CW Widget Assembly Instructions

Ver 0.0 220629
For Rev 2 PC Boards

Before we get into the nitty-gritty of assembly, please allow me to make a few comments. I've been assembling electronics kits as well as prototypes of my own design for over 40 years. I highly recommend that you learn from my mistakes, of which there have been many!

Below are some product recommendations. I get nothing if you choose to buy one of these based on my links. I provide the links only for the reader's convenience.

If you don't have a good small (~4.5 inch – 12 cm) diagonal cutter, consider getting one, and only use it for soft metals, like component leads. It will make trimming leads easy and if not abused the tool will last you for decades. Consider the Xcelite S1415JSNN, currently priced at $33 on Amazon. https://www.amazon.com/Xcelite-S1415NJS-Tapered-Relieved-Diagonal/dp/B000JCRYKI If you don't do a lot of electronics assembly there are other lower cost diagonal cutter options available.

I've only recently started working with a silicon mat for soldering. I wish I had discovered these years ago (and my workbench surface would look better as well). Consider the Kaisi Heat Insulation Silicone Repair Mat currently priced at $9 on Amazon. https://www.amazon.com/gp/product/B07DGVJ17H/?th=1

My eyesight is getting worse as I get older. I find an magnifying hood as well as a 5x loupe to be very helpful. Don't waste money on a cheap loupe. If you are going to get a loupe, bite the bullet and get the Bausch & Lomb loupe, not some cheap knock-off. I got my magnifying hood from Amazon for about $21 and use the included 2.5X lens. https://www.amazon.com/gp/product/B07M7H3P95/ I use it constantly when I'm in my workshop. The "Loupe by Bausch & Lomb, 5x" I found on Amazon for about $20 https://www.amazon.com/gp/product/B000OVNM6Y.

For solder repairs or solder cleanup on boards with fine width traces use solder wick instead of a solder sucker. I like a solder wick that has been infused with flux (rather than no-flux wick). I've gotten very good results with this solder wick product for about $12: https://www.amazon.com/gp/product/B0195UVWJ8

When soldering surface mount ICs I find that applying a bit of liquid solder to the board and (once the corners are tack-soldered to the board) the IC leads themselves, to be very helpful. I find the applicators that are like big magic markers to work well, such as this product for around $8 on Amazon: https://www.amazon.com/Liquid-Flux-No-Clean-10ml-0-34oz/dp/B07B53LNGX. I don't bother with flux (other than that provided by resin core solder) on passive surface mount components or any through-hole components.

I use tin-lead based solder, not lead-free solder. If you are also using tin-lead solder, then make sure you work in well ventilated space. Additionally, use a small desk fan to keep

the air moving. I point the fan at the work piece and hold my breath when I solder a connection. If you smell solder when you are soldering then something is wrong with your ventilation system.

When soldering through-hole components don't use a soldering iron tip that is too small. If it's too small then it won't be able to quickly conduct the necessary heat to components that are connected to ground planes on the PC board (even if those ground pads feature thermal reliefs). A long slow application heat can damage PC board traces. I don't even use a narrow width solder tip when I'm soldering small surface mount devices. However, when soldering surface mount components I use the minimum solder needed for the job, as well as liquid flux. I will sometimes use a narrow soldering iron tip for solder cleanup on surface mount pins, if I can't to the clean up work with the standard tip and fluxed wick. My standard soldering iron tip that I use for 98% of my soldering needs (including 64 pin TQFP ICs) is a Weller ETA tip, which is conical with about a 1/16" width at the tip.

If you find you make a soldering error and have soldered the wrong component in the wrong place, I strongly recommend that you do not try to remove that incorrect component from the holes on the board. Instead cut it off of the board and then tack-solder or sweat-solder the correct part onto the board (either top or bottom side). I've destroyed too many boards trying to remove components from small, thin, plated through holes. If the holes are oversized you might be able clean out the holes with solder wick, but do not use a solder sucker unless the traces leading to the hole are very heavy (like those on a 20A power supply).

