Atmega32u4 USB Breakout Board Assembly Guide

Written by Avery Louie

Step 0: Ignore this warning.

Release notes/known issues/readme:

This is a breakout board for the atmega32u4. It is compatible with the cdc bootloader used on the adafruit breakout board, complete with reset button to put it into programming mode. To assemble, you will need a soldering iron, solder (pretty much any kind), desoldering wick and tweezers. To debug (which I strongly recommend), you probably want a voltmeter with continuity testing. Almost any kind of tip can be used, but the finer the tip, the easier of a time you will have. The instructions are to be used by someone who may not be experienced with SMD soldering.

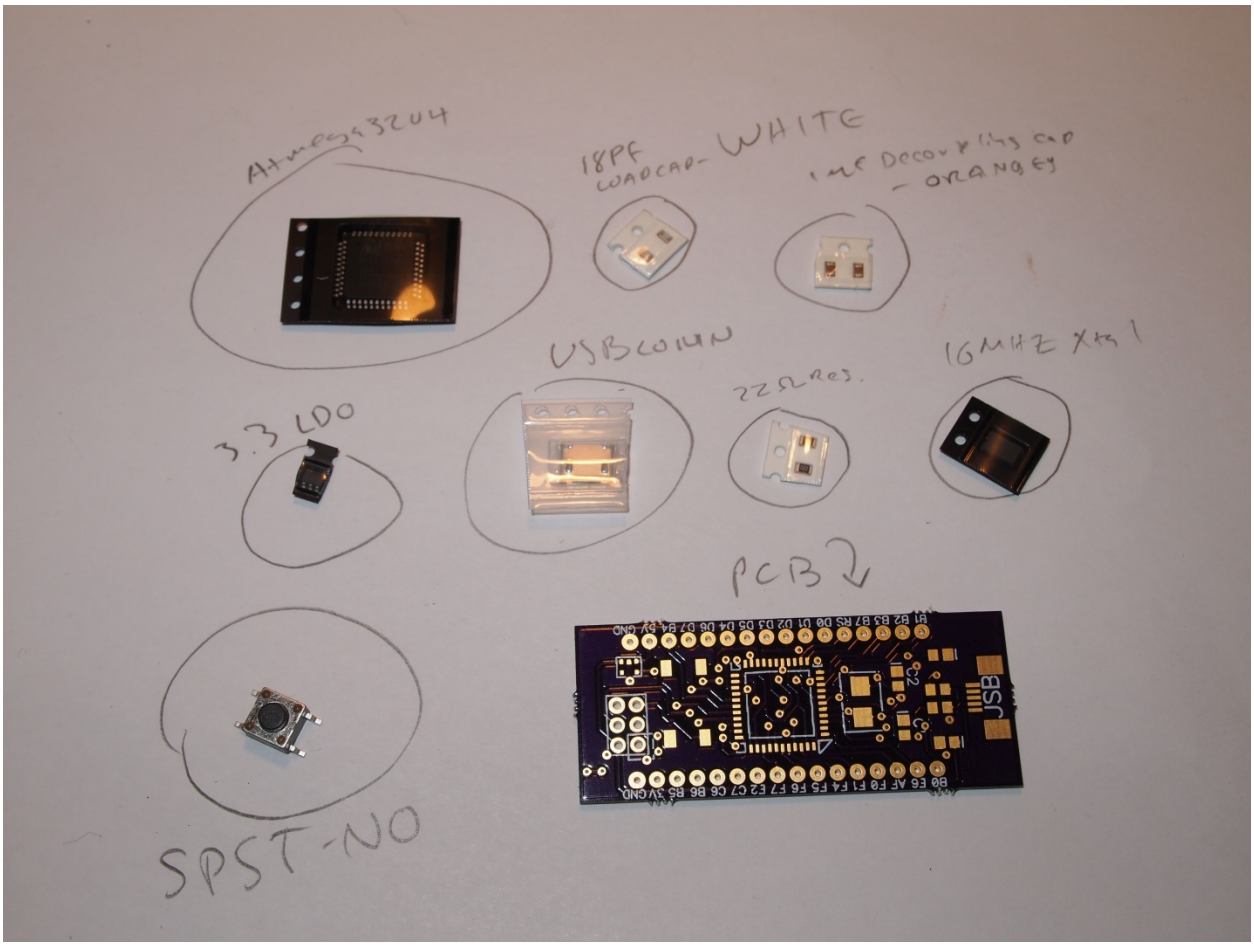
Issues:

- sot-23 footprint for LDO too small
- power and ground nets on mcu not connected (easily fixed with provided jumper wires)

Step 1: Open your envelope and find all the parts for the board. Each board needs:

1	PCB
1	Atmega32u4 qfp-44
36	header pins (probably 2 sticks of 20 pins each)
2	18 pf capacitors (0805, milky white)
2	1 uf capacitors (0805, orange-ish)
2	22 Ω resistors
1	usb connector
1	16 Mhz crystal (smd-2)
1	SPST-NO switch
1	3.3 LDO
2	jumper wires

Here is a picture of the components in their packaging (header pins not included)



2. Solder the usb connector

a. start by removing the connector from the packaging

b. tin the small usb connector pads (cover them with a thin layer of solder). The usb pads are on the edge of the board.

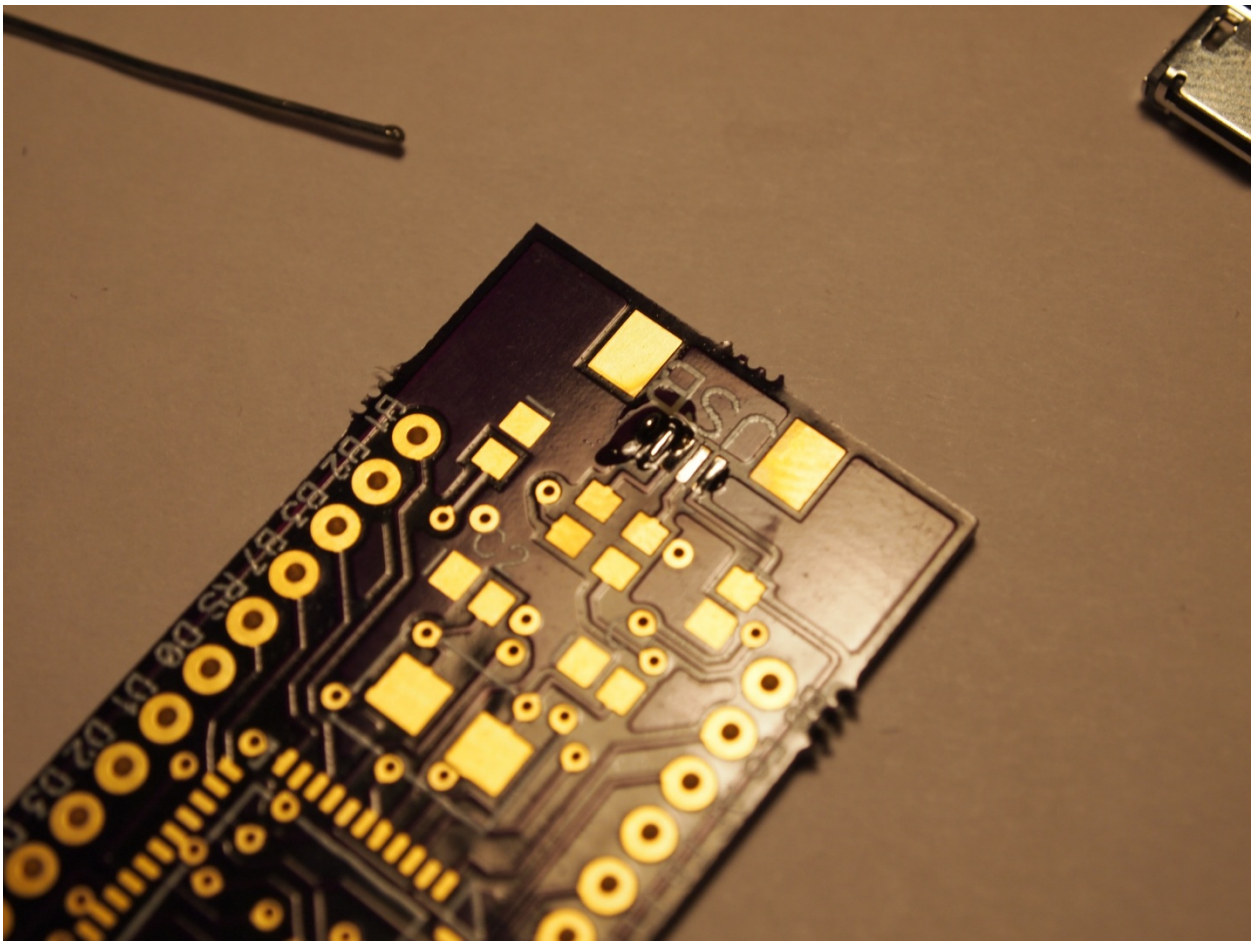


Figure : tin the pads

c. tack the usb connector onto the board with the small usb pad using **only one** of the pads.

d. adjust the connector by heating the pad and moving it. The center lead of the USB connector should be on the center of the center pad of the usb footprint. The footprint is extra wide to make bridging the pins less likely.

e. visually inspect the pins- they should line up with the pads. If this is the case, heat the pads so the solder flows onto the pins.

