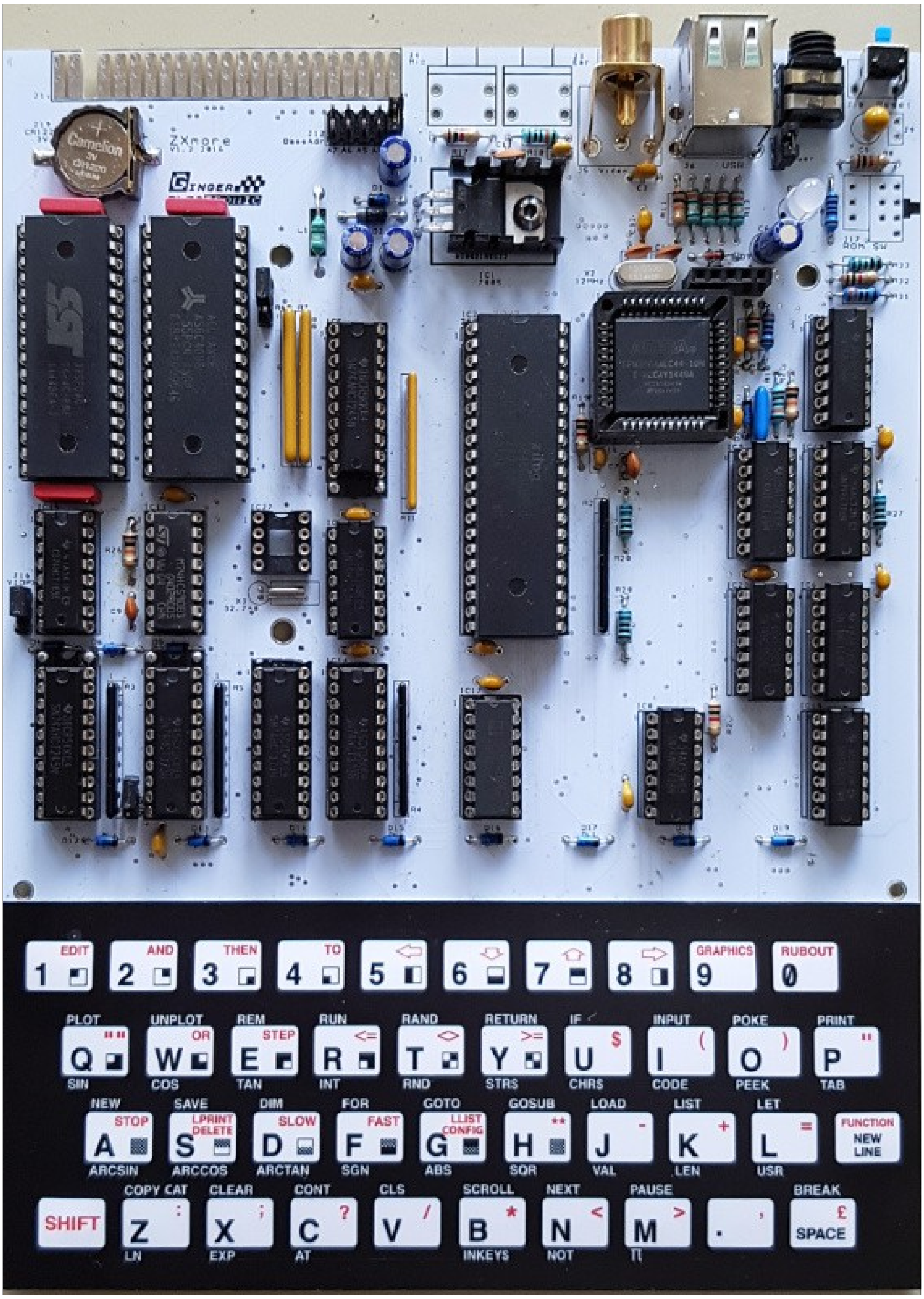
ZXmore V2

notes for the construction kit



Designed by ginger-electronic.com

Contents

Short description:

The ZXmore is an 8 bit computer which is compatible to many systems based on the Z80 processor, especially to Sinclair's systems ZX80, ZX81 and ZX Spectrum but compatible to CP/M developed by Digital Research (*) as well. ZXmore has an integrated keyboard, a video connector for a monitor or TV with composite video and a USB interface for loading and storing programs.

A separate serial terminal may be connected via an USB/RS232 adapter but CP/M may be used with the integrated keyboard and connected video monitor / TV as well (restricted to 40 chars per line). USB may be used with a flash medium for mass storage. You will need a separate power supply with 9V/500mA DC while the actual power consumption is about 150mA only.

ZXmore offers 512k flash ROM and 512k RAM, divided into 8 separate instances while the first instance is used for control the other instances and handling with additional peripheral hardware (USB for example). The other 7 instances may be configured individually and may contain different operating systems (or system rom's) either parallel and switched with the keyboard or concurrently on request (multi tasking system).
You may find more about this and other features in the ZXmore user manual.

