USBMorseKOB Assembly Instructions

Instructions Ver 1.1 (221108)
For Rev 3 PC Boards (220304)

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 encourage others to 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 these links. I provide the links only for the reader's convenience. You certainly don't need to buy or even have any of these items to assemble an electronics board, but I've found that good tools provide a path to better results.

If you don't have a good small (~4.5 inch – 10 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. If you don't do a lot of electronics assembly there are several low cost diagonal cutter options available. But if you do a lot of electronics assembly and component lead cutting, then consider the Xcelite S1415JSNN, currently priced at $33 on Amazon. https://www.amazon.com/Xcelite-S1415NJS-Tapered-Relieved-Diagonal/dp/B000JCRYKI.

One tool that I've had and used for about the last 40 years is a simple plastic jig for bending ¼ resistor leads for 400 mil pitch holes. The device is an Eagle Plastic Devices 5166-801 lead bend former. Mouser has them for about $5. They are about $8 at DigiKey. I have some extras which I can make available for reimbursement cost.

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 that a magnifying hood as well as a 5x loupe are very helpful. 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. If you get a loupe don't waste money on a cheap one. Bite the bullet and get the Bausch & Lomb loupe, not some cheap knock-off which will likely show you distorted images (ask me how I know). 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 flux 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 rosin 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 of 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 clean up the work piece with the standard tip and fluxed wick. My standard soldering iron tip that I use for 99% 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 through-hole 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 the component off of the board and then either (a) tack-solder or sweat-solder the correct part onto the board (either top or bottom side) or (b) ensure that the component lead on the bottom side of the board is cut flush with the board, then cut the component leads on the top side so as to leave about ¼ of an inch remaining, then gently pull on that remaining stub with needle nose pliers which heating the joint, and gently pulling out the stub lead, then clean the hole with solder wick, and insert the replacement component. However, I've destroyed too many boards trying to remove components from small, thin, plated through holes, so I think that option (a) above is the best approach. If the holes are very large (like those on a PCB for a 20A power supply) you might be able clean out the holes with a solder sucker, but only try that if the traces leading to the hole, and the hole itself, are very large.

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 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 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 before installing them into the board, to ensure that you are putting the correct value into the desired component location.

Bill of Materials

Mechanical

1x PC Board, Rev 3
4x M2.5x6 nylon screw
4x M2.5x12 nylon hex standoff
2x Fuse Clip, Littlefuse FC51FL
1x Fuse, 5x20mm, 500 mA, Littlefuse 0617.500MXP

Connectors

1x 2.1mm Coax Power Jack, J2
1x 2.1mm Coax Power Plug
1x 3.5mm TRS Jack, J3
2x 2 pin header 100 mil pitch, PH1, PH5
1x 2 pin shunt, 100 mil pitch

USB

1x USB Mini Through Hole Connector, J4
1x CH340C SOIC-16, U1
1x 10uF Cap (yellow, 106) C1
2x 100nF Cap (yellow, 104) C2, C3,

Semiconductors

3x 2n3904 TO-92 NPN Transistor, Q1, Q2, Q3
1x 2n3906 TO-92 PNP Transistor, Q4
4x MPSA42 TO-92 NPN 300V Transistor, Q5, Q6, Q7, Q8
2x 1N4148 diode, D1, D2
1x 1N400x diode, D3 (1N4001 through 1N4007 are OK)
1x Littlefuse P6KE150A, UniDirectional Clamp Diode 150V, D4
1x LED 3MM, DS1

Resistors

1x 6.8 Ω ½ W resistor (for 4 ohm sounder) R1
1x 33K Ω ¼ Watt 5% resistor, R2
3x 100K Ω ¼ Watt 5% resistor, R3, R5, R6
4x 10K Ω ¼ Watt 5% resistor, R7, R8, R9, R11
5x 1K Ω ¼ Watt 5% resistor, R10. R13, R15, R17, R19
1x 47 Ω ¼ Watt 5% resistor(for 5V loop driving voltage), R12
4x 3.9 Ω ¼ Watt 5% resistor, R14, R16, R18, R20

Install and Solder

With all 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 this project we will take a slight detour.

