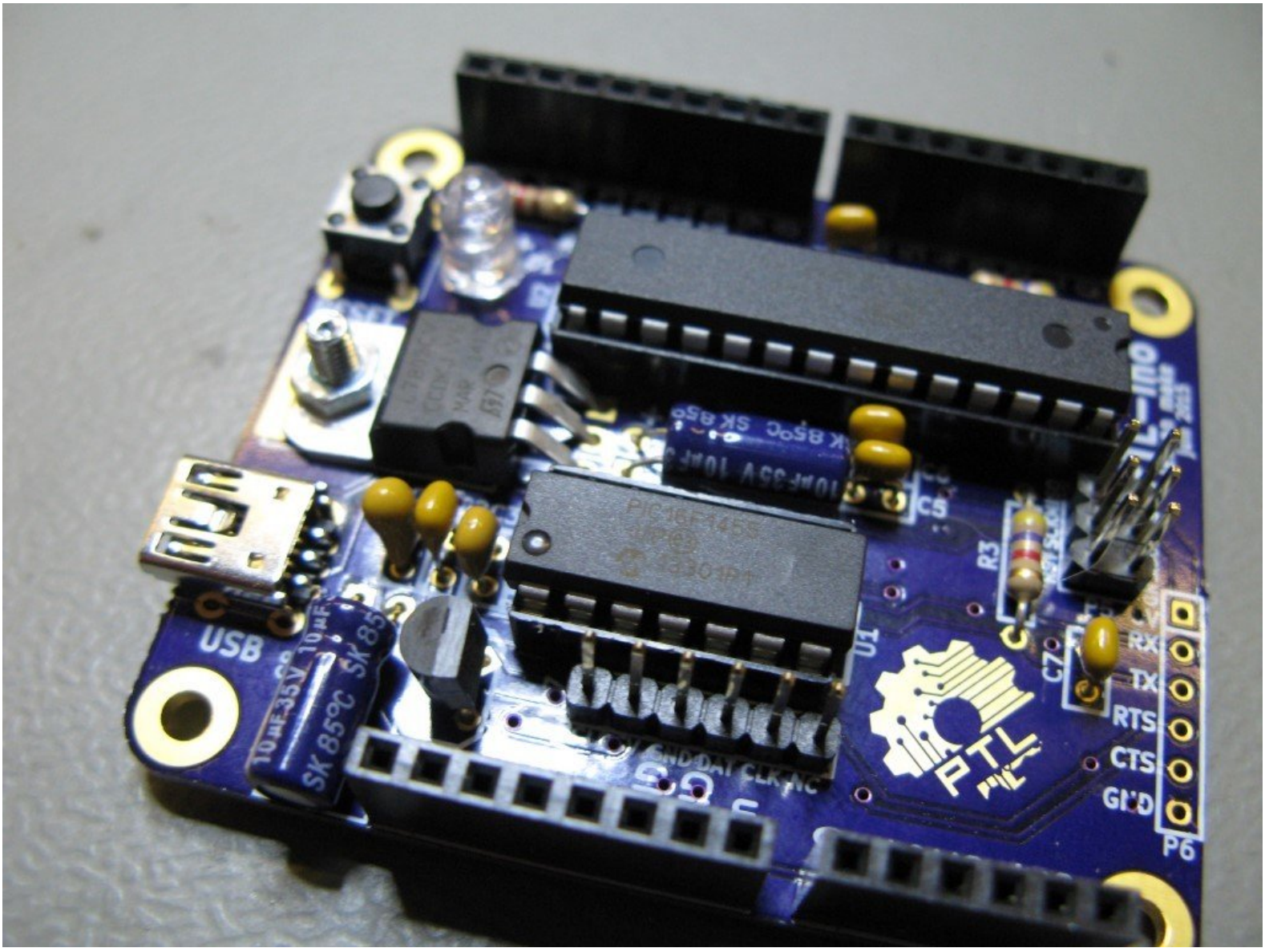
Post Tenebras Lab

How to build a PTL-ino board

PTL-ino is an Arduino comptaible board, made entirely out of through-hole components. It is a perfect project to learn how to solder and start getting into the world of micro controllers.

Written By: Post Tenebras Lab



This document was generated on 2021-12-21 06:57:19 PM (MST).