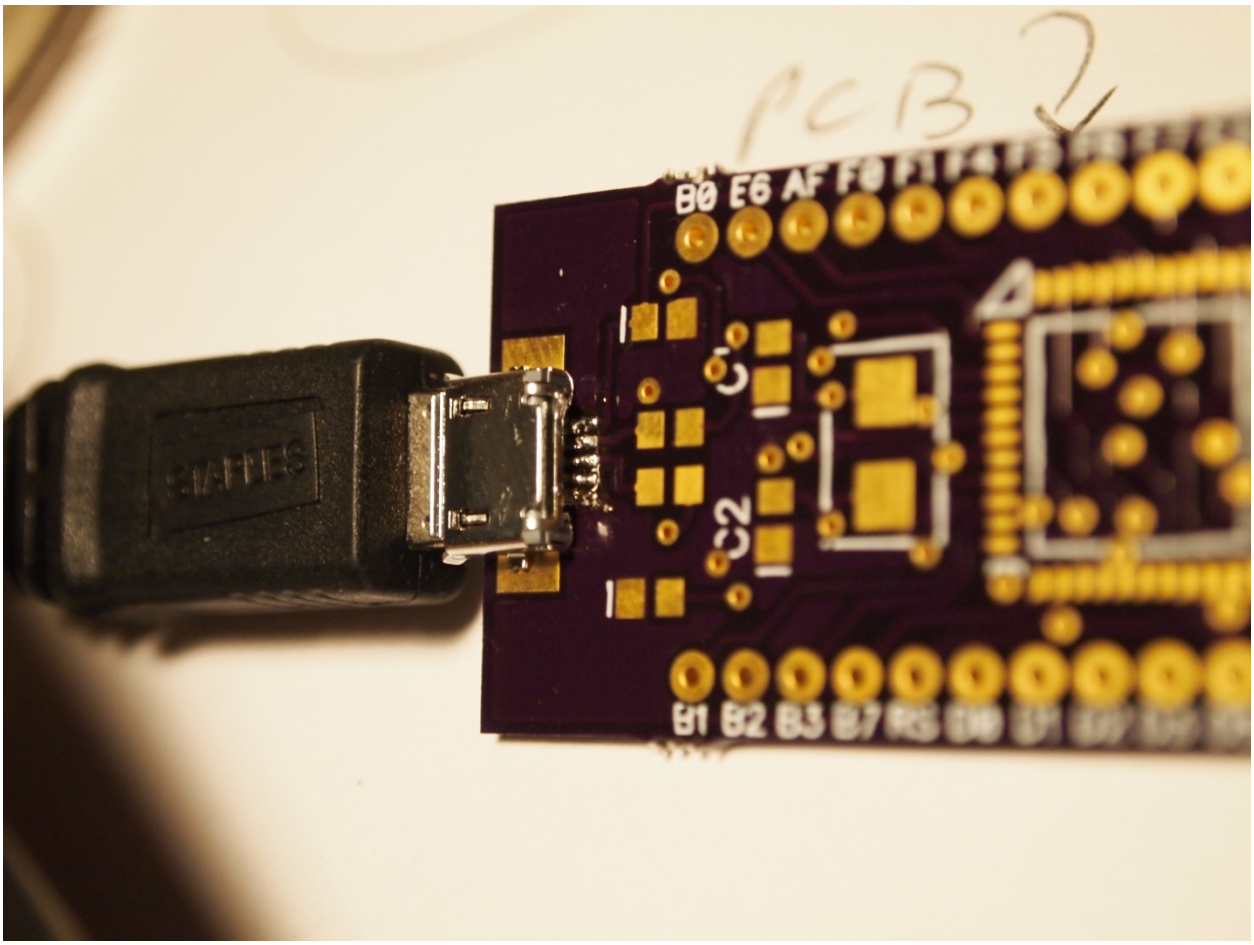


Figure : reflow the pads to the connector and test continuity

f. gently plug a usb cable into the connector and test the continuity between the usb pads and the end of the cable. The two pads closest to the usb footprint are also useful- they are the middle usb pins (D+ and D-) and the one next to B0 is a decoupling cap, so the side closest to the usb connector is V+ and the other side is ground. There should be one usb pin that is continuous with each of these pads.

g. Once the continuity is established, solder the usb connector housing tabs to the large pads. These tabs are on the sides of the connector. Be generous with the solder here.

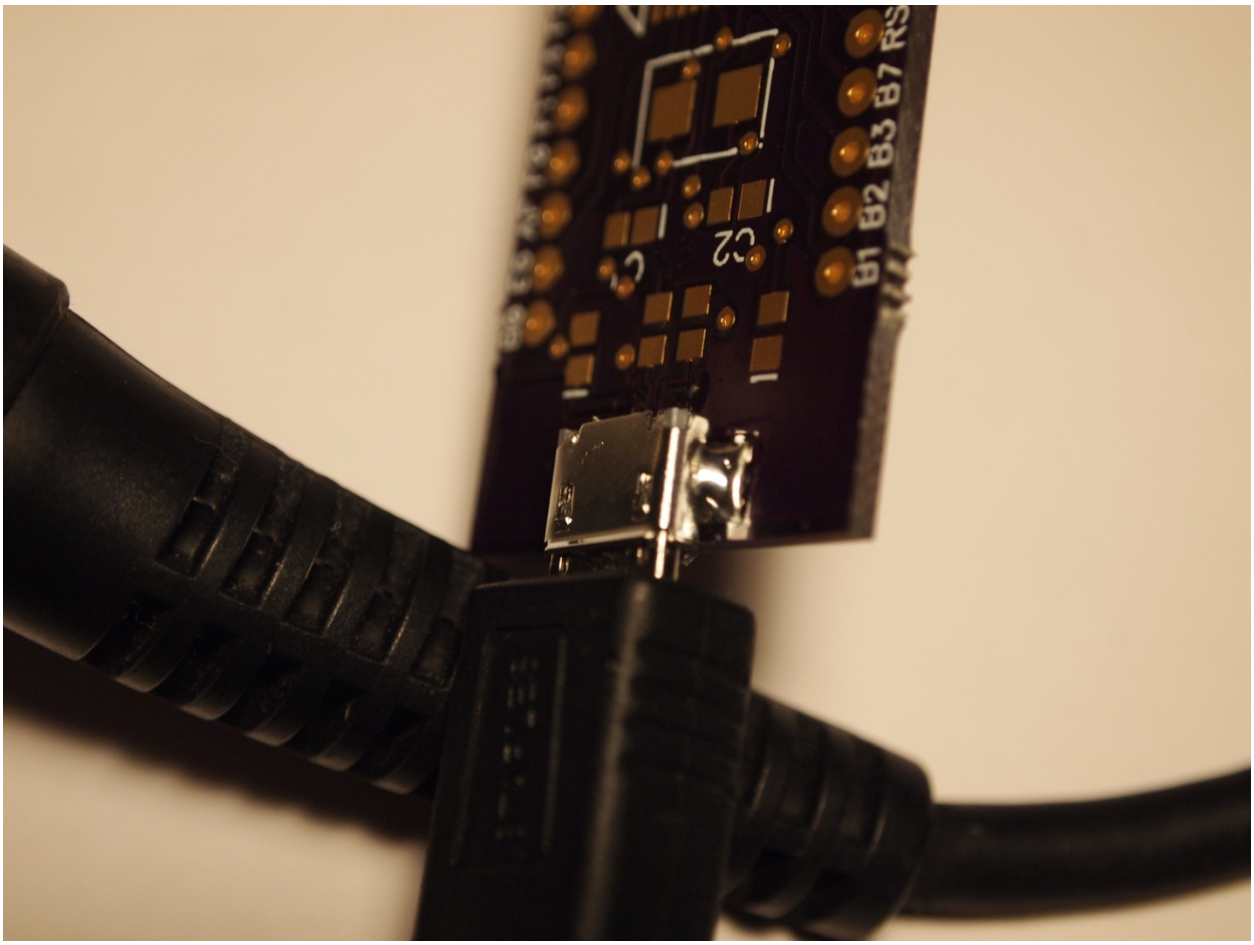


Figure : soldered tabs to the pads

h. You can now safely remove the usb cable from the connector- the large solder joints on the tabs will keep it from being pulled off the board.

