle pin is now a few mm shorter than the other two, this helps inserting the pins, but it will be even easier when you also cut a few mm of from one of the other two pins, so all 3 pins have different lengths, which makes interring them easy. like this:



If you want you can add the joystick connector

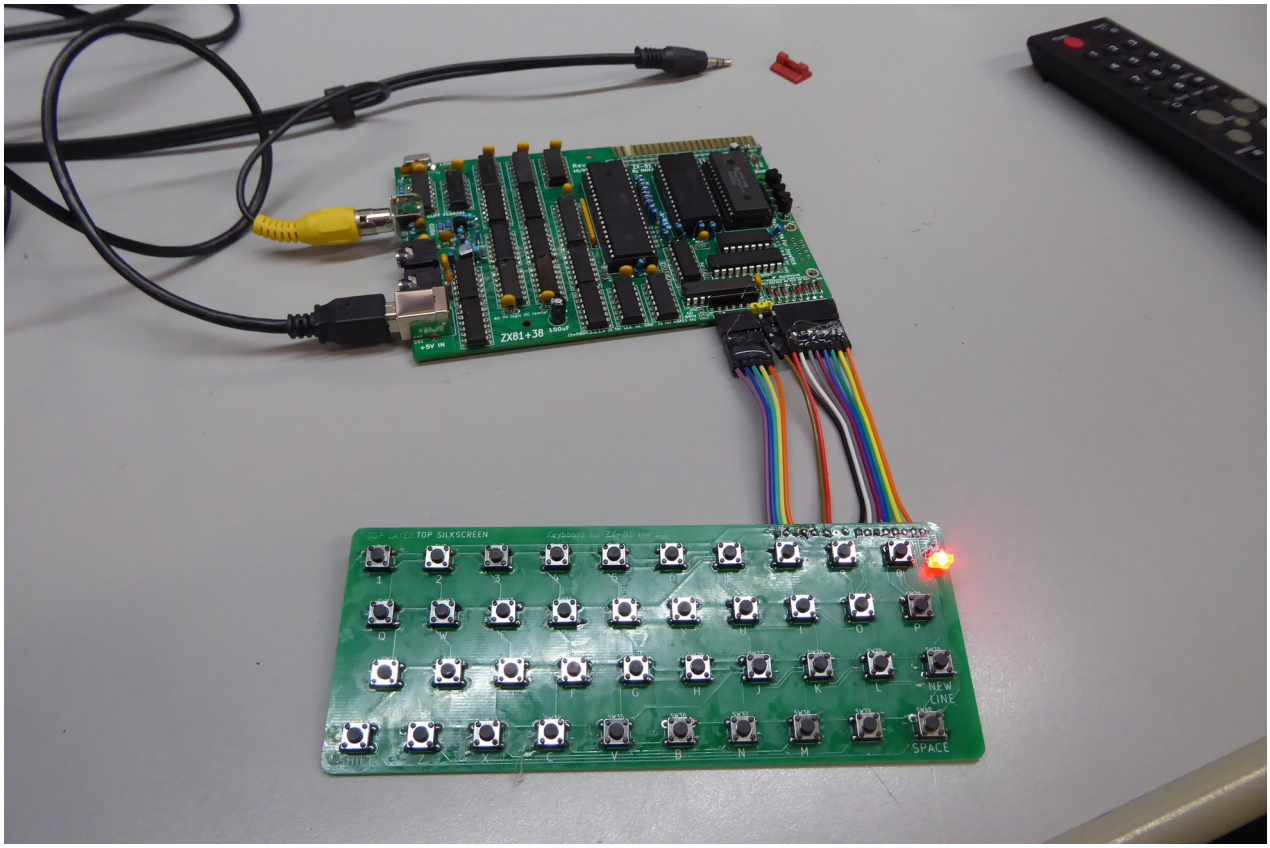
You can add the male DE09 connector (often erroneously called a DB9 connector). But you may wait if you plan to use the original ZX81 enclosure, as you have to hack an opening in the case and you might not want to do this yet.

Finish by adding the socketed IC’s

Now insert the Z80 CPU, the programmed ROM, and the 32K RAM chip, make sure the notch in the IC point to the back, same as shown in the component placement drawing. Note that ROM's especially don't like placing the in reverse! you will destroy them if you power up with them "upside down".