(*) The current available release 2.0 of ZXmore does support only system rom's for ZX81 and ZX80 and loading and saving programs via the USB flash medium. Future release will support ZX Spectrum rom with monochrome display or even CP/M. The used roms for ZX80 and ZX81 are covered by the GNU General Public License v2 and may be used free of charge. You will find more notes in the ZXmore user manual.

Before starting:

check components :

You will need all in the appendix listed parts and components to construct the ZXmore. The construction kit may be equipped without keyboard or with some special „hard to get" parts only. So please check the completeness before starting.

tools/equipment:

You will need the following tools and equipment:

- temperature controlled soldering station with small copper bit
- side cutter
- tweezer
- thin soldering tin
- optional – but helpful – is a magnifying lamp or at least a magnifying glass

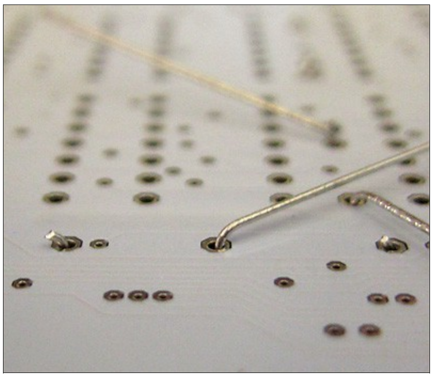
Soldering is required in 1/10 inch pitch only which isn't too hard even for beginners. But this shouldn't be your very first soldering project as this is a kind of computer with about 800 soldering joints which requires quite precise work. You may consider some assistance from a more experienced friend if needed.

The time you need for completion of this construction kit is about 3 hours depending on your skills.

Order of construction:

The easiest way to complete a construction kit is to start with the components in order of their heigt – the smaller parts first. This way it is easy to lay the board straight on the table. It would be more easy to start with all equal value components first, step by step.

1st step - resistors:

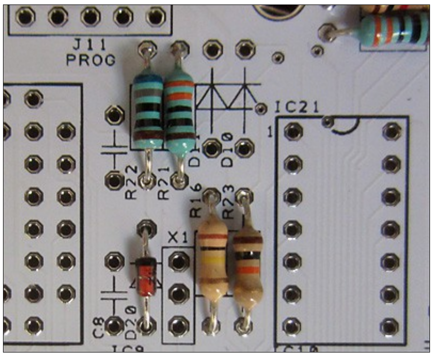


It is easy to find out all item positions using the imprint of all component names. Resistors use a rectangle symbol with two lines as wires, indicating the holes to use. This is quite a game similar to finding the doors on an advent calendar but this way you get more familiar with the board also.
All positions for all components are printed on the board.

After every turn the soldering points should be controlled for maybe forgotten solder joints. This could be done while holding it against the light to see those candidates or you may count the solder points during soldering (15 resistors require 30 solder points). This way you are sure not to forget some solder point.

The resistor arrays R1-R7 are fitted later and should be kept separate.

2nd step – diodes:

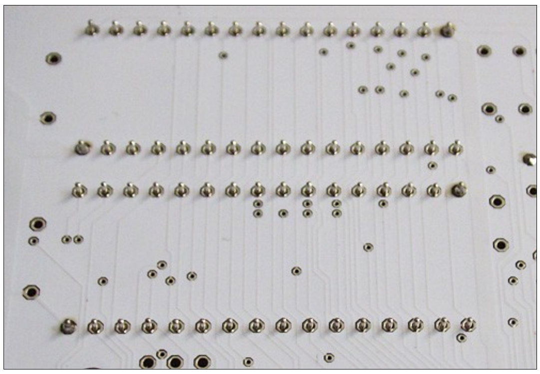


Diodes may be recognized as a triangle symbol with a bar like shown in the picture.
Important: All diodes have to be soldered in the right direction to work. Every diode has a bar printed on the case and this has to correspond with the bar on the board resp. the triangle has to point to that bar mark.

Diodes are often too small to print the value/type on it. You may orient also on the quantity when taking it out of a bag or the value/type printed on the bag. When finished all solder joints should be checked as well carefully.

The 3-colour RGB led should be soldered at the end (step 9).

3rd step – IC sockets:



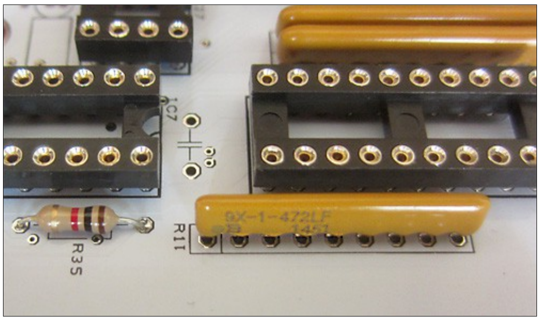
IC sockets are processed the same way and should be sorted by size and soldered from the biggest to the smallest size. This way you can avoid soldering a 14 pin socket in a 16 pin place on the board accidently. Sockets are aligned by soldering two diagonal edges first and be sure to put them flat on the board.

Take attention about your fingers when pressing them on the board while soldering. After fixing one size of sockets in the edges you should solder row after row and pin after pin to avoid forgotten solder joints here.