3. The next step is to solder on the atmega32u4 itself. I used the drag and wick technique for this.

a. remove atmega32u4 from packaging

b. tin ONE pad on the atmega32u4 footprint

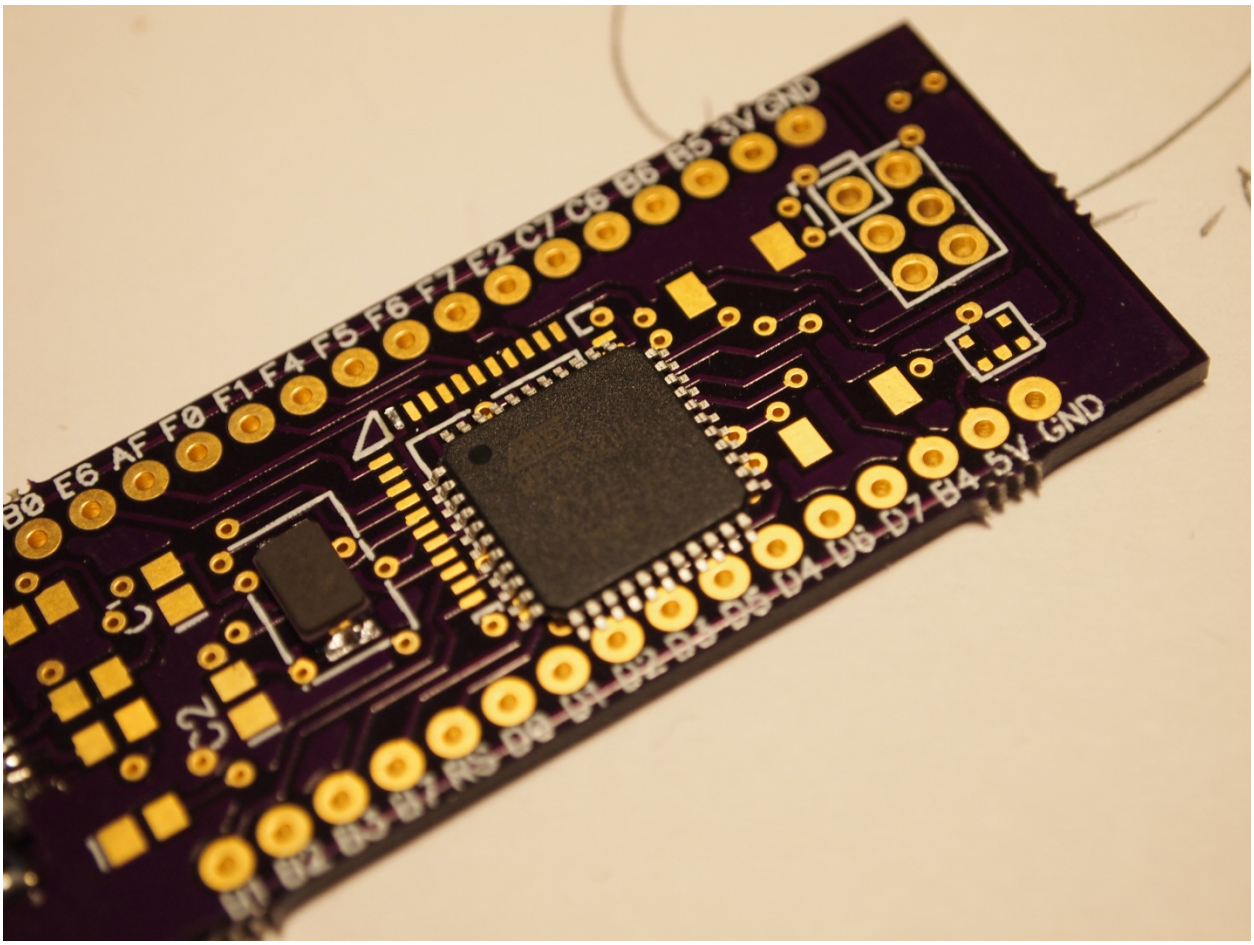


Figure : proper allignment of chip on board. one pad is tinned.

c. align circle on chip with screenprinted triangle on board. Triangle and circle can be seen in fig 4.

d. align chip over pads. Ensure that the alignment of every pin is correct. Once the alignment is correct, heat the tinned pad and tack the chip to the board.

e. cover the pins in solder! This will ensure that all the pins are connected to the board…and each other.

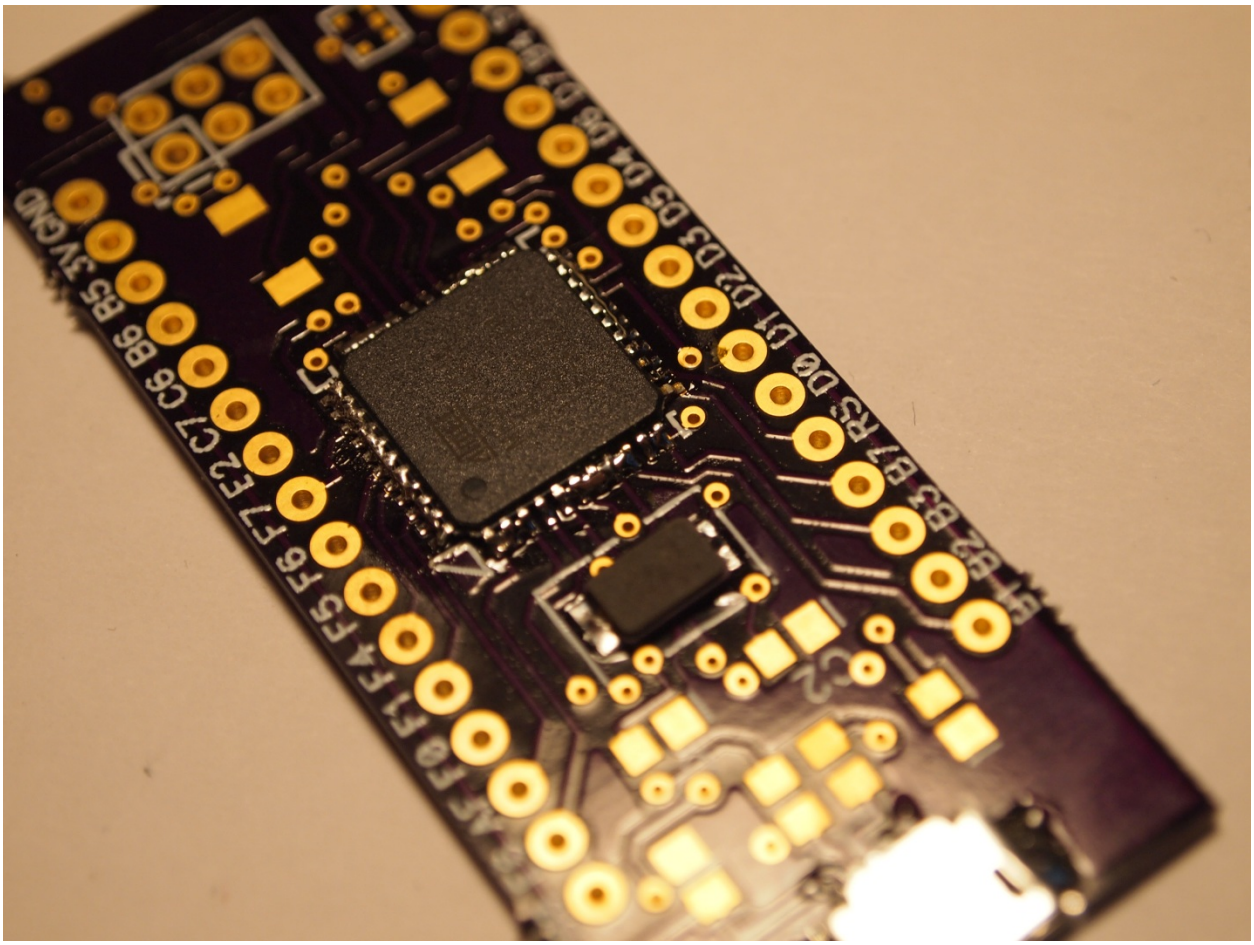


Figure : pins bridged with solder.

f. Wick up the excess solder with the desoldering wick. Just apply heat to the wick when it is positioned in the figure below. Do this for each side.