Board Legs

[ ] 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.

USB to Serial Converter IC

Because the CH340C is an SOIC surface mount part, I will pre-assemble the CH340K to the board as well as the USB jack, C1, C2 and C3. That way I can test the USB interface before other parts are committed to the board.

Alternatively, assemblers can add the CH340C to the board as follows:

Note that there is a silk screen dot on the board to mark where pin 1 of the CH340C should be positioned. My recommendation is that you proceed as follows: If you have some liquid flux then place it onto the PCB pads for U1. Then wet the tip of your iron with some solder. Then line up the CH340C on the board and, while maintaining correct alignment of legs to pads, 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, to solder it to the pad. 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 legs are well aligned with pads, then solder the most distant pin on the opposite side of the component (diagonal). Again check the alignment of all legs and pads. If that looks good then apply some flux to all the pins/pads and solder all the remaining legs of the component to the pads on the board.

[ ] Use a 5x or greater loupe to inspect your soldering of U1, the CH340C SOIC part. Touch up your soldering if necessary using solder wick.

[ ] At C1 solder a 10uF non-polar cap (yellow, marked 106).

[ ] At C2 and C3 solder a 100nF non-polar cap (yellow, marked 104).

[ ] At J1 or J4 install and solder a 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 listed (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.

Resistors

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 R3, R5 and R6 install 100K ohm resistors (Brown-Black-Yellow-Gold *or* Brown-Black-Black-Orange-Brown).

[ ] At R4, R7, R8, R9 and R11 install 10K ohm resistors (Brown-Black-Orange-Gold *or* Brown-Black-Black-Red- Brown).

[ ] At R13, R15, R17, and R19 install 1K ohm resistors (Brown-Black-Red-Gold *or* Brown-Black-Black-Brown- Brown). *** DO NOT YET ASSEMBLE R10. THAT WILL BE DONE AT THE END OF ASSEMBLY PROCESS.***

[ ] At R14, R16, R18 and R20 install a 3.9 ohm resistor (Orange-White-Gold-Gold *or* Orange-White-Black-Gold-Brown).

[ ] At R12 install 47 ohm resistor (Yellow - Violet -Black-Gold *or* Orange - Orange - Black-Black- Brown).

[ ] At R2 install 33K ohm resistor (Orange - Orange -Orange-Gold *or* Yellow-Yellow-Black-Red- Brown).

Diodes and Transistors

[ ] At D1 and D2 install a 1N4148 diode. This looks like a small old-time glass diode, that has an orange tint to it, and is marked as 1N4148. Make sure that for D1 band on the diode (cathode) is closest to fuse F1 and that for D2 the band is closest to R3.

[ ] At D3 install a 1N4007 diode. This looks like black power diode with a silver band and is marked 1N4007. Make sure that the band on the diode (cathode) is closest to transistor Q1. Actually anything from 1N4001 to 1N4007 will work for D3. NOTE: IF YOUR INTENDED LOOP CURRNET IS LESS THAN 100 MA THEN USING *TWO* 1N400X DIODES IN SERIES AT D3.

[ ] At D4 install a P6KE150A transorb diode. This looks like black power diode with a silver band and is marked P6KE105A. Make sure that the band on the diode (cathode) is closest to transistor Q1.

[ ] At Q1, Q2 and Q3 install 2N3904 transistors (maybe marked as 3904). Match the flat side of the component body with the flat side of the silk screen. Double check the transistor number and orientation before soldering these components to the board.

[ ] At Q5, Q6, Q7 and Q8 install MPSA42 transistors (may be marked as A42). Match the flat side of the component body with the flat side of the silk screen. Double check the transistor number and orientation before soldering these components to the board.