The pins should be soldered from the opposite side to avoid releasing the wire accidently when holding the copper bit to near. Also keep note not to damage the isolation when soldering the capacitor in front of the IC later and hold the wire back with a tweezer.
The square PLCC socket should be soldered last as it is higher than the other sockets and the board won't lay straight anymore when soldered.

4th step – resistor arrays:

After mounting all sockets you can place the resistor arrays on the board. Important is the direction while the arrays are coded. The two 2.2k arrays R6 and R7 can be mounted in either direction while they contain 5 single resistors with both ends wired (10 pins).

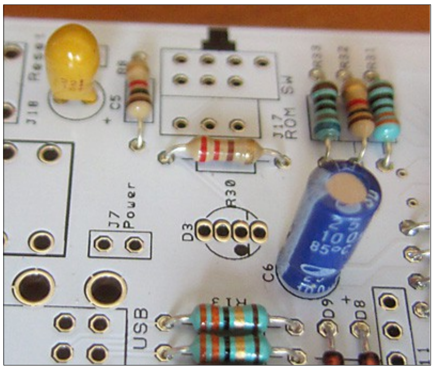


The other arrays have to be placed with pin 1 marked with a star on the component while pin 1 on the board is marked with a optically separated pin.
On the picture you can see the 4.7k array R1 which shouldn't be confused with R11 while pin 1 is marked with a 1 too, very near to R1 which may be accidently read as R11.

Be sure to place all arrays in the right direction which isn't same for all !

5th step – capacitors:

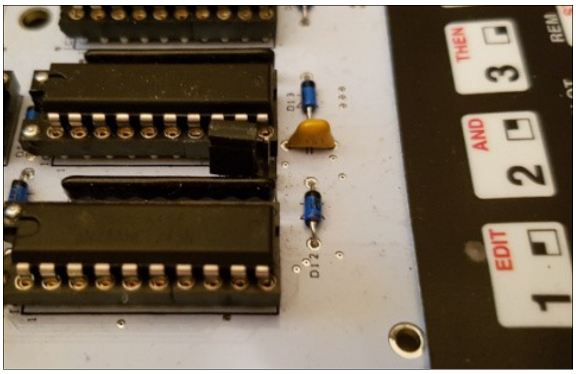
The capacitors are used with different pitch with 1/10, 2/10 or 3/10 inches. If you don't have the corresponding pitch you may bend the wires carefully to match the required pitch as well. C1,C2,C6,C31 (100uF) and C5 (22uF) are processed later because the polarity has to be regarded as well as all 100nF capacitors should be spreaded over the board as power supply stabilization only.



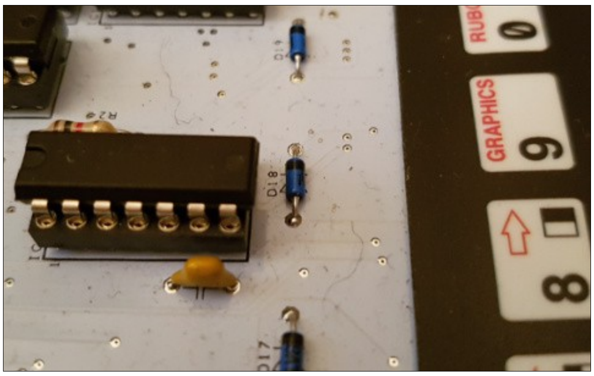
All other (named) capacitors are placed first, then the bigger 100nF with 3/10 inches pitch in front of IC3, IC4, IC15.
If you don't have matching pitches you may bend the wires carefully. The other 12 ceramic capacitors for stabilization of the power supply are placed in front of the IC's marked with a symbol with two parallel lines and 2 wire indicating the drill holes to be used. The names are missing here on the board due to space.

All electrolytic capacitors and the tantalum are soldered last and always with the marked + on the capacitor matching the drill hole with a plus mark. The electrolytic caps have mostly the negative wire marked with a big bar or minus – this way the corresponding pin should NOT be matched with the marked plus on the board.

There are two capacitor accidently not named on the board (silk print) and shown in the pictures below (C33 and C34).



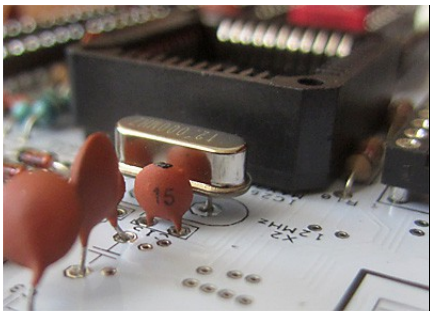
(C34, 150pF)

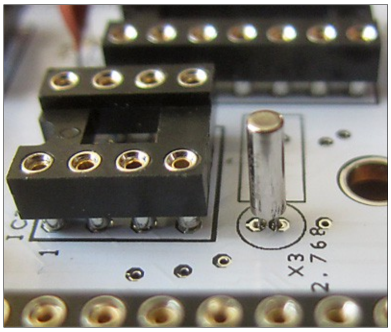


(C33, 470pF)

6th step – crystals & filters:

Finally it's turn for the crystals and filters. The3 pin filter with 6.5 MHz has no polarity and can be simply mounted on it's space marked with X1. The other two crystals have no polarity either and should be mounted with a small distance to the board as shown in the pictures.