INTRODUCTION

For more infos on the PTL-ino check out: http://PTL-ino.com

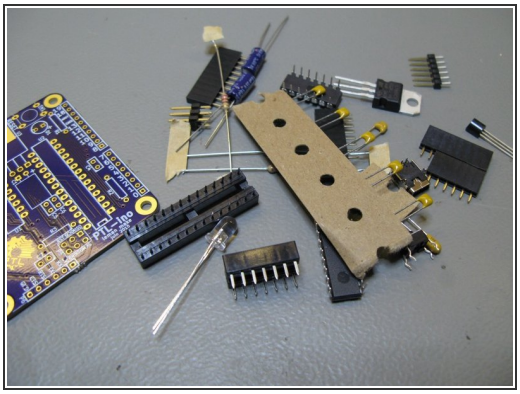
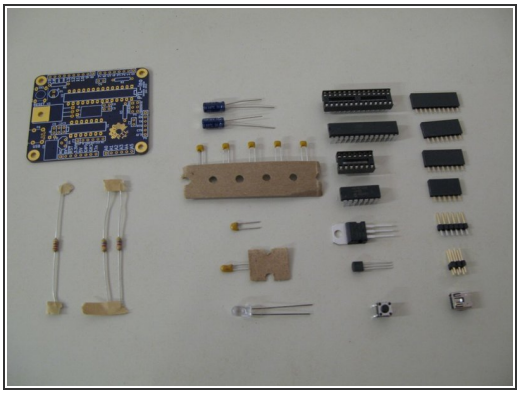
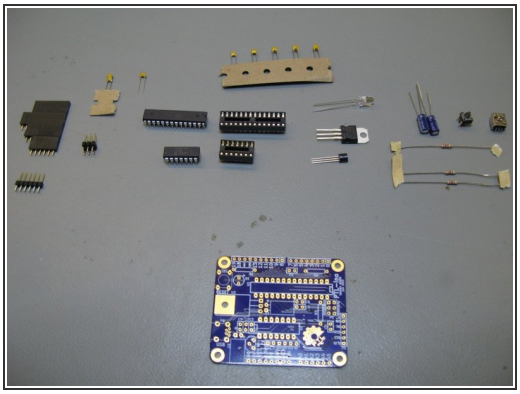
TOOLS:

- Soldering iron (1)
- Solder (1)
- Wire cutter (1)
- Multimeter (1)

PARTS:

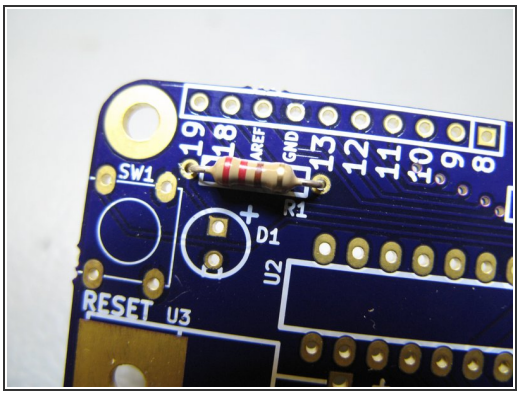
- PIC 16F1455 (U1) (1)
- 2×7 2.54 pin IC socket (for U1) (1)
- AVR ATmega328P (U2) (1)
- 2×14 2.54mm pin IC socket (for U2) (1)
- LM7805 (U3) (1)
- MCP1700 (U4) (1)
- LED (D1) (1)
  5mm, any colour
- capacitor 10uF (C1, C8) (2)
  electrolytic
- capacitor 10uF (C9) (1)
  tantale
- capacitor 100nF (C2, C4, C5, C6, C7) (5)
  ceramic
- capacitor 220nF (C3) (1)
  ceramic
- resistor 220Ω (R1) (1)
  controlles LED current/brightness
- resistor 4.7kΩ (R2, R3) (2)
- 1×8 2.54mm female connector (P1, P4) (2)
- 1×6 2.54mm female connector (P2) (1)
- 1×10 2.54mm female connector (P3) (1)
- 2×3 2.54mm male connector (P5) (1)
- 1×6 2.54mm male connector (P7) (1)
- mini USB female (P8) (1)
  trough-hole version with 4 through-hole mounting pins
- SWITCH (SW1) (1)
- Cable mini-USB (1)
- PCB (1)

