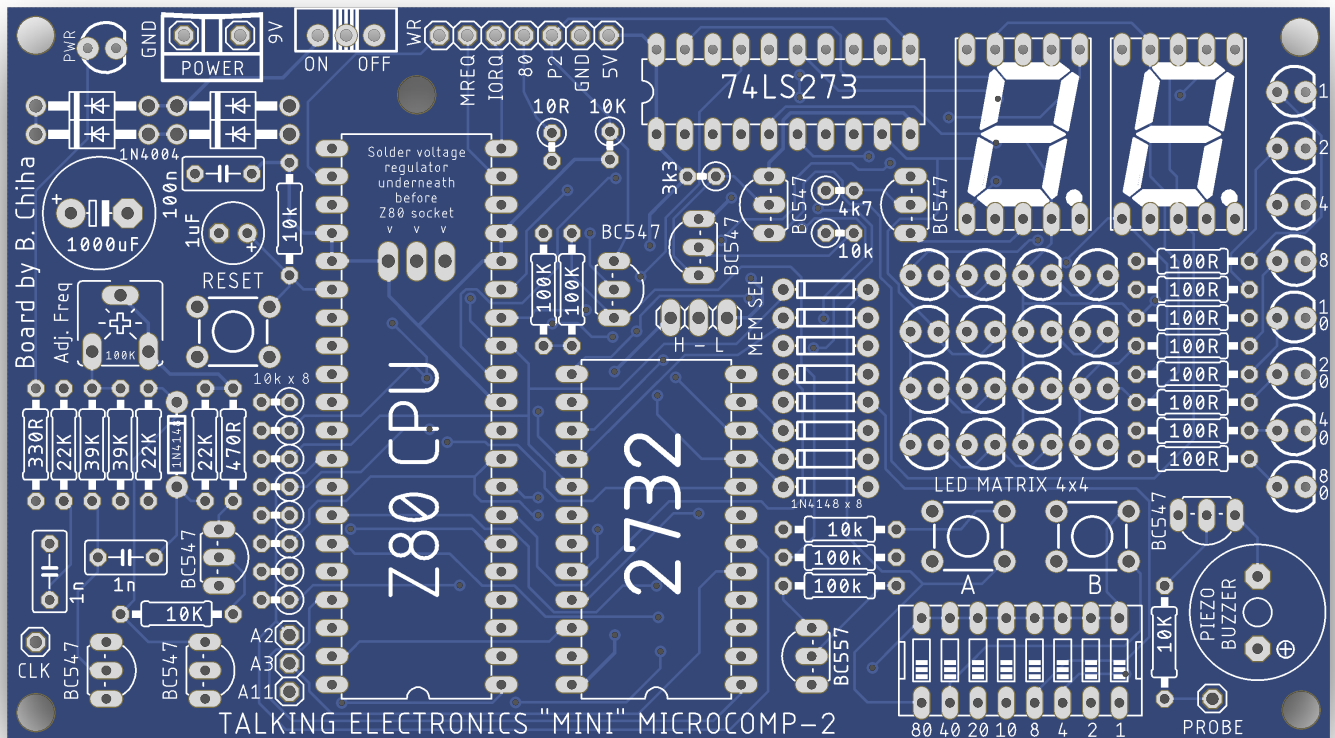


MICROCOMP-2 Project build

Thank you for your interest in building the
Talking Electronics Micro Computer v2.
Project website: <https://github.com/bchiha/microcomp>



Package contents

1 x Microcomp-2 PCB, 1 x 27C32 fully programmed EPROM, 1 x 74LS273 D-Flip Flop latch, Some loose components and build notes.

Construction notes

Use a smaller soldering iron head if possible. When inserting some components be careful of the polarity. Optionally use 4 x M3 Screws/Nuts/Spacers to raise the board.

- All LED's have their cathode (short leg) on the right. A small 'k' is printed on the back for reference.
- The Electrolytic Capacitors have their Anode (long leg) '+' pin marked.
- All diodes have their cathode (bar) to the left and one diode has it to the top. Note the white bar on the symbol.
- The 8 diodes to the right of the ROM are 1N4148. The 4 diodes in the top left corner are 1N4004.
- The sole BC557 PNP transistor is on the left side of the DIP switch. Maybe insert it first as to not mix it up.
- 4x4 Matrix and Binary LEDs can be replaced with Yellow, but not Blue or White.
- The 8 upright resistors to the left of the CPU are 10KΩ in value.
- Ensure that the 7805 Voltage regulator (TO-220 package) is soldered underneath the board flat side down first before soldering the Z80 IC Socket. Pin positions are marked. Use an M3 Screw/ Nut to secure. A heatsink is not required.
- To match the DIP Switch positions with the tutorials in the TE magazine, ensure the 'closed' or 'on' position is down. The DIP Switch might need to be flipped (optional).
- Memory selection (MEM SEL) headers and jumper can be replaced with an SPDT mini slide switch.

The board has been designed for beginners with all components being through-hole and easily sourced. There's a bit of soldering to be done and some pads are near each other, go easy on the amount of solder used and take your time. Components have been marked on the PCB as to their value and position. Refer to the PCB picture on this sheet for the component value after insertion as some values get covered.

Fault finding

The board has been tested and is fault-free. The IC's in this package have been tested before sending. If it doesn't work, go through basic fault finding ie: solder joints and component position. Look for help in Issue 13 or the onboard Probe to assist.

Powering

9V/GND from the screw terminal connection, or 5V/GND from a pin header at the top of the board. The 5V connection is unregulated, **be careful!**

Using

The 4kb EPROM provided has the original 2k code in the lower half and 5 larger programs in the upper half. To swap between the upper/lower ROM, use a 2-pin header jumper placed on the 'MEM SEL' three-pin header (the centre pin is always connected). The headers can be replaced with an SPDT mini slide switch which will make ROM selection easier. For instructions on how to use the lower ROM and upper ROM, please refer to TE magazine Issue 13 and 14 and Part 3 on the project Github address.

See <https://github.com/bchiha/microcomp> for all project-related material.

Parts List

IC's		Resistors		Capacitors	
Part	#	Part	#	Part	#
Z80 CPU	1	10 Ω	1	1 μ F 50v Electrolytic	1
2732 / 2816 (E)EPROM	1	100 Ω	8	1000 μ F 25v Electrolytic	1
74LS273 D-Flip Flop	1	330 Ω	1	1nF Polyester or MKT	2
7805 Voltage Regulator	1	470 Ω	1	100nF Polyester or MKT	1
Transistors		3.3 k Ω	1	Switches	
Part	#	4.7 k Ω	1	Part	#
BC547 NPN	8	10 k Ω	14	6mm Tactile	3
BC557 PNP	1	22 k Ω	3	SPDT mini Slide Switch	1 or 2
Diodes		39 k Ω	2	8 Way DIP Switch	1
Part	#	100 k Ω	4	Other	
1N4004	4	100 k Ω Horizontal Trimpot	1	Part	#
1N4148	9	LED's		Piezo Buzzer	1
Connectors		Part	#	2 pin header jumper	1
Part	#	FND 560 Seven Segment	2	20 pin IC socket	1
5mm 2 pin screw terminal	1	3mm RED	17	24 pin IC socket	1
Header pins	15	3mm YELLOW/RED	8	40 pin IC socket	1