CV-NUC+
MINIATURE COLECOVISION CLONE

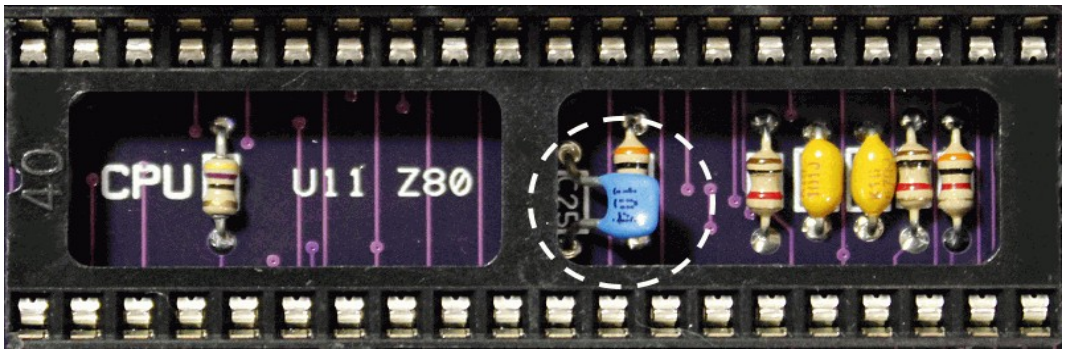
PCB ASSEMBLY MANUAL

This is not meant to be a detail by detail guide to assembling printed circuit boards, but instead serves to point out things to watch out for that are particular to this build.

MAIN PCB

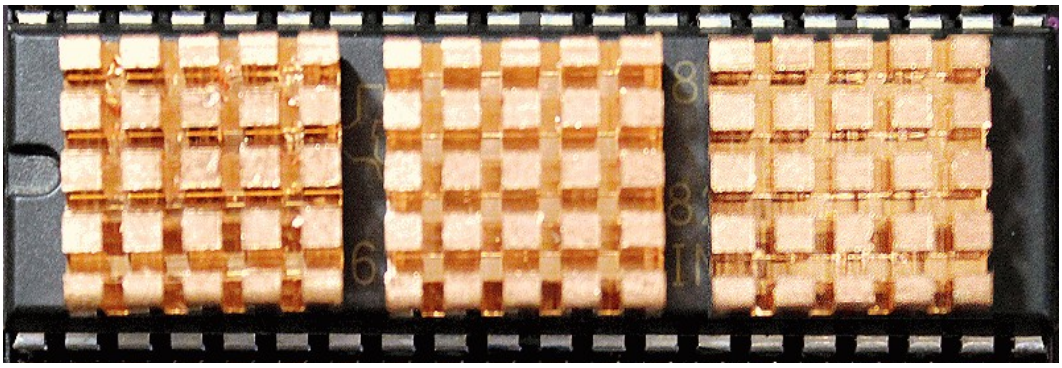
Before any IC sockets are installed on the Main Board, the 3 female headers (QUAD A, B, C) should get soldered in first followed by the IC sockets that surround them. This is a very closely spaced layout so just that little bit of slop in all the pad holes can be enough to cause an issue if you do the sockets first.

Nested capacitor C25 needs to be bent over for clearance underneath the CPU chip. This should be the last thing you do during assembly. Feel free to substitute an axial 0.1uf capacitor for C25 if you have it. I was just trying to keep the number of parts required in the BOM to a minimum, while also using radial lead capacitors wherever I could for easier assembly and less lead bending.



