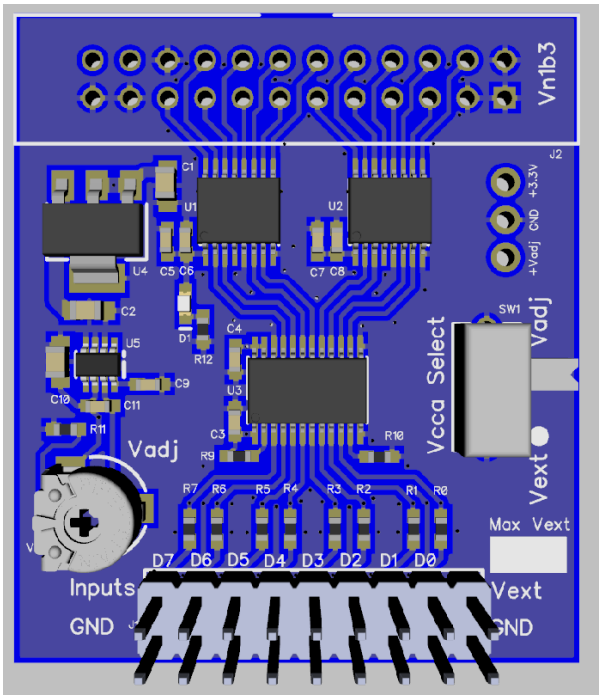
Build notes for Probe PCB vn1b3

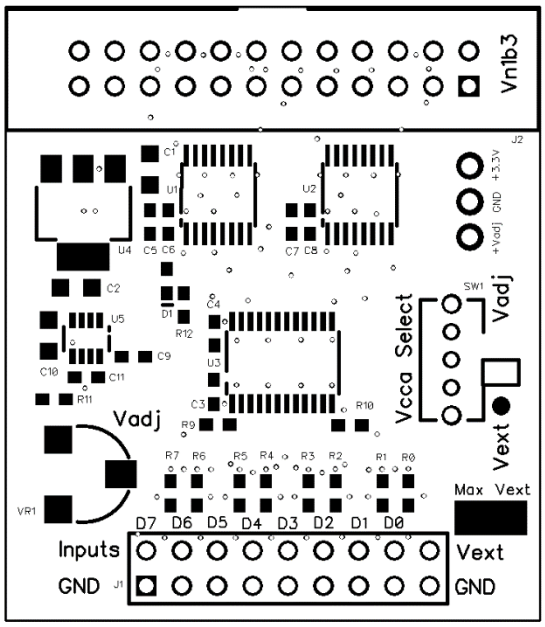


Here's a picture to help with the orientation and location of all the components.

General Notes

Believe it or not, I use a fairly large T12-K soldering iron tip running at 28˚C, 0.38 mm multicore solder, and a no clean flux pen. When soldering the finer pitch ICs, you may bridge the pins but you can go back afterwards and use the wide edge of the iron on a piece of solder wick braid that you've wiped with the flux pen; creep up on the pins several at a time and use the wick to absorb the excess solder.

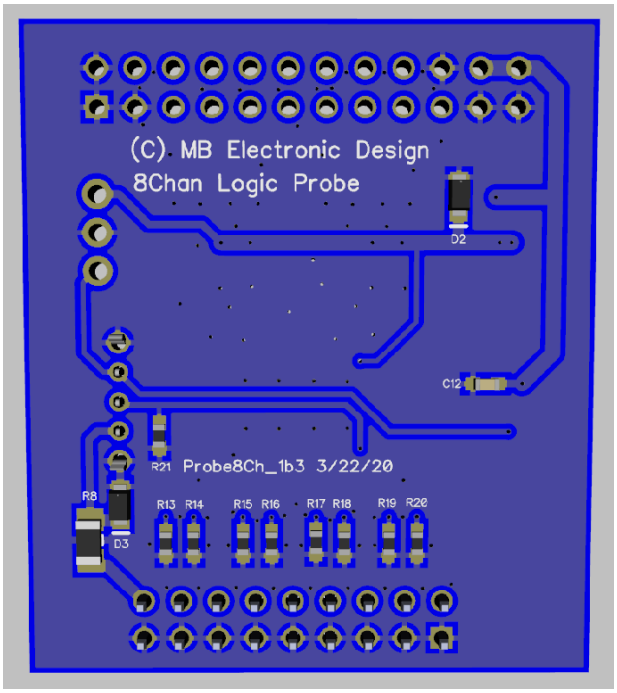
I use acetone to clean up afterwards but it's bad for you so try not to breath it in.

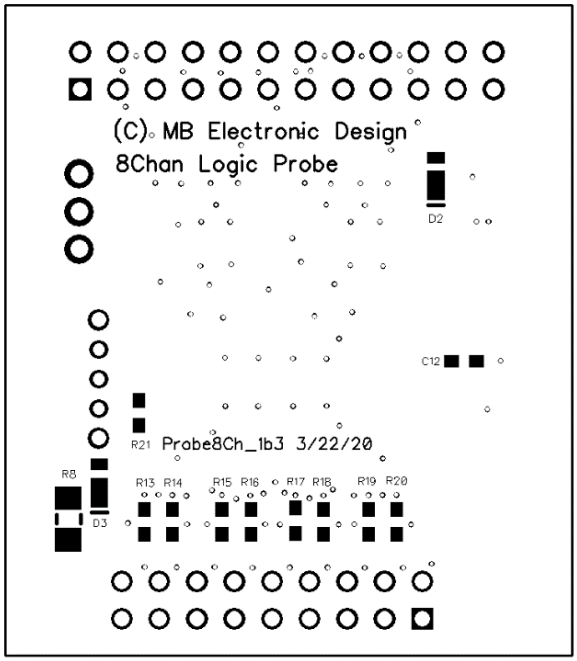


Top Side

1. Place U5 making sure the pin 1 '^' marking is nearest D1 (top right)
2. Place C9, C10, C11, R11, VR1.
3. At this point you can apply 4.2V to J2 pin 13 (top left of PCB) and ground and test that Vadj is varying smoothly between 0.65V and 3.9V as you rotate VR1.
4. Place U4, C1, C2, apply +4.2V and test +3.3V is good.
5. Place U1, U2, U3 (pin 1 is bottom left) and then remaining top side components saving J1 and J2 until last.
6. Note that the gap in J2 needs to be upwards (at the edge of the PCB).
7. Don't forget that D1, R12, U3, and R10 are different if you're building the +5.5V version.

Bottom Side





1. Place all bottom side components.
2. Don't forget that D3 is different if you're building the +5.5V version
3. Note that D3 in the CAD rendition seems to show a dot at the top side, ignore this, it's a bad CAD model for the diode, the marking on the diode is for the cathode and it should be towards the line as expected.
4. R21 is 100k and it can be placed to pull Vcca down if there's no phantom activity when there's no Vcca attached.

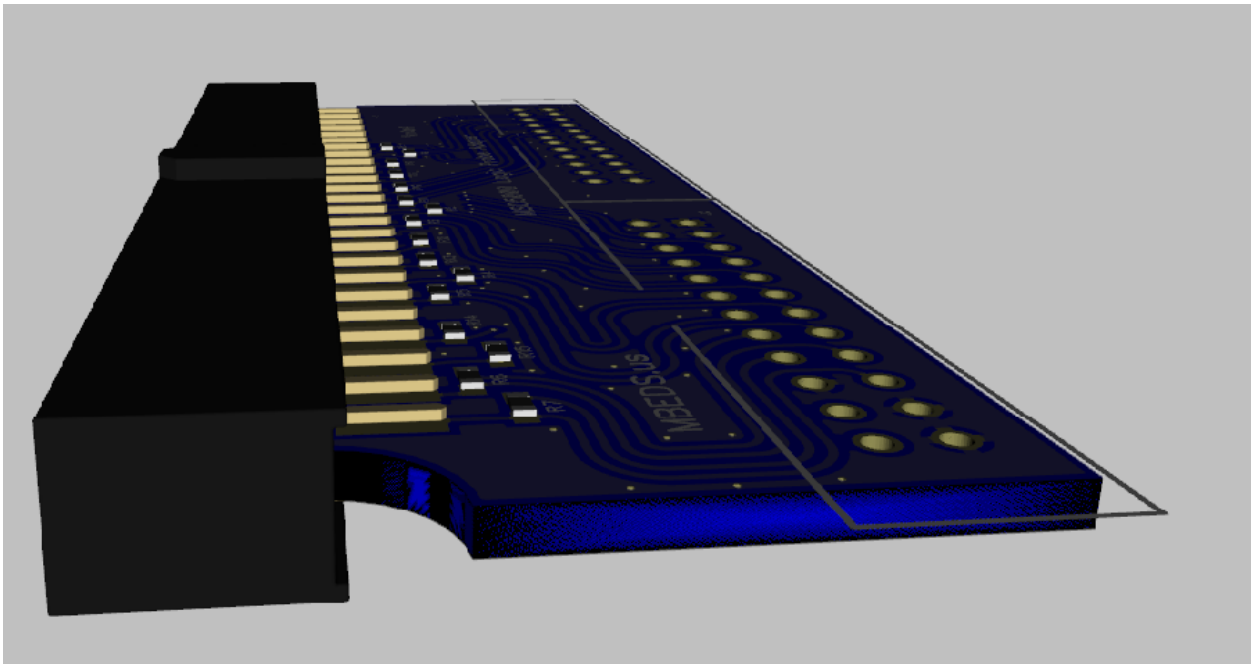
General

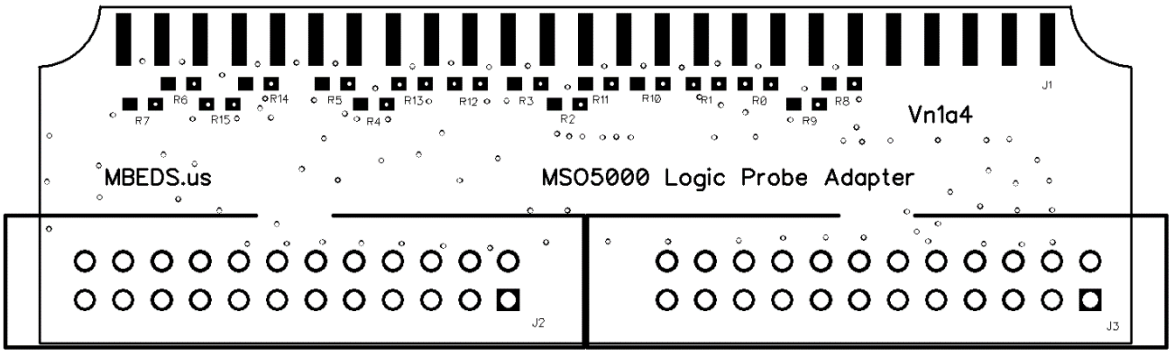
Clean with solvent such as acetone to remove flux residue.

Write '3.6V' or '5.5V' in the white rectangle silk screen area in permanent marker pen.

Build notes for Connector PCB vn1a4

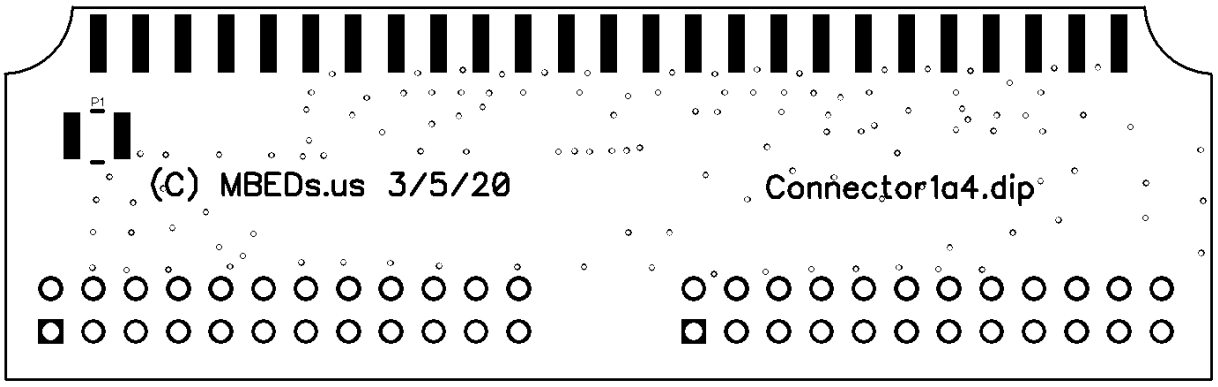
Here's a picture to help with the orientation and location of all the components.





Top Side

1. Note, the top side is the one that has the 16 0603 resistors on it and they have via in pad for some of them.
2. Fit the 16 x 100 Ohm resistors, R0 – R15
3. Fit J1 by carefully bending the pins slightly towards each other and then sliding them over the PCB from one side with the LUG on the top side of the connector.
4. Making sure J1 is aligned well with the pads, solder one pin on one side and then look from the side to check that the connector is aligned properly before soldering the remaining pins. If J1 needs realignment do it using heat, don't try to bend the pins or you'll pull the pads off the PCB.
5. Place J2 and J3 with the gaps facing upwards (towards J1). Solder in place. Yes, they hang over the sides of the PCB, it's a feature, not a bug.



Bottom Side

1. Place P1 which can either be a PTC or a ferrite bead.

General

Clean with solvent such as acetone to remove flux residue.