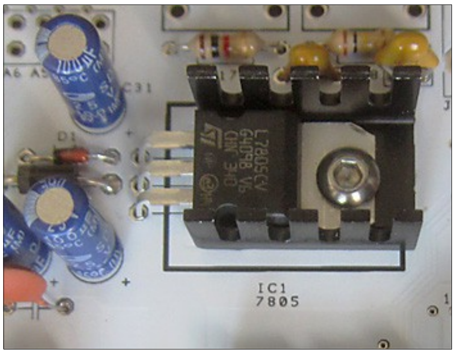


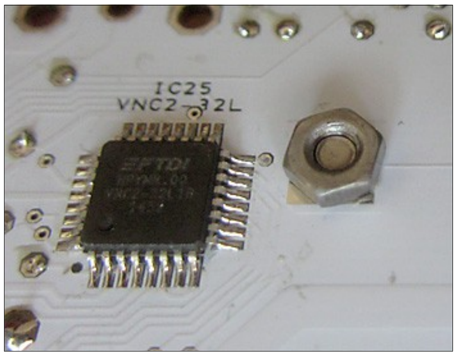
Putting the RTC chip in the 8pin socket will protect the small crystal against mechanical stress.

7th step – voltage regulator:

The following 2 pictures show mounting the voltage regulator IC1 (7805). The regulator is decoupled from the GND potential with two diodes for possible use with a USB power supply.

The regulator should be mounted with a screw first and then solder the contacts to avoid mechanical strength.





8th step – mechanical parts:

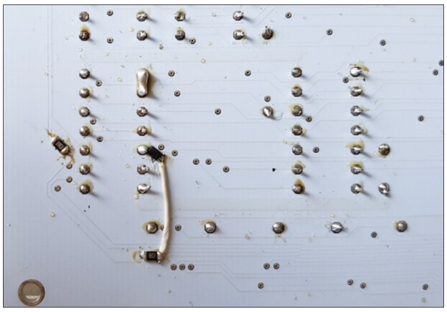
Time to solder the missing connectors, switches and pin headers in it's place. The small contacts should be soldered first and the bigger connecting the case at last with increased soldering temperature. The outer cases and bigger copper areas will take much of the heat from the solder iron and this may take several seconds to let the solder melt nicely.

When soldering the pin headers it might be helpful to place a jumper first to hold it on that point safely with a finger (protected from heat).

9th step – missing parts:

There are some components already added as smd parts to the mainboard. The diode at pin 5 of IC19 (74HCT74) need to be soldered when adding the IC or socket for that IC. Pin 1 and 2 of the same IC have to be connected with a solder bridge or a small wire or cutted leg of a resistor or capacitor. Be sure not to interchange the pins as it is mirrored on the solder side.

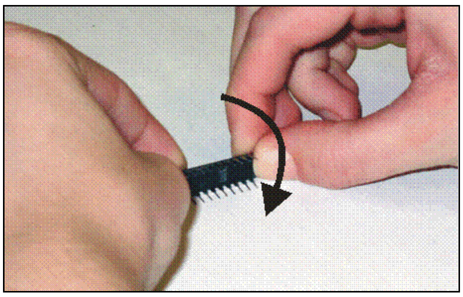
One capacitor (C100, 47pF) has to be added after soldering the RTC chip IC27 (DS1302 with 8 pins) between pin 4 and the via near pin 3 as seen on the picture.



The RGB LED can be soldered finally, too. It should have a distance to the board of about 5mm (0.2 inch) and the longest leg goes into the drill hole marked with the point.

10th step – mounting IC's:

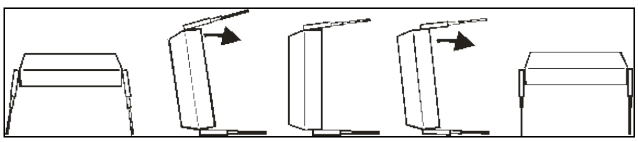
New ICs have pin rows which have to be first right-angled manually prior to putting them into the sockets.