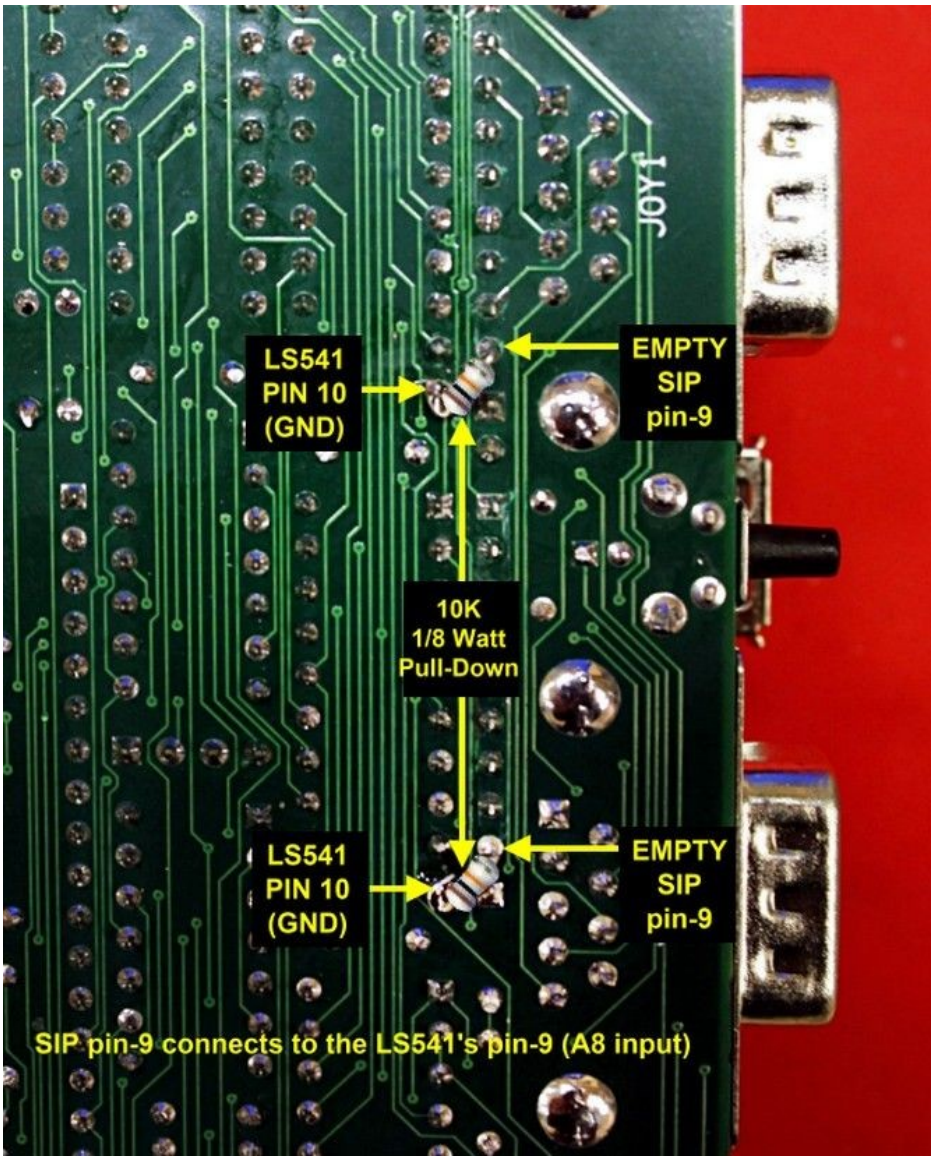
CPU Socket Showing Nested Components

The VDP chip runs hot and therefore requires a good heat sink via 3 copper self-adhesive heat sinks spaced as shown below (pre-clean chip with alcohol).



Heat Sinking the TMS9128NL Video Display Processor

In order for AtariSoft games to work properly, two 10K 1/8 watt resistors identified as RX in the BOM need to be added to the bottom of the PCB as a controller input fix. This step should be done last after all other soldering has been completed.



Controller Input Fix for AtariSoft Games

WARNING: the EzSBC PSU2 5V Regulator (U1) needs to be mounted in the proper orientation.

Proper installation will be with the component side facing forward towards the 1000 uF electrolytic capacitor as shown in the photo.

This puts the 12V input of the regulator on the left, and the 5V output on the right.