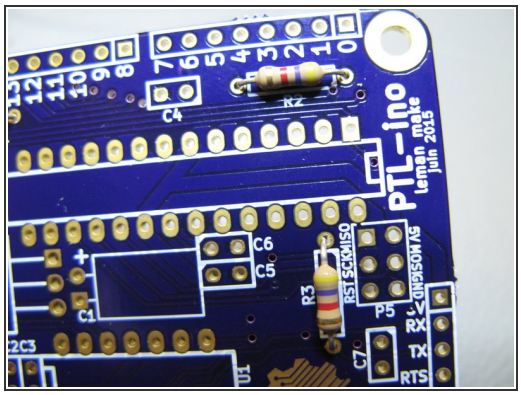
Step 1 — Get all the components ready and lined up

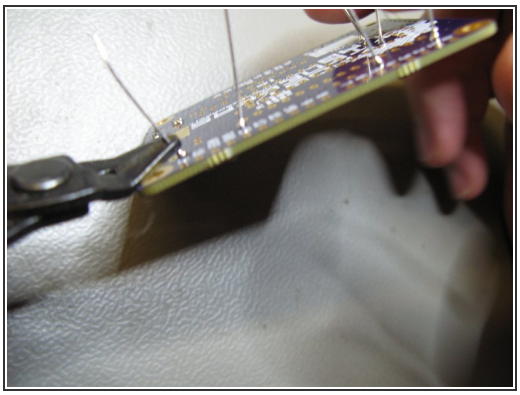




- Check that the kit contains all the components.
- Arrange the components so you can easily find them.

Step 2 — Populate the Resistors

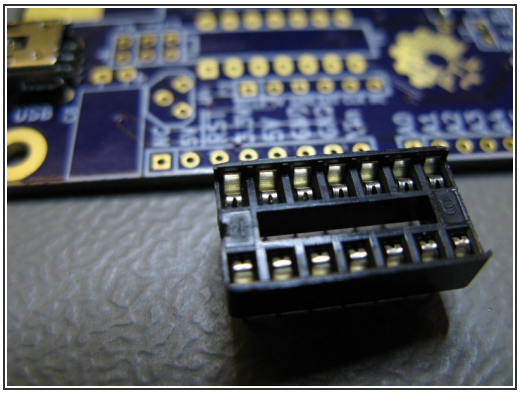
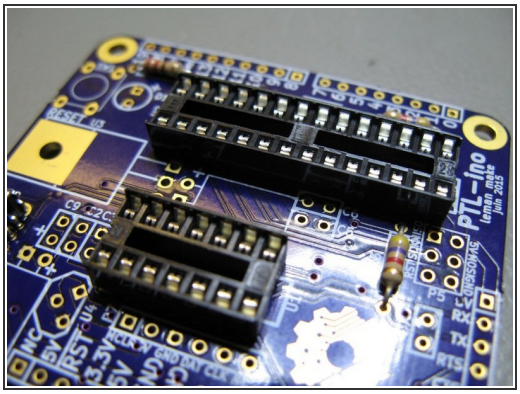


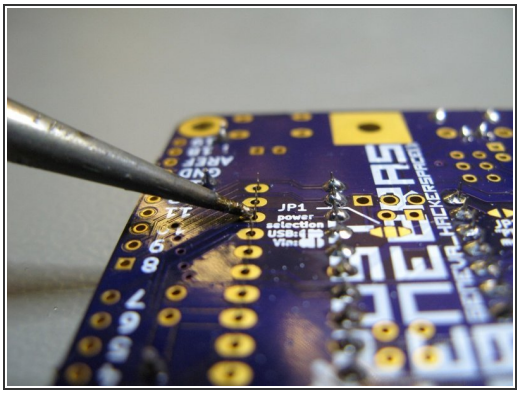




- Start with resistor R1. It is the one with red-red-brown-gold rings. Orientation is not important.
- Then R2 and R3. They are the one with yellow-violet-red-gold rings and once again, the orientation is not important.
- After placing the 3 resistors, turn the board over to solder them in place.
- Then, cut off the extra bits of wire sticking out of the PCB (shorten the "legs" of the resistors).
  - **TIP:** To keep the components in place when you flip the board to do the soldering, you can bend the wires in 45 degrees.
  - TIP: When cutting extra wires, make sure you either wear a protective goggle or that you hold the extra pice with your other hand.