[ ] At Q4 install a 2N3906 transistor transistors (maybe marked as 3906). . Match the flat side of the component body with the flat side of the silk screen. Double check the transistor number and orientation before soldering this component to the board.

Remaining Connectors and Fuse

Next, install the 3.5mm TRS jack.

You may find that the 3.5mm TRS jack is 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 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 J3 install the 3.5mm TRS jack as recommended above.

[ ] At J2 install the 2.1mm coaxial DC power jack.

[ ] Form the fuse assembly by inserting a 5x20 MM fuse into the two fuse clips and align the fuse clip pins so that they are all oriented in the same direction. Use a fuse value of about 1.5x to 2x for the typical sounder current you intend to run (500mA for 4 ohm sounders).

[ ] At F1 insert the fuse assembly and solder into place.

Current Limiting Resistor R1

Before installing a resistor at R1, you will need to select its resistance value and power rating. Ohm's law applies. In general the current limiting resistor should be 1x to 3x the value of the sounder's resistance. If you choose a resistance value that is too small (and size the voltage driver for desired sounder current) then the sounder may not have

"snappy" action when the key is closed. If you choose a resistance value that is too large, you will just waste power and generate excess heat.

For R1 I'm using a value of 6.8 ohms 1/2 watts for R1 when using a 4 ohm sounder with a 5V driving voltage. When using a 150 ohm sounder I add another 100 ohm resistor to the loop circuit (external to the USBMorseKOB interface) and use a drive voltage of 12 VDC.

[ ] At R1 install 6.8 ohm resistor (Blue-Gray-Gold-Gold *or* Blue-Gray -Gold-Brown). .

Local Loop Bias Resistor R12

The purpose of R12 is to cause the common node of R13, R15, R17 and R19 to go to about 4 volts when there is a shorting block on PH5. The value of the resistor depends on the loop circuit driving voltage. For 5V use 47 ohms. For 12V use about 470 ohms. PH5 provides a means to use the interface without need for a connection to a computer (so that only the key controls the sounder). This is only a convenience feature. If all you want is a local loop without ever having computer control, then there's really no need for the USBmorseKOB interface. But if you already have a USBmorseKOB in circuit and for some reason you don't want/need computer control, then you can have a local loop by installing PH5.

Pin Posts - Test Points

My method of installing and soldering pin posts or 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 row 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 and PH5 install and solder a 2 terminal pin post. PH1 is labeled GND,GND. Note that PH2 (labeled +5V,+5V) is not normally populated and that PH3 (labeled +5V,GND) is not normally populated.

LED Install

The LED on the board is used indicate when the sounder should be active. It doesn't need to be headlight bright, nor should it be extremely dim. The LED provided to you

with the kit of parts may be of somewhat unknown characteristics, so you may find it helpful to check that you like the LED brightness provided by the particular value of LED current limiting resistor, before you solder that resistor onto the board.

In my own case, for the 3mm red LEDs that I have, a 1K ohm current limiting resistor seems to work well. Alternatively, you can test the LED brightness before committing the resistor to the board, as follows: connect a USB cable from your USBMorseKOB to a power source, you will then have a source of +5V from the test pins labeled +5V and GND (PH1 and PH2). Proceeding VERY CAREFULLY, connect a clip lead from +5V (either pin of PH2) to one end of a 1K 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 (either pin of PH1). Observe the illumination of the LED. Is it satisfactory? If so, then use a 1K resistor at R10. If the LED is too dim, then try a smaller value resistor. If it's too bright, then try a larger value resistor. It shouldn't be necessary to go below 470 ohms. When finished, disconnect the clip leads from PH1 and PH2.

[ ] At R10 install and solder the resistor value as determined by the procedure above (nominal value is 1K).

[ ] 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 R9.

Assembly Complete

This completes the assembly. Now do a careful visual check the board to look for any solder splashes or component leads that are not well soldered (or not soldered at all!). Touch-up as needed.

---ooOoo---