Finish by adding the keyboard

In this example I’m using my own keyboard PCB, also on the GitHub page, with 40 6x6mm buttons (see keyboard BOM). I connected the keyboard to the PCB using dupont wires, with male sockets (that is those with pins) on the connector side, the other side is directly soldered to the keyboard PCB, with the wires pushed through the back of the keyboard PCB and soldered on top. On the ZX-81+38 PCB use angled female headers, as these are rare, simply take the kind of female headers as used on an arduino, cut off one piece with 5 pins, and another one with eight pins, and bend the pins 90 degrees. for the three pin segment in the middle use a 3-pin male angled pinheader, and cut off the middle (unused) pin. for those two wires that connect to the 3-pin use two female dupont wires. Cut the dupont wires all to the same length (about ten centimeters) strip them, twist the wire strands together and tin them. Stick the wires though the keyboard PCB, and soldered them on top (it helps if you cut off the end bit of the tinned wire, and fold the wire flat to the top of the PCB, so the wire stays in place for soldering) look out not to create shorts. after soldering the 5+2+8 wires use a small bit of hot glue on the underside of the board. If using the real ZX-81 top case with the keyboard foil removed, you have to widen the slot underneath the keyboard so the wires can go though, with the keyboard flat on top of the top case Before you assemble the PCB and keyboard in the plastic enclosure, test the zx81+38 without an enclosure. Like this:



If all is well and you have chosen composite video (or AV) in on your monitor, if you power the board on, the power LED should light up immediately, the screen will turn white, and after a a second or so the inverse K prompt will appear. You can then mount your ZX81+38 in your enclosure, and start using it.

If not you might have made a (soldering) mistake, and to get your ZX81+38 working you need to find the foult. I will try to help in the next section:

Finding a fault in the ZX81+38

It must be a disappointment to find your ZX81+38 is not working, as you know that the design works well, and you are reasonably good in soldering, and have triple checked all solder-joints, and re-soldered any and all suspect solder joints (it helps to hold the PCB to a light if you can see light shining through an un-soldered joint).

Unfortunately the ZX-81 is NOT an easy device to debug, as it uses a very complex system to generate video, and if you don know how it works it might be almost impossible to use the normal fault finding techniques. Fortunately for most builders it simply workes "first time".

I'm describing a worst case scenario, simply a "black screen" no video at all.

So how do we fix such a fault?

The simplest reason for the problem could be that you are using the RCA connector from Reichelt, which is a piece of sh*t. made out of scrap metal and recycled plastic. I once more suggest buying the much better version sold by farnell. It is almost impossible to plug in the RCA plug into the Reichelt connector and as you try to wriggle it in (turning the plug while trying to push it in) you will immediately break the metal that connects the PCB solder-point of the signal to the contact you are trying to plug into. So no video comes out of that connector!

If this isn the problem you need to do the standard fault seeking .

I will explain all the standard stuff.

Start by checking the voltages on the chips, on the top right pin of all the logic IC's (next to the notch) you should find 5V ± 5% (0.25V) . for GND I recommend the metal of the RCA connector.

Check if the nRESET signal on the Z80 (pin 26) is high as it should be, Also check pin 25 (nBUSRQ) if it is not high, you might have forgotten to place R18, 10K near pin 20 of the Z80.

Test the clock signal,with an oscilloscope you should see a 3.25MHz on the CPU pin 6, if not start measuring at the crystal oscillator. Pin 2 of U21 should have 13MHz, then measure pin 4 of U7 where it should be 3.25MHz. Also measure at U18 pin 1 and 6.

If the CPU has a clock signal, continue by checking the databus signals and adddress signals . If there is a problem there remove the Z80, the ROM and RAM chips and look if there is a bent pin underneath any of them that might be the problem.

If you doubt the ROM content check the content with your eprom programmer. If you used a W27C512 then that is the problem! As the default jumpering is for a 27C256 not for a W27C512!

To check if the ROM is running correctly, (the code runs) you can check if the CPU NOP enforcer U1 gets nNOP pulses on pin 1, if it does you know for sure this is not a software (not running) problem, as only a correctly running software will generete nNOP pulses!

If the software runs you can try to look at the video generator, start with the Pixel Shift Register, U9, and look if a pixel stream comes out of pin 9 and (inverted) on pin 9. If not see if if a shift/load signal is available on pin 1 of U9. If not look at the logic around U28 and U29.

If a pixel stream comes out of U9, you can follow it through U10 pin 9 U18 pin 9, U10 pin 6 and U11 pin 3. A problem can also occur in the video combiner Q8, for example if R49 is absent, the output signal is always black (or sync) resulting in a black screen.

Keyboard problems

Keyboard problems often occur in keys with the same row or column. For example if both the 0 and P (“print”) do not work it is probably the KBD0 line. So measure if the connection of KBD0 from the keyboard to the ZX81+38 PCB is present.

It is also possible that the problem is that one of the keys doesn’t have all it’s four pins soldered, as the 6x6 buttons have pins connected, and if one of these pins isn soldered the matrix might be broken, so check if all pins are soldered correctly.