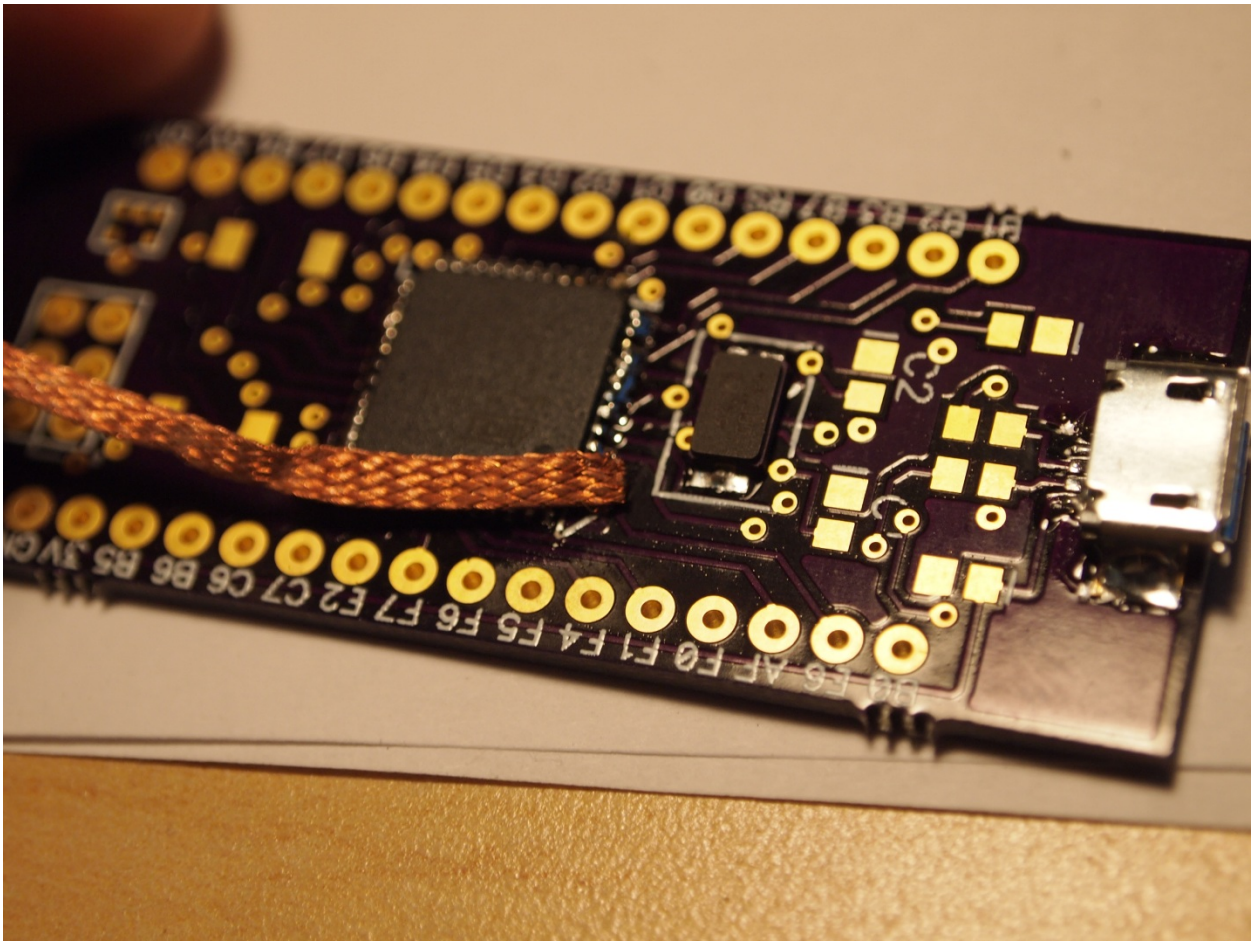


Figure : how to position the solder wick to remove solder from bridged pins

g. inspect visually to ensure that no pins are bridged together. If they are, wick up the bridge.

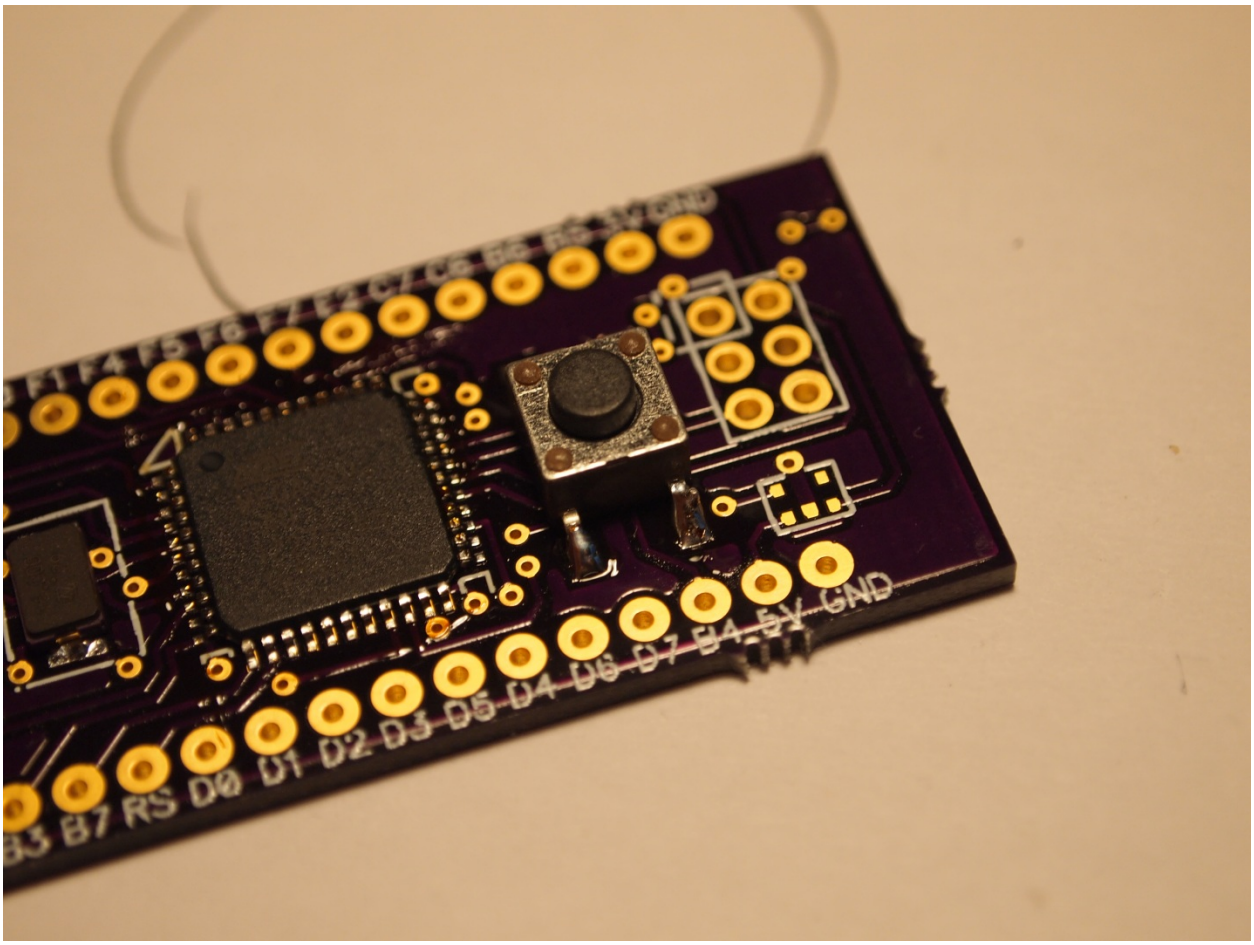


Figure : pins should look like this!

4. solder crystal to board

a. remove crystal from its package

b. tin one of the pads for the crystal. The crystal is connected to the two large pads between the atmega and the usb connector.

