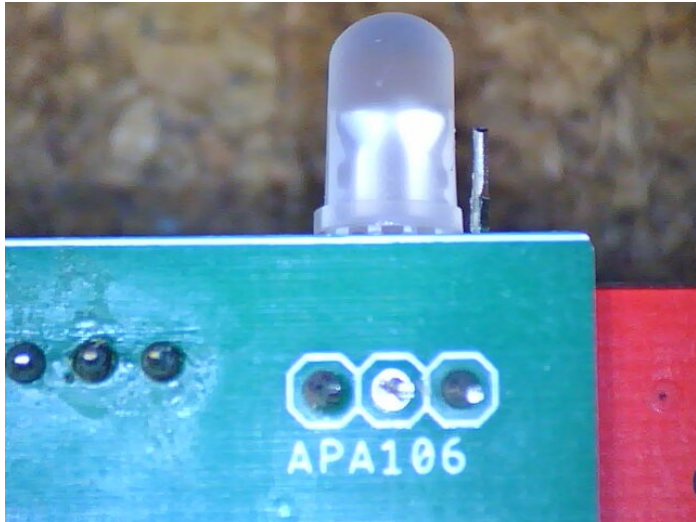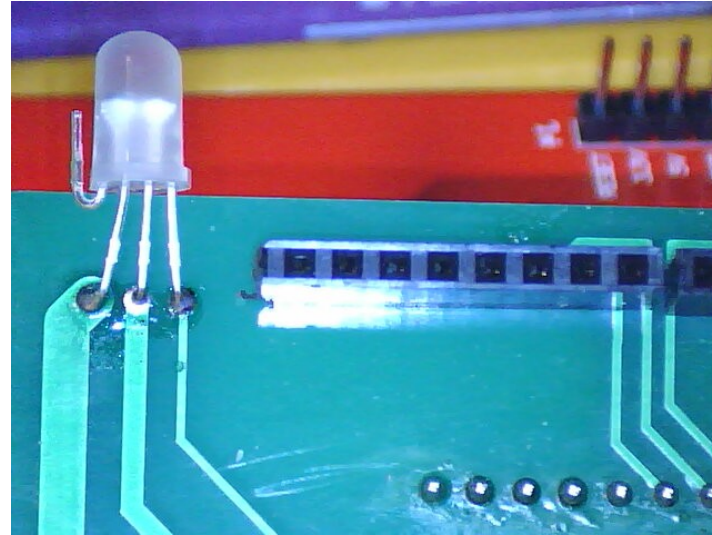# JLCPBC Build Notes

This not a step by step Instruction.
But is here to give a general
idea of what a completed board
can look like.

LED Pin Connections
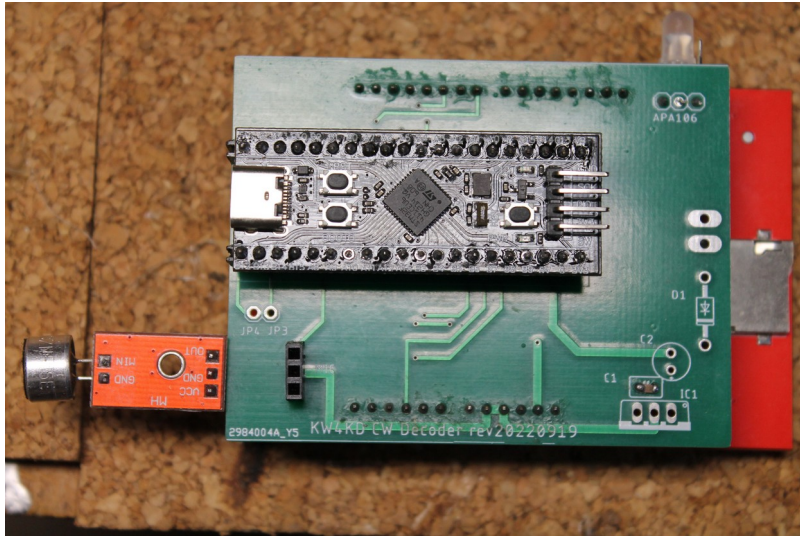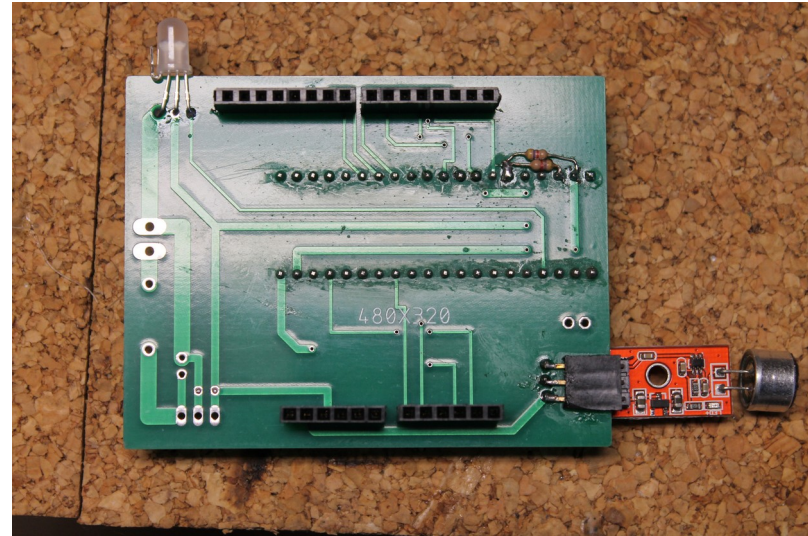

BlackPill Side

Display Side

Note, Unused LED pin is on the "flat side" of the LED

Decoder Powered via BP USB'C'

BlackPill Side:
1. 7805 regulator, & related parts, not needed.
2. Mic "out" signal goes to upper pin.
3. C1, 0.1ufd, capacitor is installed.

Display Side:
1. If needed, solder 'tone input' bias resistors directly to Gnd, & 3.3v BP header pins, as shown above.
2. Resistor value not critical (10K to 3.3K), but must be a matched pair; i.e., divide the 3.3v by 2.

Construction Tips

1. Due to spacing error in the display header pins, 1st install the 8 & 6 pin header sockets, on the display, & then fit the pin side of these sockets to the JLCPCB mother board, before soldering the mother board connections.
2. If a "wired" tone input is desired, omit the mic board, & include the resistor bias network (shown in previous slide). And finally, use a 0.1ufd blocking capacitor in series with your wired input.
3. Some Mic boards (like the one shown in the above photos) will need the bias network. Others (typically marked GY MAX4466) will not need this network.
4. Use "F411CWDecoder_202nnnnn.bin" and STM's SMT32CubeProgrammer to "flash" the program to your Blackpill MPU. Alternatively compile the scouce code using files found in F411CWDecoder__202nnnnn.zip and STM's STM32CubeIDE (ver 1.9.0). Instructions for both methods can be found in the "Build Docs" folder of this repository.
5. See "DecoderUserGuide.pdf" for a general introduction to the features found in the decoder program.