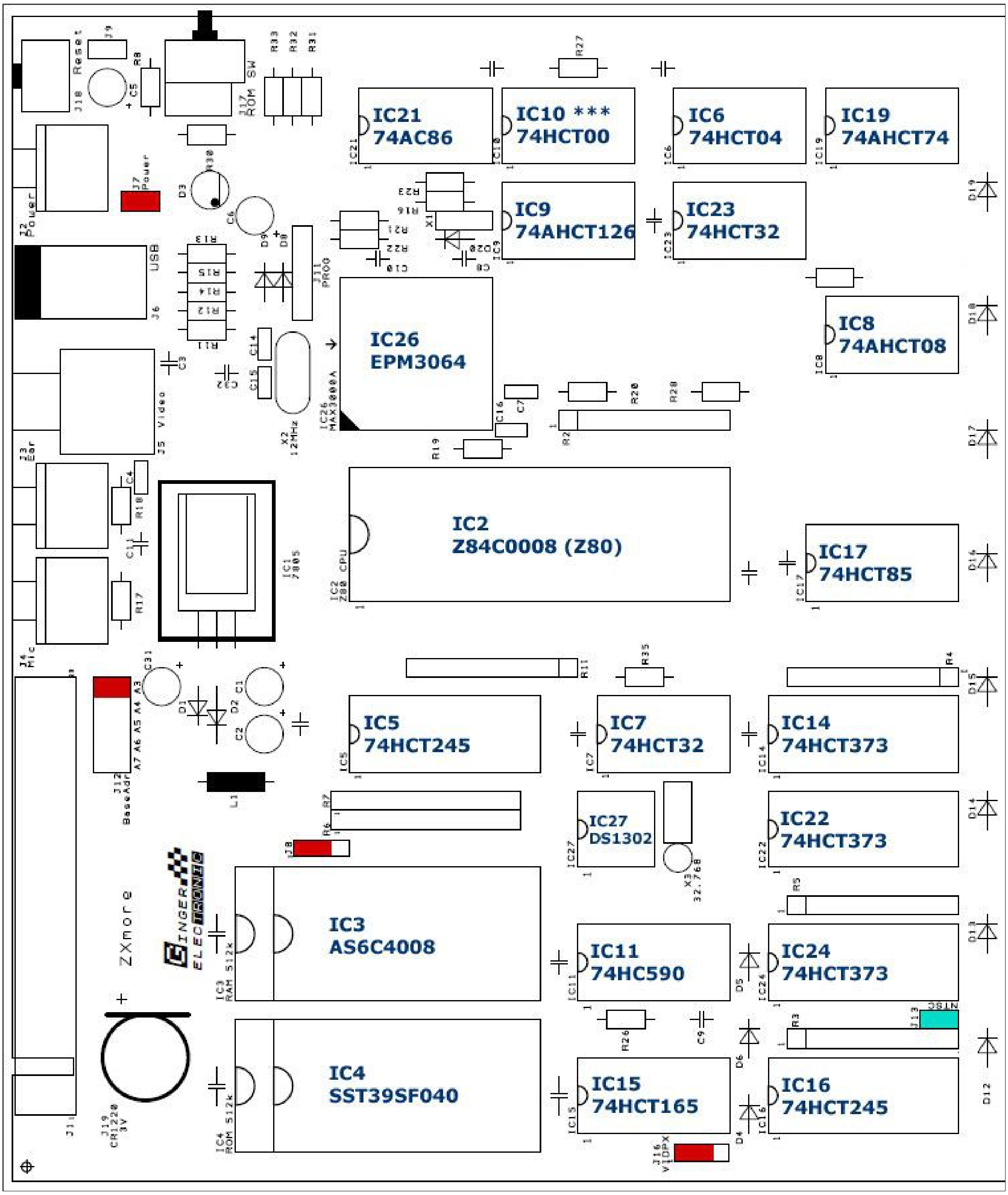
You have to bend all pins right-angle like shown in the picture. You take the body between right and left hand with your fingers, let one line of pins lay on the table and then turn the body a little bit to bend all pins slowly all together with not too much power in the right angle direction (90 degrees from body).

This must be done in a careful and sensitive way and it is better to repeat this step 1 or 2 times than to bend them too much in the first step. Bending should be done alternately from both sides. This is just a matter of excercise and will be easier with the smaller ICs.

The best way is to start with the small ICs and after proceed with the bigger ones. So you get a little bit routined and it is not too annoying to damage a cheap TTL IC which can be replaced inexpensive in case of damage.



You will find an overview where to place which IC in the following drawing:



In the picture the red marks show jumpers to be set and which position while the cyan/blue marked jumper is optional for setting up ZXmore for NTSC video (default PAL when not set).

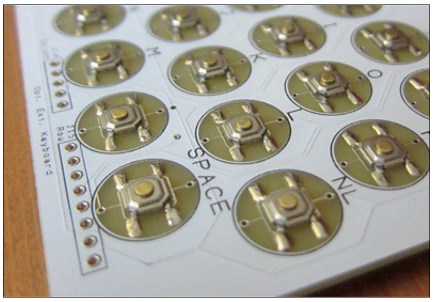
All IC's have to be placed in the same direction with the „nose" or pin 1 (marked with a star or point) in direction of the connectors side of the board.