We are now at the point where surface mount components cost less than through-hole components, in some cases A LOT less. Additionally, it's becoming hard to find 5% ¼ watt through-hole resistors on the secondary market while 1% devices are more common. If your project calls for 1% resistors used in a balanced or matched configuration (the Widget does not) then be sure to measure your 1% resistors to ensure that they are closely matched. Whether a 1% 100K resistor is really 99K or 101K isn't all that important, provided that matched pairs are very close in value to each other. In some cases for the Widget you may find that you have been provided with a 1% resistor when the Bill of Materials (BOM) calls for 5%. No worries, use the 1% resistor without concern. However, for all resistors, it's always a good idea to measure them to ensure that you are putting the correct value into the desired component location. None of the resistor values used for the Widget are critical.

Substitutions

Most all resistors used on the Widget can have wide tolerance. The key concerns are that R21 needs to be between 15 and 32 ohms, R10 needs to be between 8 to12 ohms, C7 should be between 22 and 47 nF and D2 needs to be a Schottky diode. Most all other resistors can safely be 30% of their nominal value and capacitors can be -20/+80%.

Q1, Q2 and Q3 can be most any NPN transistor in a TO-92 package.

World Wide Component Shortage

The world wide component shortage is real. Many of the microcontrollers that I use for many of my projects have disappeared from distribution sources. The Widget PCB was originally designed for 28 pin DIP MCU. Unfortunately, the DIP packages have disappeared from distribution, so the board will need to be redesigned to use an SOIC-28 pin version of the AVR128DB28.

Install and Solder

With that as background, let's start the assembly.

Normally, I'd start with the low height components (resistors) and then move to components that sit taller and taller on the board. However, for the Widget we will take a slight detour.

[ ] Place a M2.5 6mm (or 4-40 ¼") screw into a mounting hole on the top side of the board, in any corner of the board. On the bottom of the board attach a M2.5 12MM (or 4-40 1-2") hex standoff and rotate it into the screw. Repeat for the other three mounting holes.

Because the CH340C is an SOIC surface mount part, I will pre-assemble the CH340K to the board as well as the USB jack and C16 and C17 . That way I can test them before other parts are committed to the board.

Alternatively, eventually assemblers will be asked to add the CH340C to the board as well as all other components.

Note that there is a silk screen dot on the board to mark where pin 1 of the CH340C should be positions. My recommendation is that you proceed as follows: If you have some liquid flux then place it onto the PCB pads for U4. Then wet the tip of your iron with some solder. Then line up the CH340C on the board and while maintaining correct alignment use your tweezers (or similar implement) to press down on the component, compressing it against the PC board, so it can't move. Then touch your iron to one of the IC leads at one of the end of the component. Remove the tweezers. Check the position of the component to ensure that all the component's legs line up nicely with the solder pads on the board. If you are not well aligned then gently and carefully reheat the one lead that's soldered to the board and readjust the position of the component as needed. Once you have one leg soldered and all other components are well aligned, then solder the most distant pin on the opposite side of the component (diagonal). Again check the alignment of the pins and pads. If that looks good then solder all the remaining pins of the component.

[ ] Use a 5x or greater loupe to inspect your soldering of U4, the CH340C SOIC part. Touch up your soldering if necessary.

[ ] At C16 and C17 install and solder a 100nf cap (marking: 104).

[ ] At J1 install and solder the USB jack.

With those components now in place use a USB cable to connect the Widget to a Windows 10 or 11 computer. Open Device Manager and check Ports (COM & LPT) to see if the Widget is enumerated. If so it will show up as CH340. Disconnect the USB cable from the Widget and see that the Widget is no longer enumerated.

Next up is installing and soldering all the fixed value resistors on the board. For each resistor value the color code for 5% resistors (4 bands) is listed first followed by the color code for 1% or 2% resistors (5 bands). On the 5 band resistors if the last band is Brown it's a 1% resistor, if Red then it's a 2% resistor.