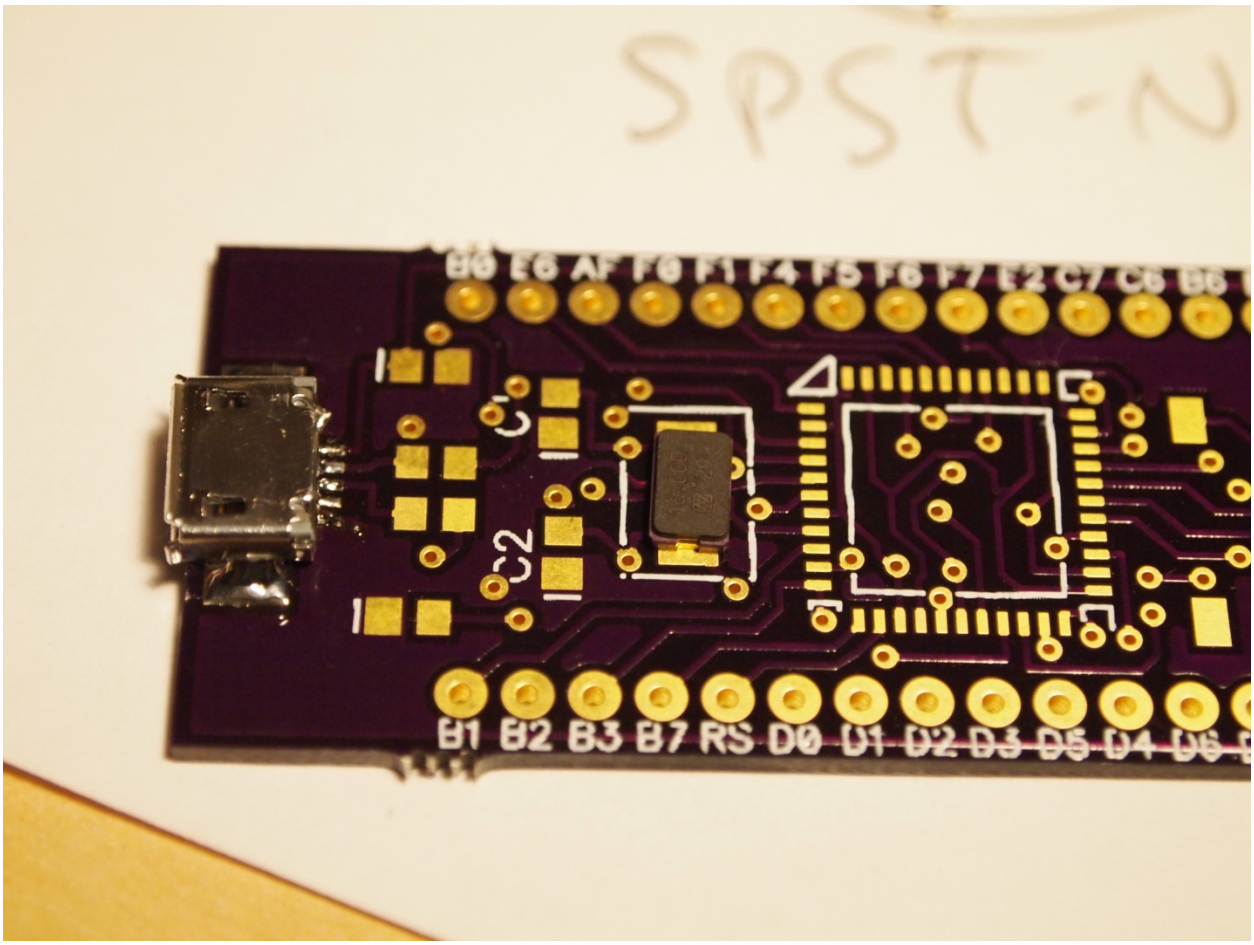


Figure : position of crystal

c. place crystal and tack one side on, then melt solder on to the other pin.

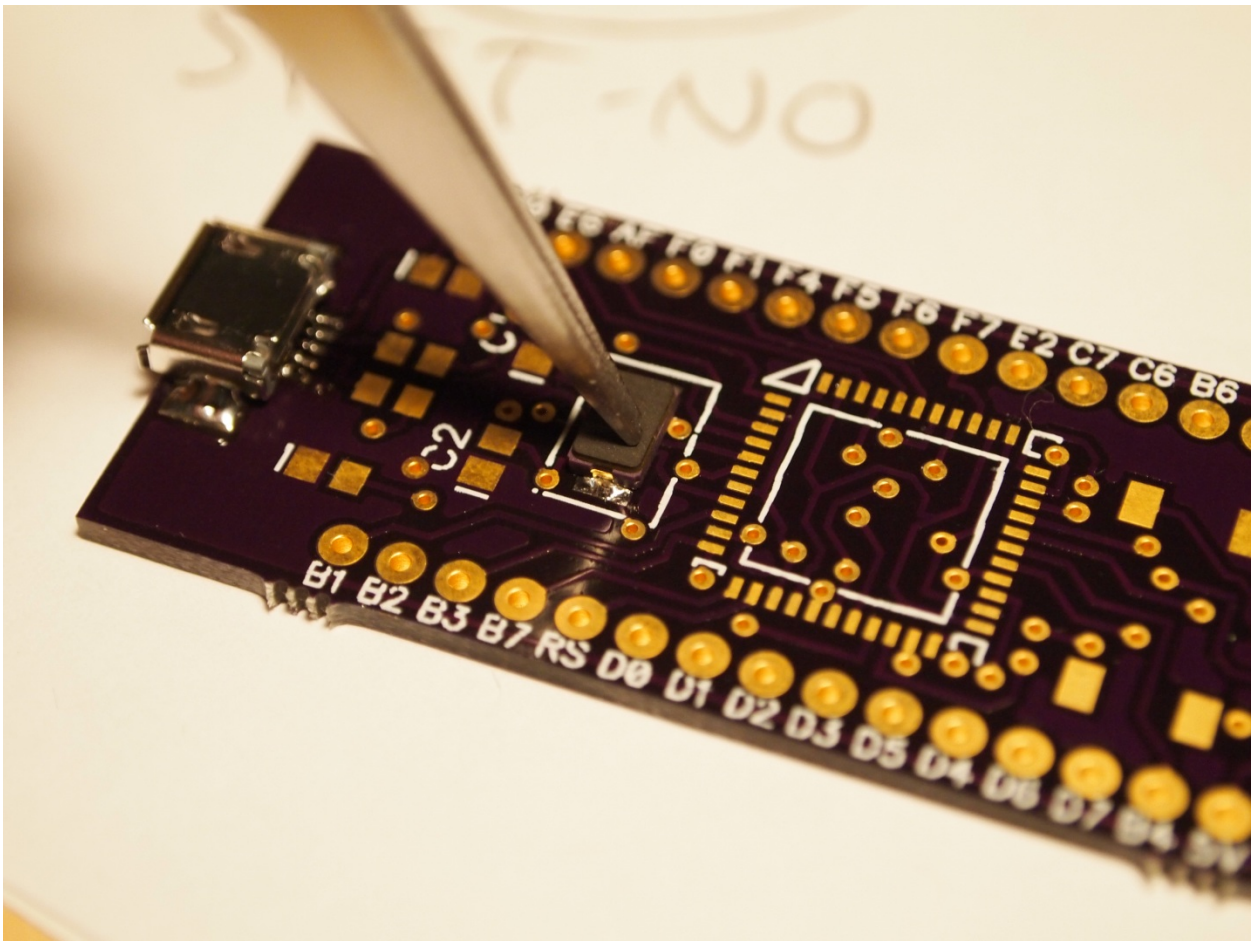


Figure : solder down both sides.

5. now we will solder on the reset button.