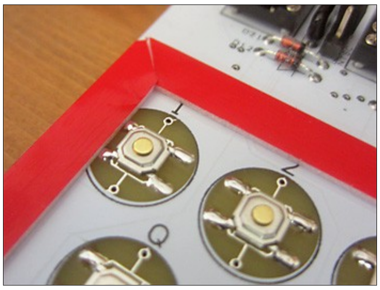
11th step - keyboard:

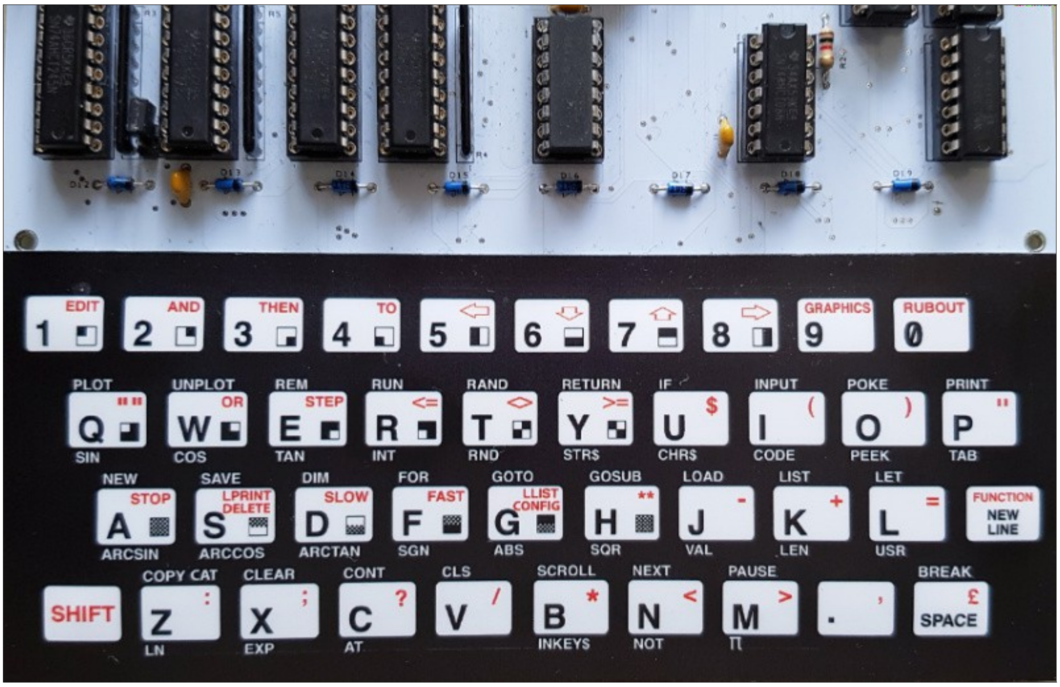
There are two variants of mounting a keyboard.

The original keyboard membrane from Sinclair is self-adhesive and has metallized holes under each key to connect the wire tracks under it when pressing keys. The small holes in the edges may help to put the keyboard membrane right in place. Especially the bottom holes should match exactly to the board holes if you want mount the board into a case.
A new keyboard membrane may have to be used quite long time to get easier to work.

The second variant uses smd tactile switches for a comfortable use of the keyboard with less force. The keys are soldered first in their place and then the small doublesided adhesive tape will be put around the keys as shown. Be sure to match the outer lines of the membrane which is put on top of the tape after. The tape matches the additional height of the switches.







Good time for a short break to keep your concentration !

Putting into operation

First you should get familiar with all connectors of the system and jumpers to be set.

Purpose of connectors and jumpers:

J1 slot edge connector for additional hardware mounted to the ZXmore
like printers, mass storage devices like ZXpand and interfaces available for ZX81

J2 power connector 3.5mm audio jack for power supply of 9V to 12V DC
polarity: tip for plus and ring for minus

J3 - optional - EAR, 3.5mm audio jack for LOAD (load programs)

J4 - optional - MIC, 3.5mm audio jack for SAVE (save programs)

J5 Video, chinch jack, composite video output (monochrome)

J6 USB, 2x USB host port for mass storage & peripherals

J7 power jumper, may be used for a power switch, must be set on operation

J8 RAM size, jumper has to be set to board connectors direction

J9 - optional - reset, to be used for an external reset switch when mounting into a case

J10 not used

J11 - optional - used for programming the USB controller, for development purposes only

J12 I/O address used for latches, default set to A3 (see user manual)

J13 select video mode – PAL for not set, NTSC for set

J14/J15 - optional - connectors for external keyboard (see user manual)

J16 video polarity, set to connectors direction for normal picture,
vice-versa for inverted picture (white chars on black background)

*J17 - optional - rom switch, for mechanical switching of instances, should be kept to „0“
(for details see user manual)*

J18 onboard reset switch for soft reset (see user manual)

J19 battery holder for CR1220 battery, used for the RTC chip (clock)

optical inspection:

After setting all important jumpers (J7, J8, J12 and J16) you should inspect the board carefully.

Is there any missing part or any marked free area on the component side of the board ? (J9 and J11 are not used and may be kept free.)

Much more important is the optical inspection of the solder side of the board.
A missing connection may have deep impact on operation and require a sophisticated troubleshooting. So please inspect all solder joints carefully, row to row with a magnifying lamp or a magnifying glass. The more time you spend on this the more chance of a quick start you will have.