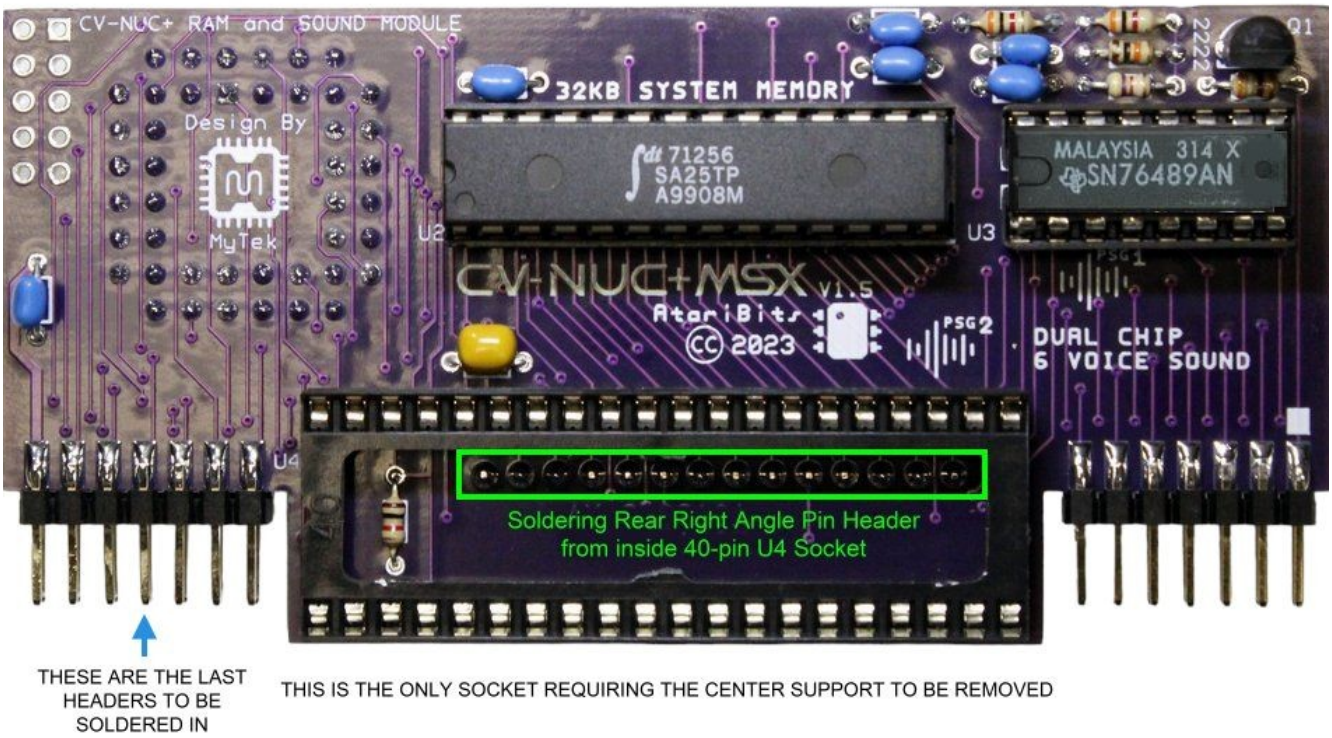
If not mounted this way it could result in failure of the regulator and/or damage to other components.

MSX MODULE

The MSX Module has an unusual method of picking up it's signals from the Main Board, where instead of an edge card connector and PCB fingers, it uses a combination of two DIP 14-pin headers and a single SIP right angle pin header. This was done in order to keep the board's height to a minimum, by allowing the 40-pin sound chip to sit very low on the board. Something that would not have been possible with a conventional card edge socket.

The tricky part of the assembly is the right angle pin header which has it's solder side nested inside U4's IC socket (the YM sound chip). This also requires breaking out the socket's center plastic support.

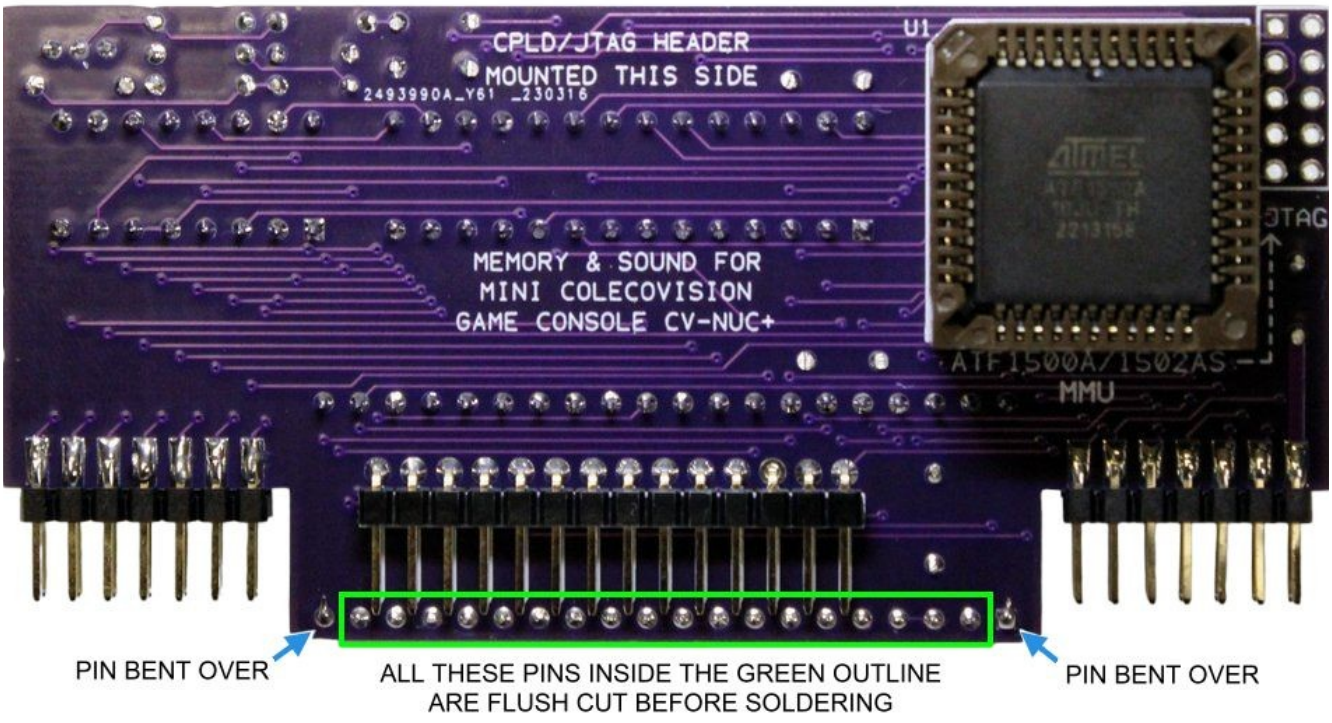
The best way to break out the center plastic support rail is to first cut it in the middle with some very small side cutters same as would be used for trimming component leads. After the middle has been cut clear through, then proceed to flush cut the opposite ends even with the inside wall of the IC socket. That concludes this operation and the socket can now be installed in the board.