a. tin one of the pads for the button

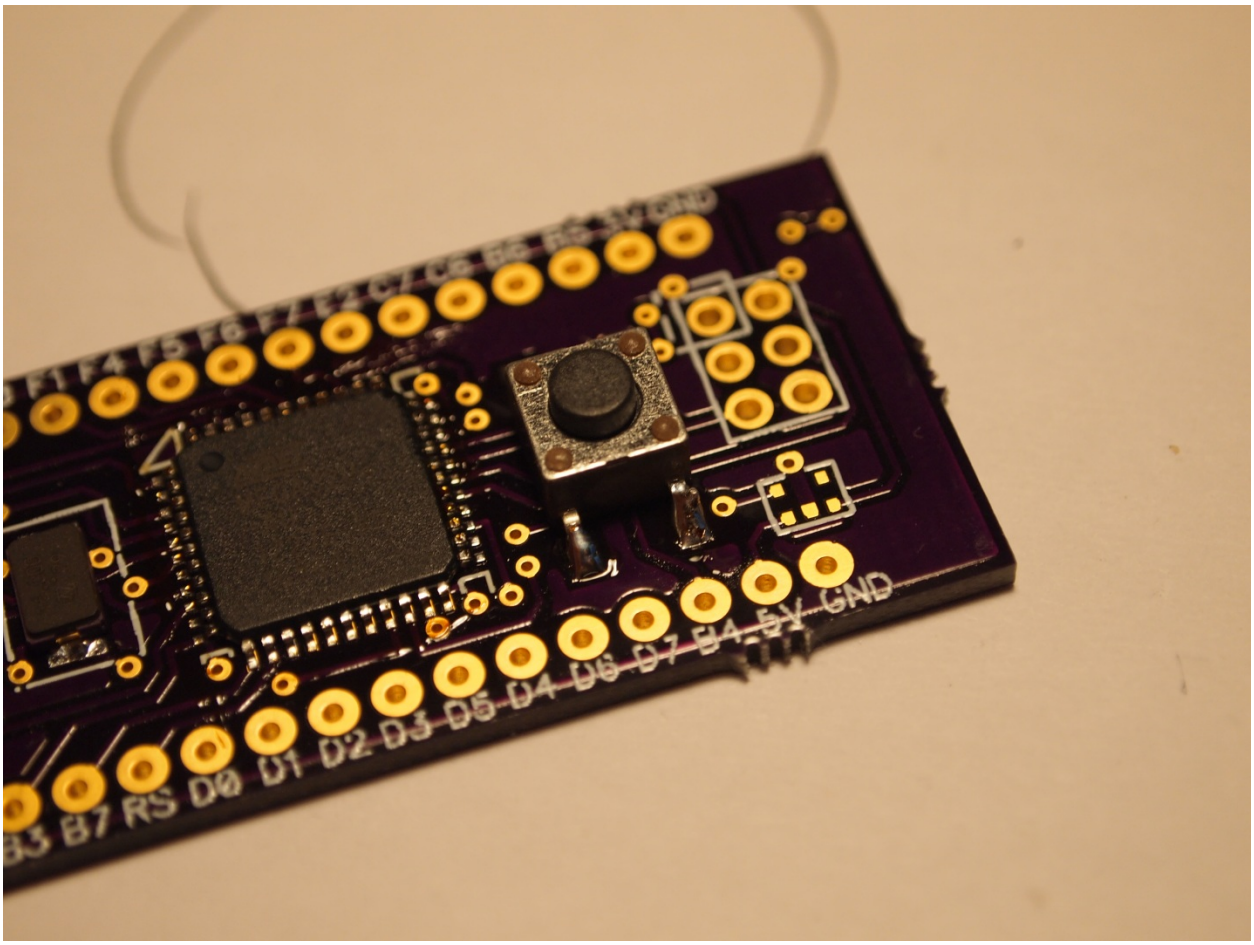


Figure : button position

b. tack the button on and position it

c. solder the rest of the pins to the board

6. Now we will solder on the resistors and capacitors. It is best to do each pair of components one at a time to avoid confusion. Also, note which hand you solder with when tinning pads. I solder with my right hand, so I tin the right hand or bottom pad for whatever orientation the board is in. This way, I can manipulate parts with my left hand when I tack them down without having to move the board around.

a. tin one pad of each of the 6 0805 pads between the usb connector and the atmega

b. Identify the 1 uf capacitors by their orange color. Remove them from the packaging, there should be 2. Solder them as shown in the picture below.

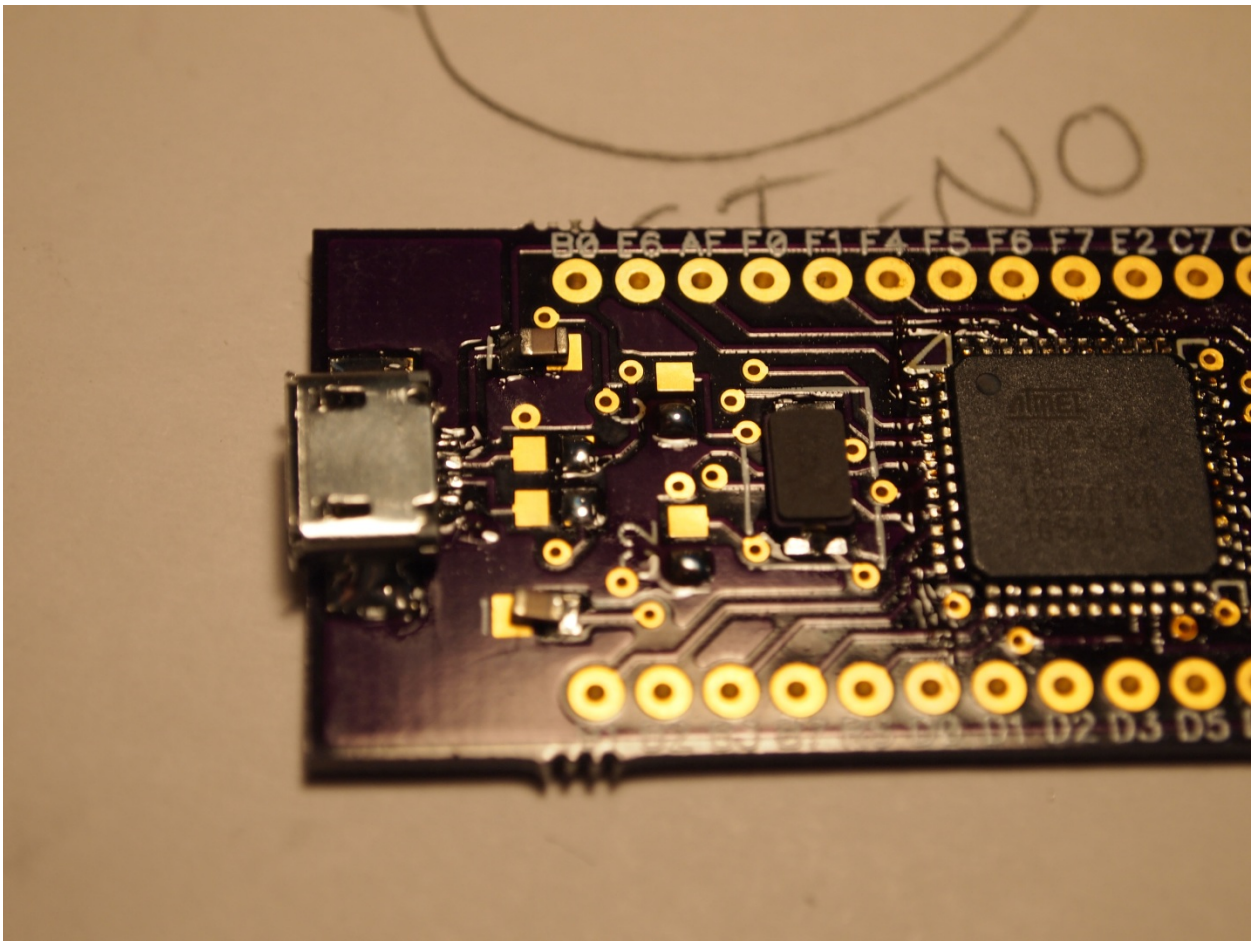
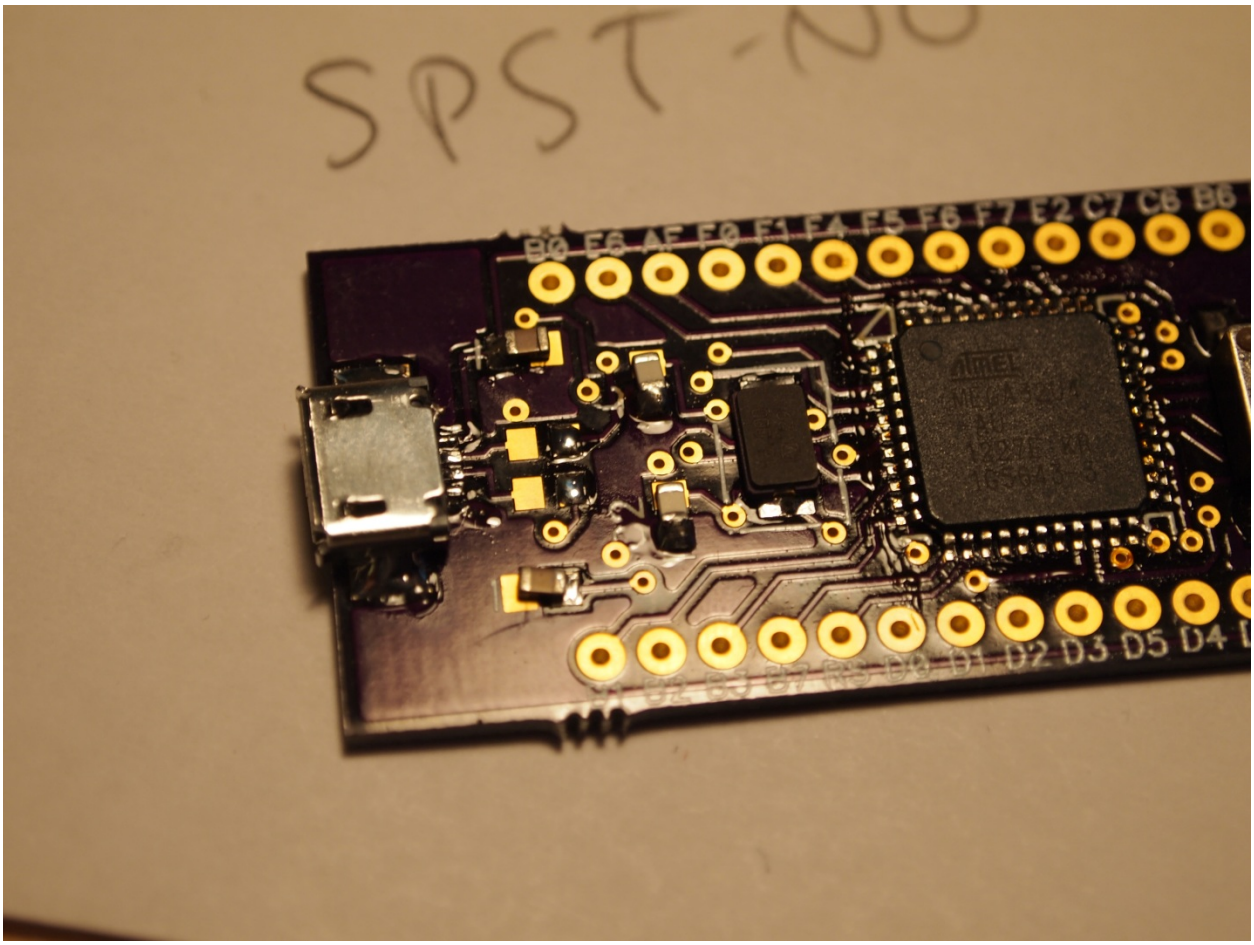
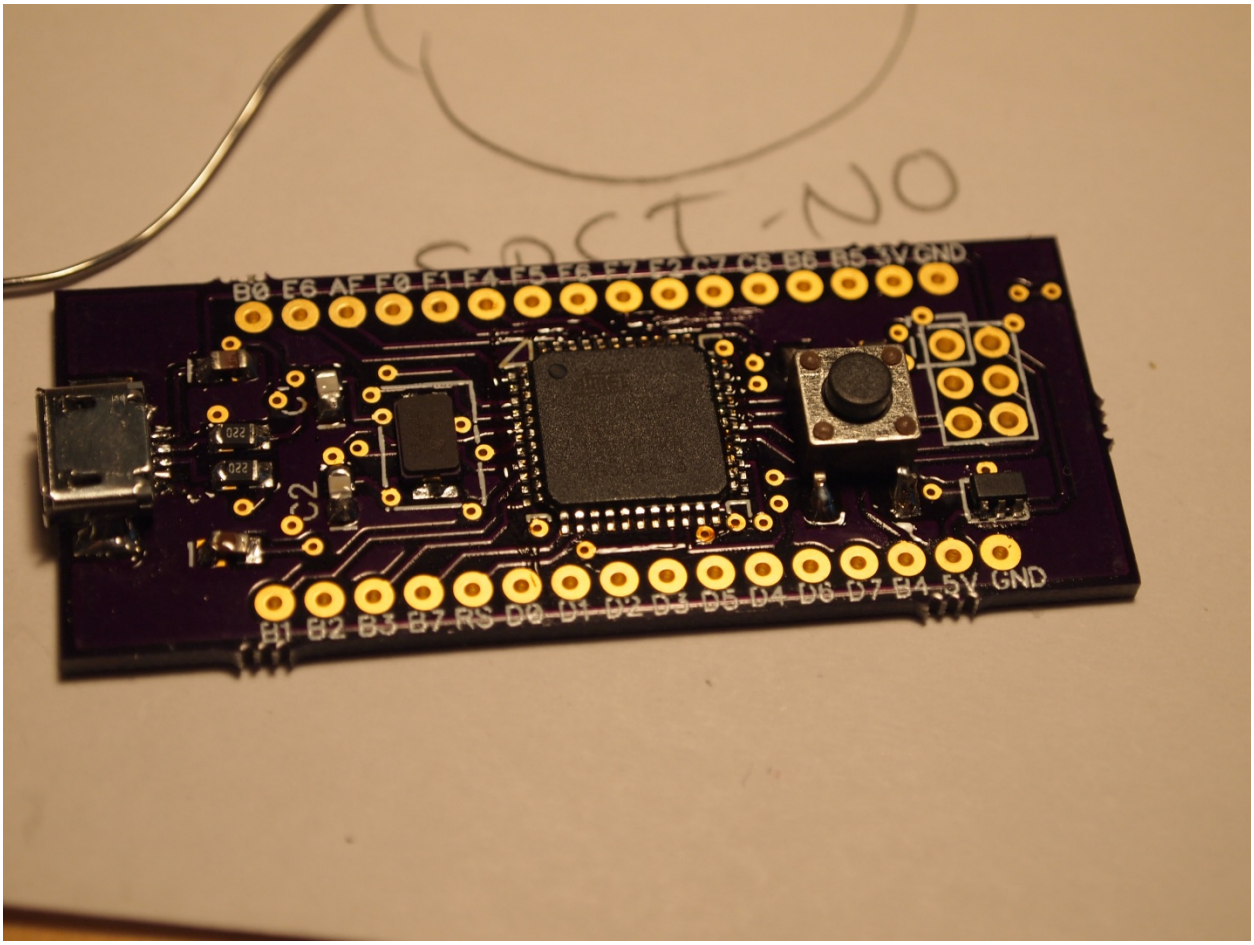