[ ] At R20 install a 10 ohm resistor (Brown-Black-Black-Gold *or* Brown-Black-Black- Gold-Brown/Red)

[ ] At R21 install a 27 ohm resistor (Red-Violet-Black-Gold *or* Red-Violet-Black-Gold-Brown/Red) at R21.

[ ] At R26 install 470 ohm resistor (Yellow-Violet-Brown-Gold *or* Yellow-Violet-Black-Black-Brown/Red). DO NOT YET ASSEMBLE R22 or R23. THAT WILL BE DONE AT THE END OF ASSEMBLY PROCESS.

[ ] At R12, R24 and R25 install 1K ohm resistors (Brown-Black-Red-Gold *or* Brown-Black-Black-Brown-Brown/Red).

[ ] At R1, R2, R5 and R13 install 2.2K ohm resistors (Red-Red-Red-Gold *or* Red-Red-Black-Brown-Brown/Red)

[ ] At R27 and R28 install 4.7K ohm resistors (Yellow-Violet-Red-Gold *or* Yellow-Violet-Black-Brown-Brown/Red). DO NOT YET ASSEMBLE R3, THAT WILL BE DONE AT THE END OF ASSEMBLY PROCESS.

[ ] At R4, R9, R10, R14 and R19 install 10K ohm resistors (Brown-Black-Orange-Gold *or* Brown-Black-Black-Red-Brown/Red).

[ ] At R6, R7, R8 and R11 install 47K ohm resistors (Yellow-Violet-Orange-Gold *or* Yellow-Violet-Black-Red-Brown/Red).

[ ] At R18 install 100K ohm resistors (Brown-Black-Yellow-Gold *or* Brown-Black-Black-Orange-Brown/Red).

Now install the diodes onto the board.

[ ] At D1 install a 1N4148 diode. This looks like a small old-time glass diode. Make sure that the band on the diode (cathode) is closest to Q1.

The diode at D2 must be a Schottky diode, which has the characteristic of low forward voltage. It will look like a conventional power diode. If you have any concern that the diode you are about to install at D2 may not be a Schottky, there is an easy test you can perform to verify that it is a Schottky. Obtain a 9V battery, one of the 4.7K ohm resistors for this project and the candidate Schottky diode. Using clip leads form a series circuit starting at the positive terminal of the 9V battery to one end of the 4.7K resistor. Connect the other end of the 4.7K resistor to the diode anode (the end opposite the band) of the diode. Connect the cathode of the diode (the end with the band) to the negative terminal of the battery. The current through that series circuit should be about 1.85 mA. Measure the voltage across the diode. If it is a Schottky diode the voltage will be less than 250 mV. If it is a conventional silicon diode the voltage will be more than 600 mV. If the diode has Schottky characteristics then install it at D2. If the diode does not behave like a Schottky diode, then you will need to find a true Schottky diode before installing that diode at D2.

[ ] At D2 install and solder a 1N5819 (or most any Schottky) diode. The banded end of the diode should be closest to U1.

Next, install the capacitors.

The capacitor are usually yellow ceramic capacitors with a lead spacing of 5 mm. The marking is usually three digits that show the value in pF, where the first two digits are the significant figures and the third digit is the number of zeros to append after the significant figures. For example, 47 pf is marked 470, 220pf is 221, 3.31nF is 333, 47nF is 473, 100 nF is 104, 1 uF is 105 and 10 uF is 106.

[ ] At C7 install a 47nF ceramic capacitor (marked 473 or 0.047).

[ ] At C2, C3, C6, C8, C9, C10, C12, C13, C16 and C17 install a 100nF ceramic capacitor (marked 104 or 0.1).

[ ] At C1, C4 and C5 install a 1uF ceramic capacitor (marked 105 or 1.0).

[ ] At C15 install a 10uF 16V (minimum) ceramic capacitor (marked 106 or 10).