Now for the assembly: the 40-pin socket needs to be soldered in first prior to installing the right angle pin header, and it requires that a special method is used to insure that the IC socket's pins don't interfere with the header.

Basically the idea is to minimize the height that the pins on the bottom edge of the socket stick up, and that is done by first flush cutting them 'before' soldering them in place. Prior to doing this, I like to bend over the two outermost pins and solder them as well as all the ones on the top row which will hold the socket in place. Then as a final step do the flush cutting and soldering of the bottom row.

THE YM1249F (U4) SOCKET GETS SOLDERED IN BEFORE THE RIGHT ANGLE PIN HEADER



As for the mating sockets on the Main Board, those should get soldered in during its assembly, making sure that they are perpendicular with the board and not at an angle. Then plug in the loose 14-pin DIP headers.

Next plug in the completed MSX Module, making sure that the edge of the MSX board fully slides into the middle section of the DIP pin headers and everything is as far down and flush as it will go. Double check that nothing is crooked and there is no space between the black plastic DIP pin header carrier and the MSX board edge, then go ahead and first solder the outermost pins to the MSX board, followed by soldering all the rest.

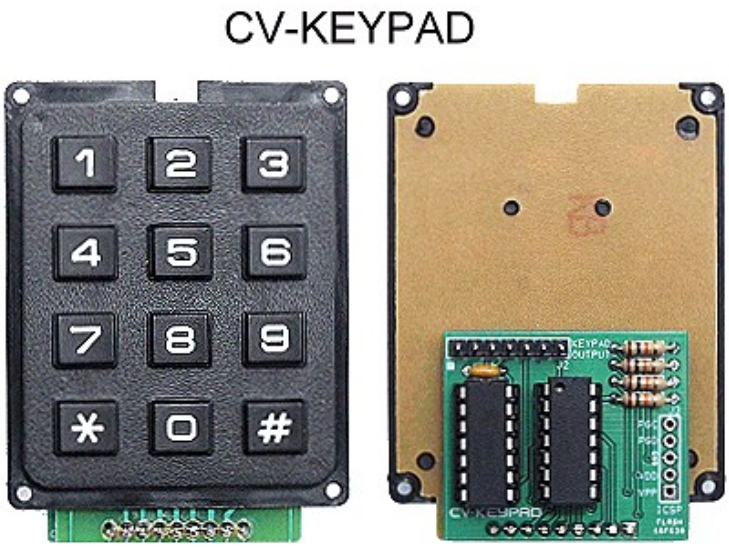
One additional note: In the early development discussions at AtariAge an AY-3-8910 chip was called for. That is no longer the case. Only the Yamaha YM2149F should be used from this point forward. It is also the easier chip to source, and it's also what fake AY chips are made from. However a real AY chip has a hotter audio output level which unbalances the mix between it and the SN sound chip in this particular design.

If the YM2149F sounds familiar (intentional pun), that's because it was the sound chip used in the Atari ST for many years, as well as on many later MSX Computer Systems.

WARNING: Before applying power to the Main Board the first time, all THT IC chips and the MSX Module should be removed and a verification made that only 5 VDC exists between U10 pin-14 and pin-7 (GND).

CV-KEYPAD

This is a little translator board that piggybacks the SparkFun numeric keypad and outputs CV compatible numbers and symbols. It serves as an internal keypad on the controller #1 port (parallel with the external DB9 connection).



The PCB gets completely assembled first, cleaned of flux, then gets sandwiched with the SparkFun keypad with a 9 pin header between. Solder is then applied to the header connections. Interconnection between the CV-KEYPAD and the CV-NUC+ is made with a 7 wire Dupont terminated flat cable assembly.