Figure : tin the pads and tack the first two capacitors

c. remove and tack on the two milky white 18 pf capacitors as shown in the picture below.

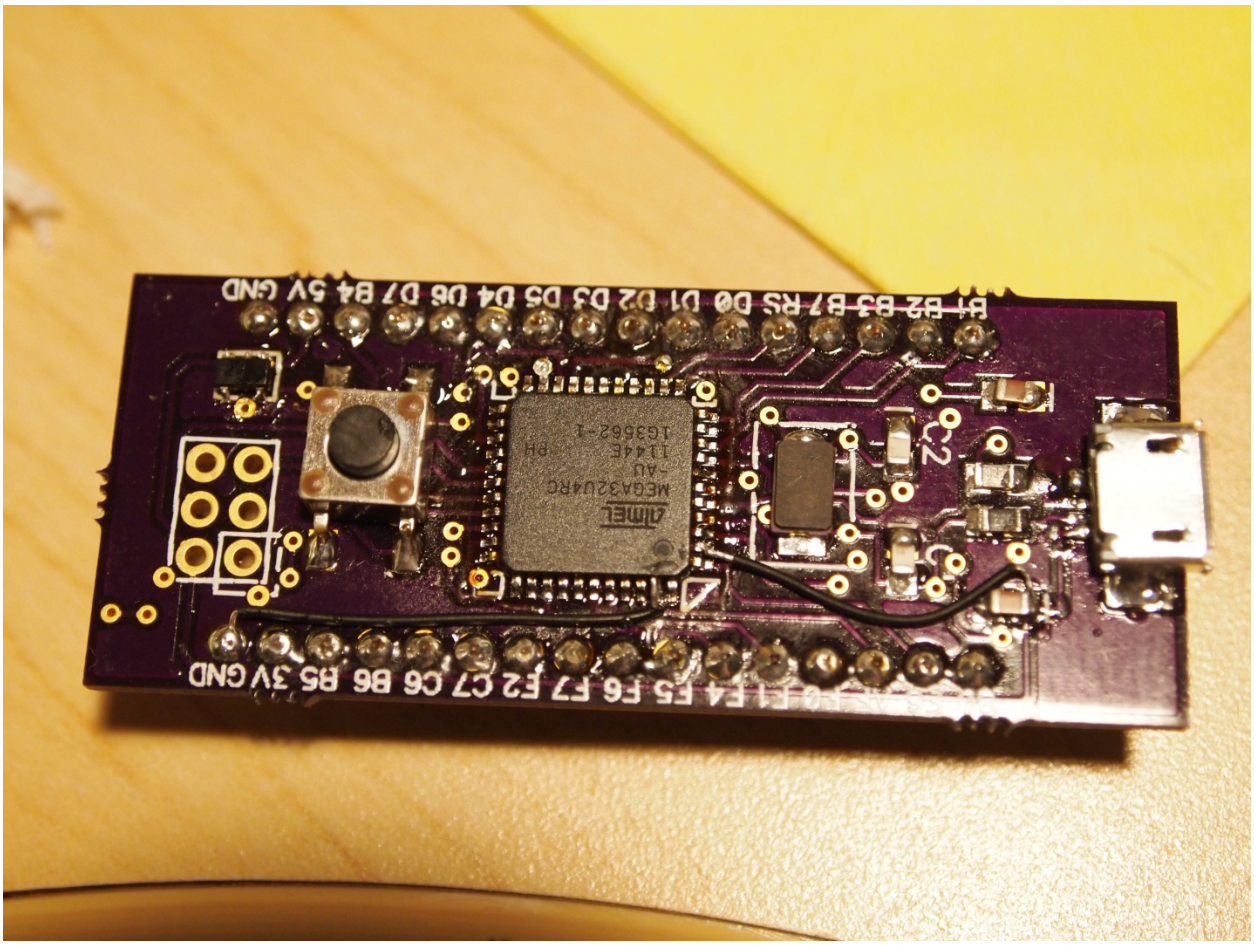

SPST-NO
B0 F6 AF F0 F1 F4 F5 F6 F7 E2 C7 C6 B6
D2 D3 D5 D4

d. tack on the two resistors (black on one side, with white letters) onto the remaining pads as shown in the picture below.