[ ] At C11 and C14 install and solder 1000uF 16V (minimum) electrolytic capacitors (4.7 or 5mm lead spacing). The short leg marks the negative lead as does the band on the body of the capacitor. The negative lead of the capacitor should be inserted into the component location on the board marked with solid silk screen. I find it helpful to solder only one lead, then press the component into the board while re-heating that one lead, to ensure that the bottom of the capacitor is flush with the board. When I'm sure that the capacitor body is flush with the board I then solder the remaining lead.

Next, install the transistors.

[ ] At Q1, Q2 and Q3 install 2N3904 transistors. Match the flat side of the component body with the flat side of the silk screen.

Next, install the 3.5mm phone jacks.

You may find that the jacks are a bit of a sloppy fit when the pins are inserted into their mounting holes. That's because the PCB footprint for these parts has been modified to accommodate two different jacks (with a slightly different pin positions for each jack). Solder the ground pin first (the pin closest to the board edge). Then wiggle the part around a a bit so that the edge of the jack is parallel to the board edge. When it's well aligned then solder the other pins of the part.

[ ] At J2, J3, J4 and J5 install and solder the 3.5mm TRS phone jacks.

Next, install the pin headers.

My method of installing and soldering pin headers is to first attach a shorting shunt to any two pins on the long side of the pin posts. I then put the short side of the pin posts into the holes on the board. I press my finger on the shunt to hold the pin posts in place when I turn the board over for soldering. I then solder just one pin at the end of the set of pin posts (and not a ground pin). I then check the alignment of the pin posts to ensure that they are perpendicular to the board and not "leaning" or "twisted". I re-heat the one soldered pin and adjust my finger on the shunt until the pin post is perpendicular with the board and that all of the pin post plastic retainer is flush with the board. With that accomplished I then solder all the remaining pins of the pin posts, and then re-touch up the first pin I soldered that held things into place. The purpose of placing a shunt onto the pin posts is to provide some thermal insulation so that your finger doesn't get burned while you solder the first pin.

[ ] At PH1, PH6, PH7, PH3, PH9, PH11, PH12 and PH13 install and solder a 2 terminal pin post.

[ ] At PH2 and PH8 install and solder a 3 terminal pin post.

[ ] At PH10 install and solder a 4 terminal pin post.

[ ] At PH14 install and solder a 8 terminal pin post.

The LED on the board is only used during the configuration process. It doesn't need to be headlight bright, nor should it be too dim. The LED provided to you with the kit of parts may be of somewhat unknown characteristics, so you may find it helpful check that you like the LED brightness provided by the particular value of current limiting resistor, before you solder that resistor onto the board

If you connect a USB cable from your Widget to a power source, you will have a source of +5V from the test pins the Widget labeled +5V and GND. Proceeding VERY CAREFULLY, connect a clip lead from +5V to one end of a 4.7K resistor, connect a clip lead from the other end of the resistor to the long lead of the LED, connect a clip lead from the short lead of the LED to ground. Observe the illumination of the LED. Is it satisfactory? If so, then use a 4.7K resistor at R3. If the LED is too dim, then try a smaller resistor. It shouldn't be necessary to go below 470 ohms.

[ ] At R3 install and solder the resistor value as determined by the procedure above.

[ ] At DS1 install a 3mm LED (your choice of color). The short lead marks the cathode end, which should be installed into the hole closest to J2, the paddle jack.

Different types of headphones have different electrical-to-acoustic efficiencies. The value of R22 and R23 will determine how "loud" the headphones are relative to the "loudness" when using a loud speaker. You can go with the nominal value of 470 ohms in series with your headphones or you can select a resistor value such that a comfortable volume on the speaker also results in a comfortable volume on your headphone. In my own case, I found that 470 ohms was a good choice for me. Whatever value you chose should be installed as listed below.

[ ] At R22 and R23 install 470 ohm resistors (Yellow-Violet-Brown-Gold *or* Yellow-Violet-Black-Black-Brown/Red) or values of your choosing.

This completes the assembly. Carefully check your board for any solder spashes or component leads that are not well soldered (or not soldered at all!). Touch-up as needed.

---ooOoo---