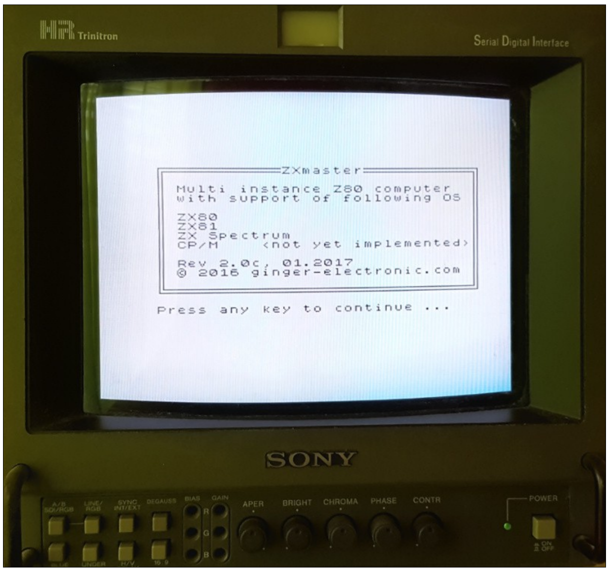
You should keep an eye on clean, smooth and shiny solder joints and accidently shorts made. Cold solder joints can be found when looking for tarnished surface and uneven or spiky surface. They should be fixed while soldering again with some very rare (!) fresh solder tin. Here is the contained flux needed. If you have much to much solder tin you may remove all with desoldering wick and repeat solder.

Are all additional components added as shown in step 9 ?

If you own a multimeter you should measure the resistance between both outer pins of the voltage regulator IC1 (7805) – it should be greater than 10k. The resistance between pin 2 and pin 3 should be minimum a few hundred ohms. Shorts may be problematical here as 9V or 12V of power voltage may damage the integrated circuits.

The power supply can be connected now while regarding the correct polarity. The tip of the 3.5mm audio plug has to be plus and the ring is minus. The voltage may be 9V or 12V DC.

In the next step the output voltage of the regulator should be checked after power-up while measuring at any 14 pin IC with pin 7 minus and pin14 plus. The voltage should be between 4.8 and 5.1V. If no USB device is connected, the typical power consumption is 80-100 mA, can be measured when removing the power jumper J7 with a multimeter.



Now the monitor or TV may be connected via the chinch plug and when all was done right the starting screen should appear as seen in the picture.

You will find additional information for operation in the user manual.

Appendix 1 – parts list of ZXmore:

resistors:

4 x	33R	R12,R13,R14,R15
1 x	100R	R30
2 x	330R	R21,R31
4 x	680R	R20,R27,R28,R33
5 x	1k	R8,R17,R19,R22,R32
1 x	3k3	R24
1 x	6k8	R26
2 x	10k	R11,R23
2 x	100k	R16,R18

2 x Array, 2k2, 5x parallel (10 pin), R6,R7
1 x Array, 4k7, 8x star (9 pin), R1
4 x Array, 10k, 8x star (9 pin), R2,R3,R4,R5

capacitors:

2 x	18pF, RM 2.5	C14,C15 (alternative 15pF)
1 x	33pF, RM 2.5	C7
2 x	47pF, RM 2.5	C16,C100
1 x	150pF, RM 5	C34
1 x	470pF, RM 2.5	C4
1 x	470pF, RM 5	C33
1 x	1nF, RM 5	C8
2 x	47nF, RM 5	C9,C11
12 x	100nF, RM 5	C3,C10,C17,C18,C19,C21,C24,C25,C26,C29,C30,C32
3 x	100nF, RM 7.5	C22,C27,C28

1 x 22uF/16V, RM 2.5, tantal C5
4 x 100uF/16V, 6.3mm, C1,C2,C6,C31

integrated circuits (ICs):

1 x 7805	IC1 (voltage regulator TO220, alt. LMS78_05-1.0, RECOM R-785.0-1.0)
1 x Z84C0008	IC2 (Zilog Z80 CPU, 8 Mhz, DIP40)
1 x AS6C4008	IC3 (SRAM 512 kByte, DIP32)
1 x SST39SF040	IC4 (Flash ROM, 512k Byte, DIP32)
1 x 74AC86	IC21 (DIP14, only advanced CMOS, no ACT, no HCT, no HC)
1 x 74HC590	IC11 (DIP16)
1 x 74HCT00	IC10 (DIP14)
1 x 74HCT04	IC6 (DIP14)
1 x 74HCT08	IC8 (DIP14)
2 x 74HCT32	IC7,IC23 (DIP14)
1 x 74HCT74	IC19 (DIP14)
1 x 74HCT85	IC17 (DIP16)
1 x 74HCT126	IC9 (DIP14)
1 x 74HCT165	IC15 (DIP16)
2 x 74HCT245	IC5,IC16 (DIP20)
3 x 74HCT373	IC14,IC22,IC24 (DIP20)
1 x VNC2-32L1B	IC25 (LQFP32)

1 x EPM3064ALC44-10N, IC26 (PLCC44)
1 x DS1302 IC27 (DIP8)

diodes:

13 x BAT46 D1,D4,D5,D6,D12,D13,D14,D15,D16,D17,D18,D19,D20
2 x 1N4148 D8,D9
1 x 1N5817 D2
1 x LED RGB D3, common anode

connectors and jumpers:

1 x CL1384 J2 (3.5mm audio jack mono, manufacturer CLIFF)
1 x S8411-45R J19 (CR1220 battery holder, SMD, manufacturer Harwin)
1 x Chinch J5 (female, right-angled, pcb version, RM 11mm)
1 x USB plug J6 (USB plug female, 2-port, right-angled, pcb version)
1 x MJTP1105 J18 (Reset switch, right-angled, pcb version, manufacturer APEM)
5 x Jumper for 2-pin male pinheads
2 x pinhead male 1x2, J7,J13
2 x pinhead male 1x3, J8,J16
1 x pinhead male 2x5, J12^

optional:
1 x pin head male 1x2 J9 for external reset
1 x pin head female 1x5 right-angled, J11,J14 (for external keyboard, program USB)
1 x pin head female 1x8 right-angled, J15 (for external keyboard)
2 x CL1384 J3,J4 (for audio load and save, 3.5mm audio jack mono, manufacturer CLIFF)
1 x KDR16H J17 (ROM switch, hex coded switch, right-angled, manufacturer OTAX)

J1 is the slot edge connector directly on the pcb and is not to be mounted. :-)

miscellanous:

1 x DIP8 IC socket, precision socket with gold plated contacts
8 x DIP14 IC socket, precision socket with gold plated contacts
3 x DIP16 IC socket, precision socket with gold plated contacts
5 x DIP20 IC socket, precision socket with gold plated contacts
2 x DIP32 IC socket, precision socket with gold plated contacts
1 x DIP40 IC socket, precision socket with gold plated contacts
1 x PLCC44 IC socket PLCC
1 x 12 MHz X2, crystal, HC49U, 12.000 MHz
1 x 32.768kHz X3, crystel, 2.1 x 6 mm, 6pF, clock crystal
1 x 6.5 MHz X1, SFE 6.5, ZF filter 6.5 MHz, (Murata)
1 x 1uH L1, inductance, RM 12.5, alt. other values between 1 und 5 uH
1 x CR1220 battery 3V
1 x heatsink for TO220, max. 23 x 17 mm (length/width) with screw
1 x pcb board Zxmore, 204 x 156 mm, 2 layers
1 x keyboard ZX80 keyboard membrane
40 x ALPS SMD tactile switches Alps, SKQGAFE010, 1 newton force

optional accessories:

1 x power supply, 9V DC, 500mA, with jack 3.5mm
1 x cable, 1-3 meters chinch shielded for monitor/TV connection
1 x USB flash stick as mass storage
1 x USB adapter RS232, CHIPI-X from FTDI, for serial terminal connection with CP/M

Appendix 2 – colour codes for components:

colour code:

The value of most components are not printed in clear text due to shape and size but coded with colours or numbers.

Resistors, coils, rarely some capacitors are coded with coloured rings. Depending on precision 2 or 3 rings are used to indicate the value, the forelast is the multiplier and the last one the tolerance. Every colour represents one digit.

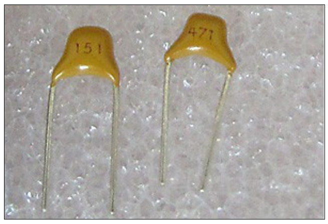
value	multiplier	tolerance
0 = black	x1	
1 = brown	x10	(1%)
2 = red	x100	(2%)
3 = orange	x1000	
4 = yellow	$x10^4$	
5 = green	$x10^5$	
6 = blue	$x10^6$	
7 = violet	$x10^7$	(0,1%)
8 = grey		
9 = white		
* = gold	x0.1	(5%)
* = silver	x0.01	(10%)

Resistors with 5% tolerance have four coloured rings with the first two for the value and the third for the multiplier. For example black is a multiplier of 1 (10^0) and orange represents 1000 (10^3). The value is multiplied with this.

Resistors with 1% tolerance or better own 5 rings, three for the value, the following as mutliplier and the last one for indicating the tolerance. Two resistor examples are shown in the picture above.

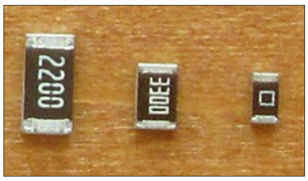
The first has 330 ohms with orange-orange-black and multiplier 1 (black).
The second has 680 ohms with blue-grey-black and multiplier 1 (black).

numerical code:



Capacitors are specified often with a 3 digit numerical code. The first two digits give the value and the last one the multiplier. The value is specified in pF (pico Farad). On the picture you see 150pF ($15*10^1$) and 470pF ($47*10^1$).

1000pF is 1nF – so a 10nF capacitor has an imprint of numerical code 103 = $10*10^3$ = 10.000 pF or a capacitor with 10uF is marked with 106 (10.000.000 pF). Sometimes you may see just 10u if there is enough space to print.



SMD resistors are classified in the same way with the same numerical code. In the picture 2 resistors with 220 ohms and 330 ohms are shown. There are existing „resistors" with zero ohms as well marked only with a simple printed zero.

Unfortunately SMD capacitors have no imprints at all and should be taken from their packing only when directly processed.