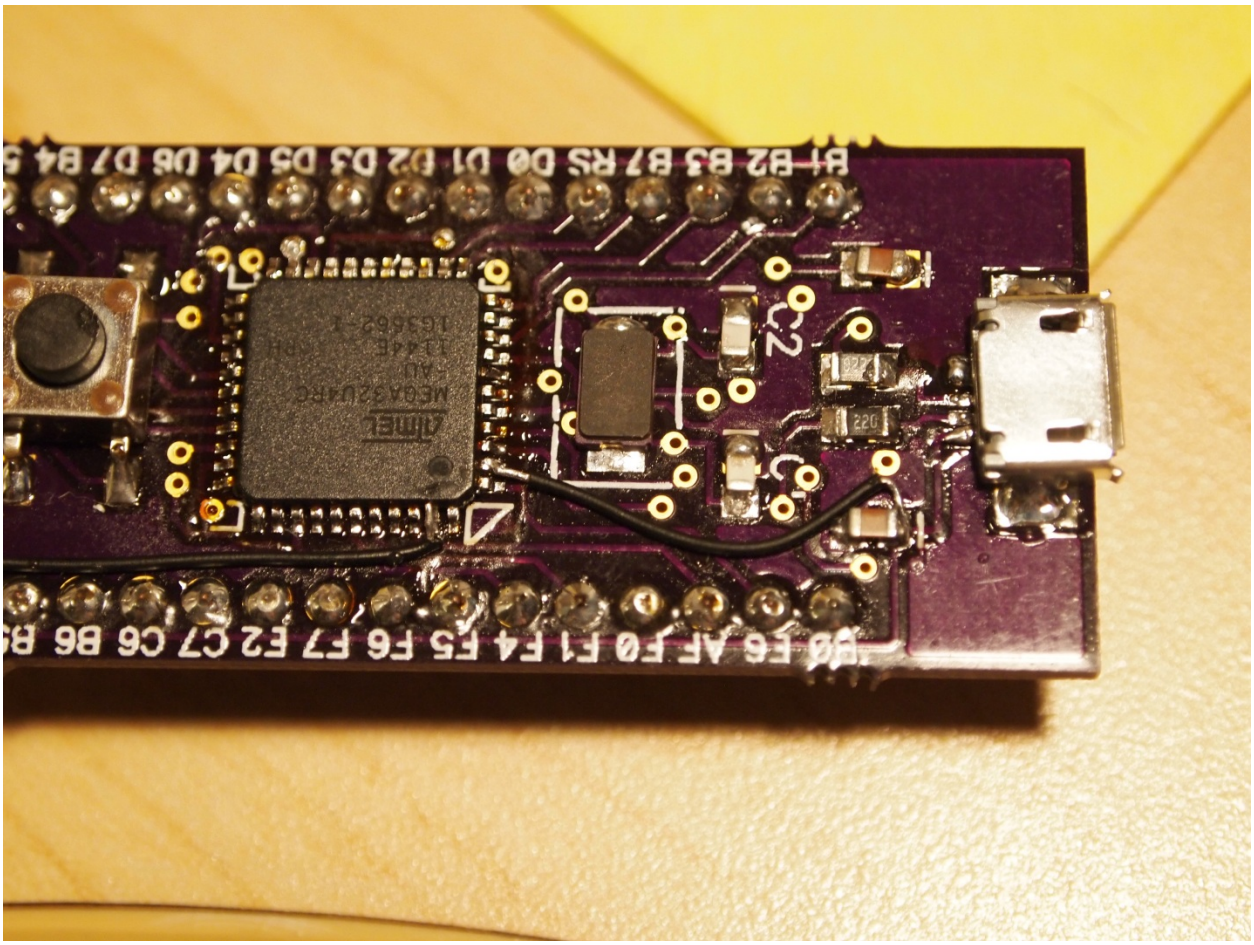


e. make sure all the parts are properly aligned, and then solder the side that was not tacked to the board.

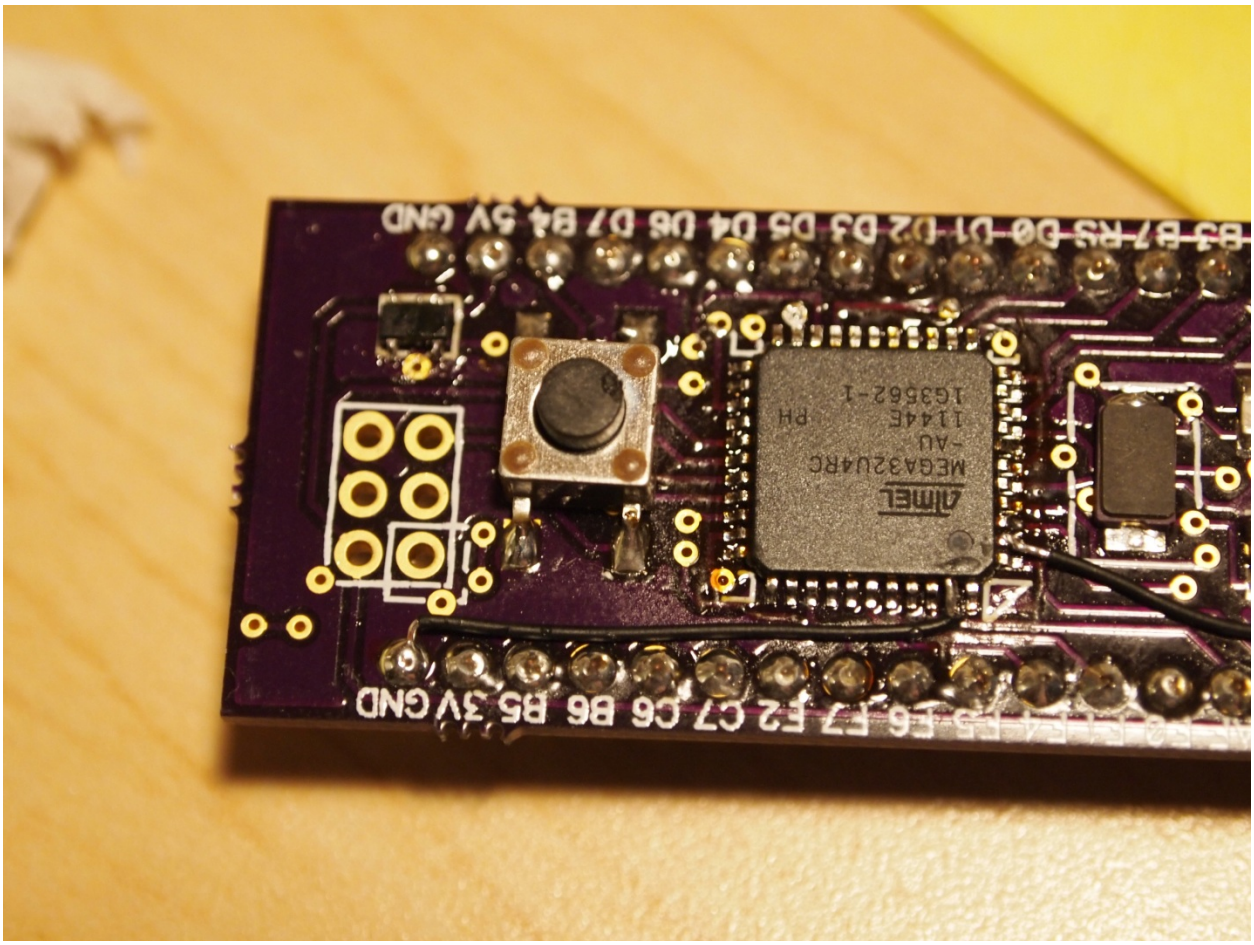
7. solder the jumper wires. The power and ground connections on the chip were accidentally not routed. In order to fix this, two wires need to be soldered onto pins on the chip, and the board. These should be rather durable- mine have not fallen off after a few days of living in my backpack. The secret is to keep them low to the board, as I this picture:



Here is a closeup of the power wire. It goes from pin 2 to the side of the decoupling capacitor nearest the usb connector:



And one of the ground wire, which goes from pin 43 to the ground header pin on the breakout:



Optional step: solder in the header pins. This can be done many ways, so that the breakout can be breadboarded, or connected to wires, or you could even get female headers like the arduino has and use those!

a. Break pins off to correct length (should be 16 pins long).

b. Place pins in breadboard correct width apart (pcb and breadboard should be perpendicular to pins)

c. Solder the tops of the pins to the board.

The board is now completely assembled! The sot-23 pad was much too small, so the voltage regulator will not fit. The ISP header can be used to program the board.

Schematic of the actual traces on the board: