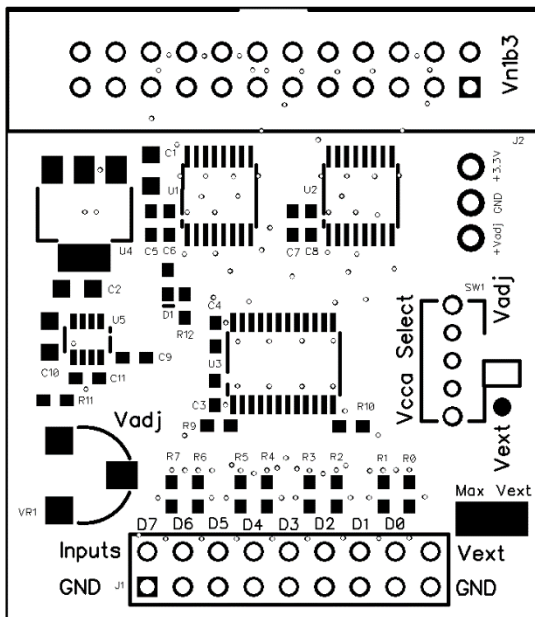
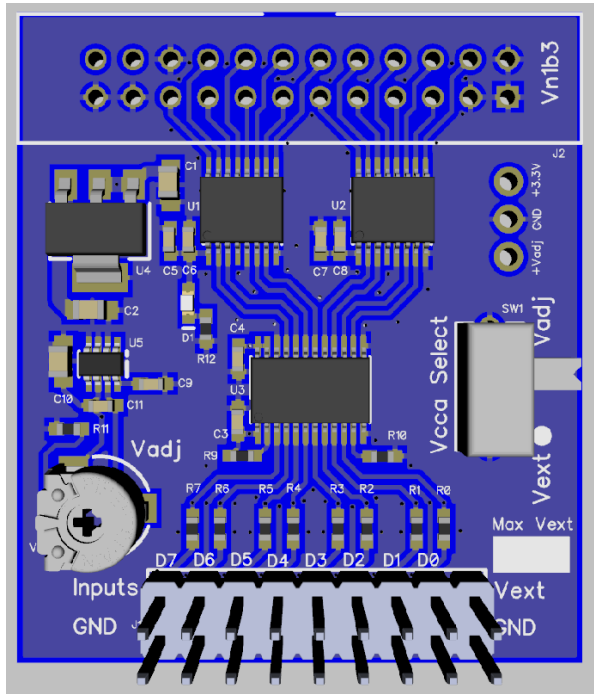
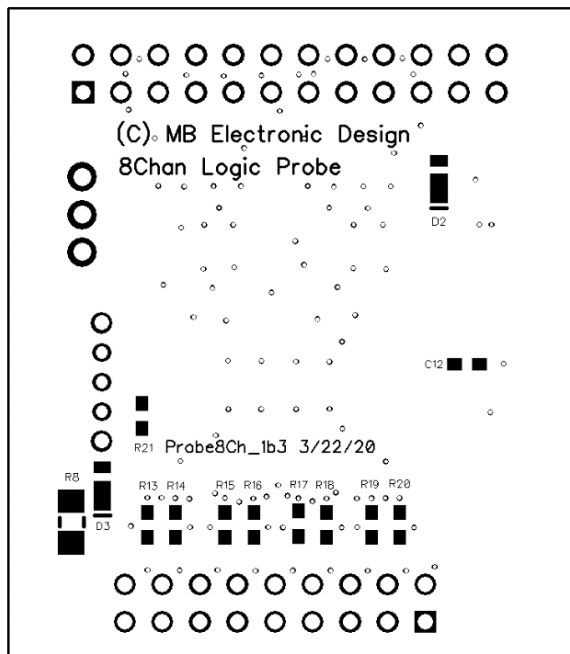
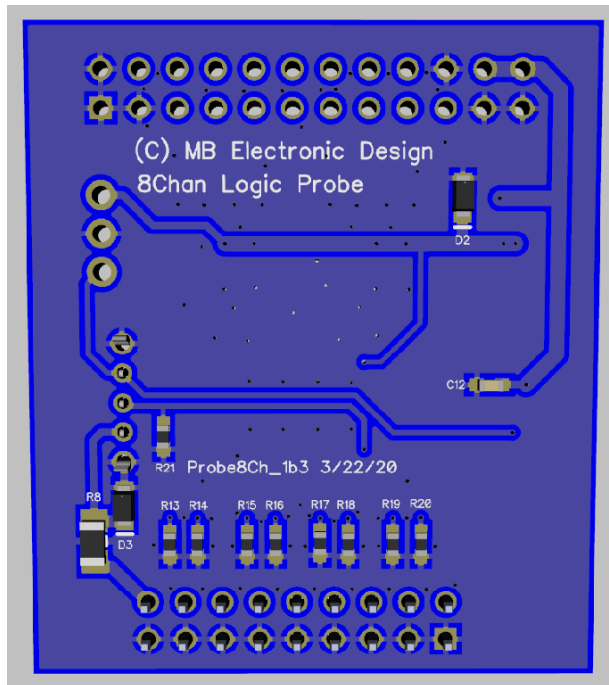


**Build notes for Probe PCB vn1b3**

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

**Top Side**

1. Place U5 making sure the pin 1 'Λ' marking is nearest D1
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 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 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**

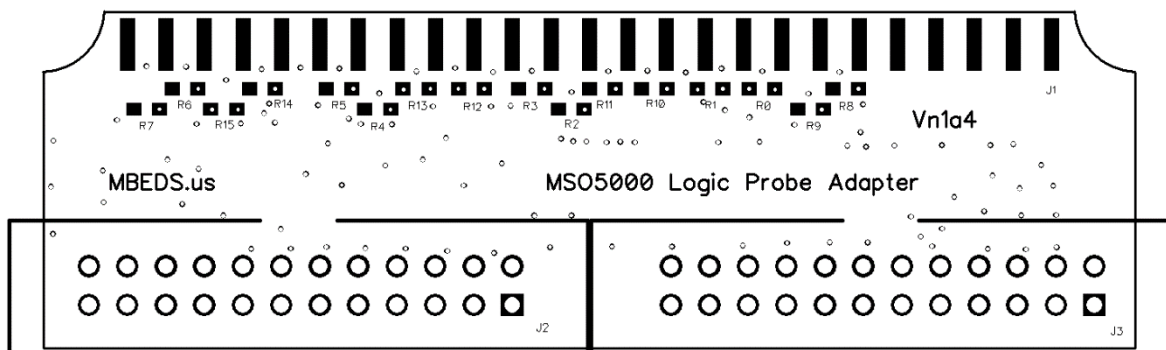
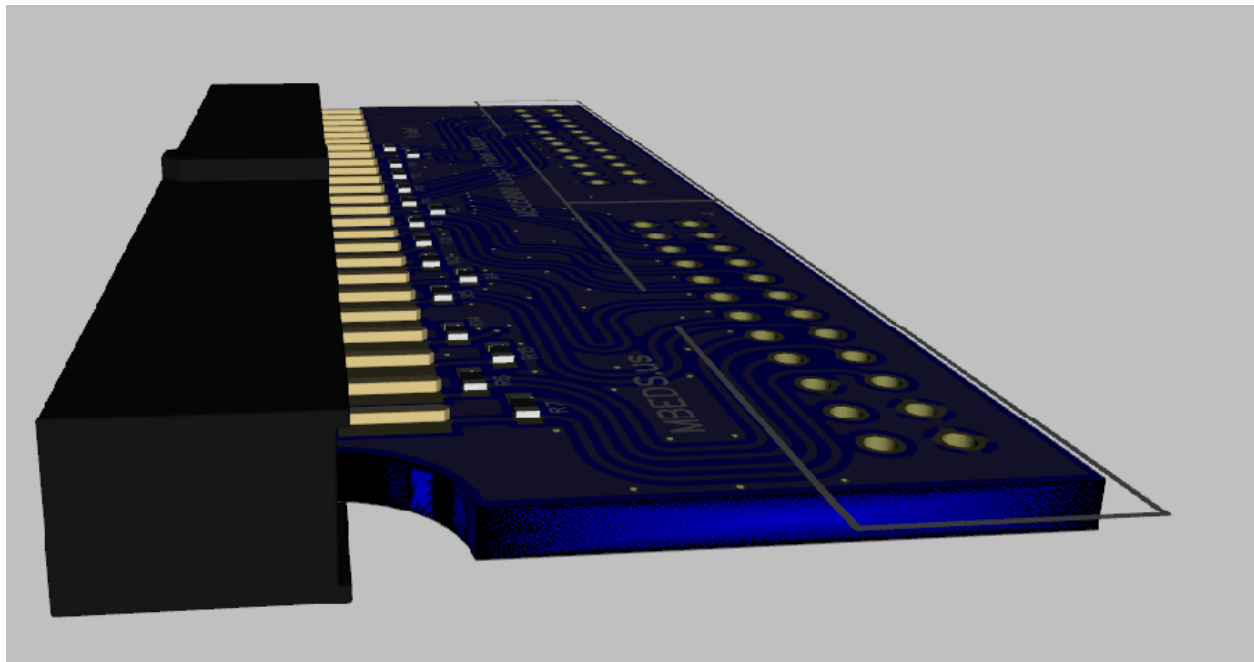
1. Place all bottom side components.
2. Don't forget that D3 is different if you're building the +5.5V version
3. Note that D3 seems to show a dot at the top side, ignore this, it's a bad CAD model for the diode, the marking on the diode is for the cathode and it should be towards the line as expected.

**General**

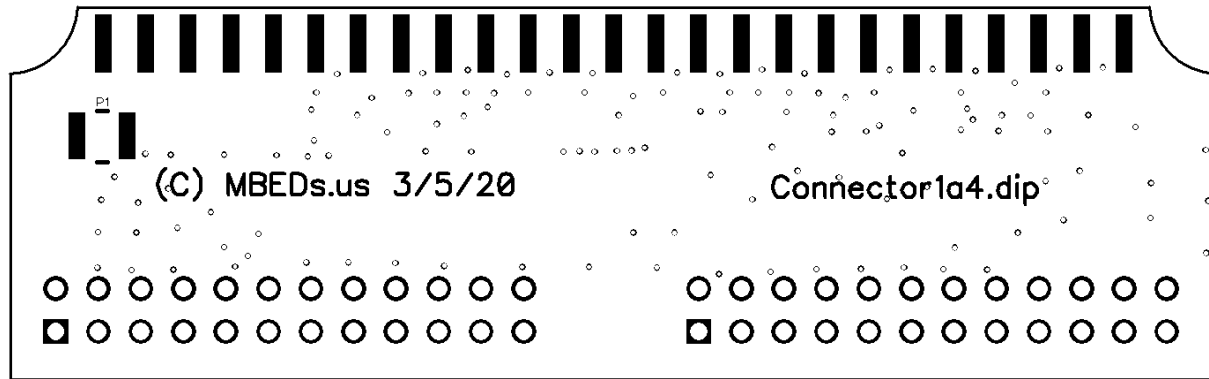
Clean with solvent such as acetone to remove flux residue.

**Build notes for Connector PCB vn1a4**

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

**Top Side**

1. Place the 16 x 100 Ohm resistors, R0 – R15
2. 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.
3. Making sure J1 is aligned well with the pads, solder one pin on each side and then look from the side to check that the connector is aligned properly before soldering the remaining pins.
4. Place J2 and J3 with the gaps facing upwards (towards J1). Solder in place.



### 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.