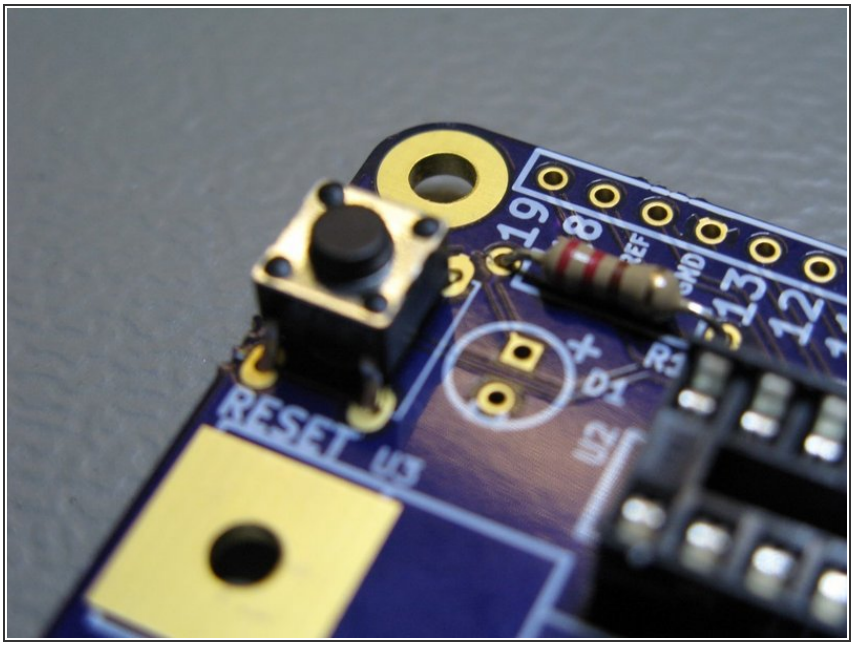
Step 3 — Populate the IC sockets



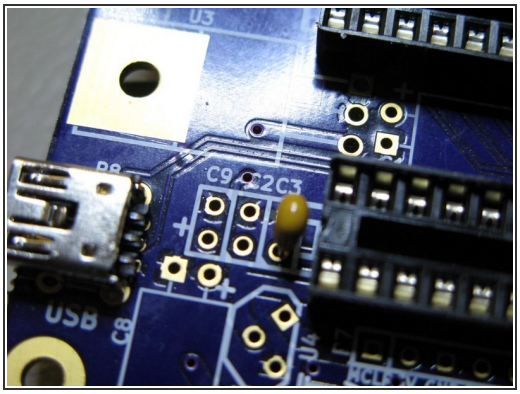
- Populate the 2 IC sockets. They have a small notch, that should be on the same side as the small markings on the PCB.
- Turn over the board and solder the 2 IC sockets. You can use masking tape to keep the sockets in place when you turn the board.
- The sockets fit perfectly and there is no need to cut any extra 'wires'

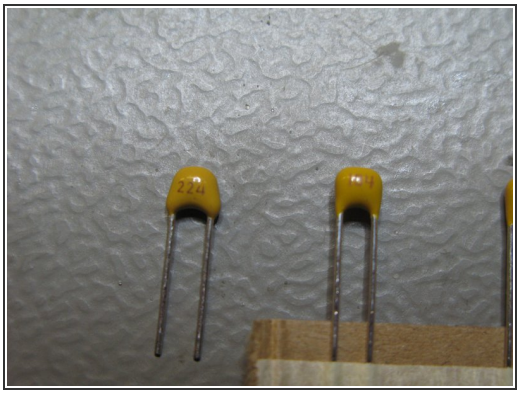
Step 4 — Populate the Reset switch

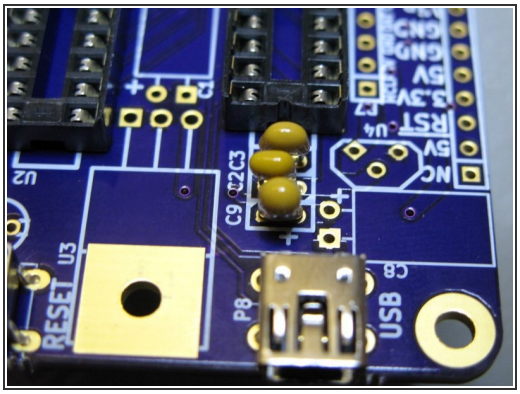


- insert the switch and solder it.
- Orientation of the switch does not matter.
- Note that you will have to force it a little bit to make it a tight fit. Also, it is normal to have a small space between the board and the switch.

Step 5 — Populate the capacitors

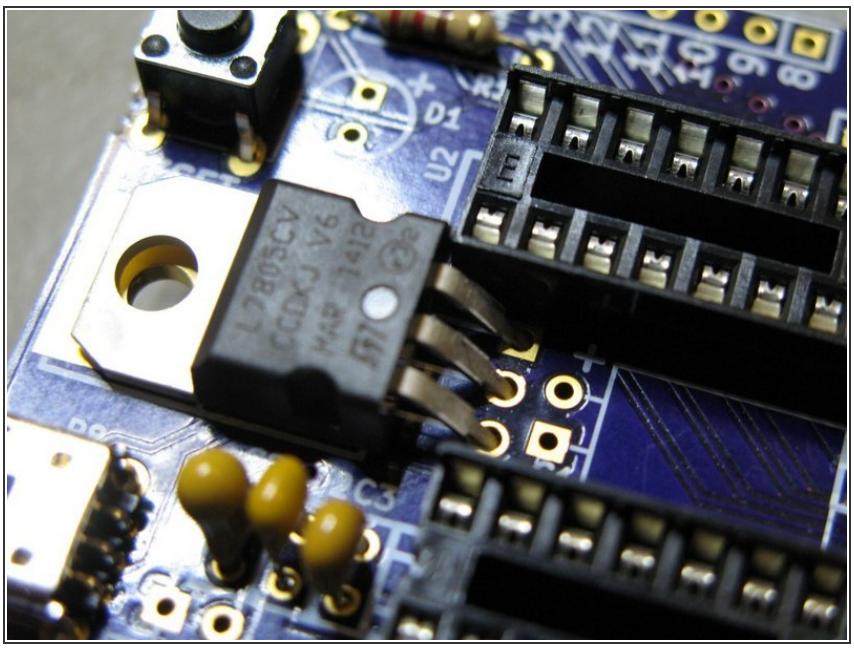






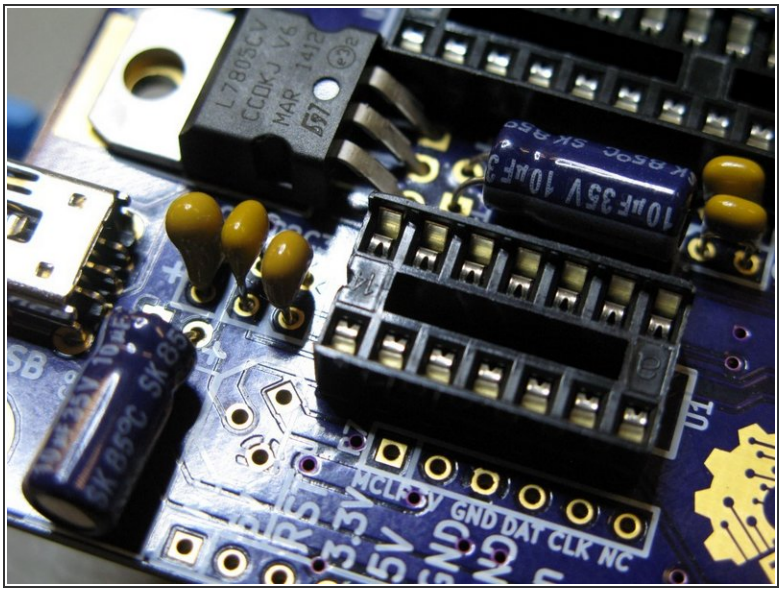
- Put C3 in place. C3 is the 220nF with 224 written on it. Polarity is not important for this component.
- Put C2, C4, C5, C6 and C7. These are all the same, and are 100nF with 104 written on it. Polarity is not important for these components.
- C9 is the tantalum capacitor, and ***the polarity IS important***. You have to match the side with the +sign on the capacitor, with the +sign on the PCB.

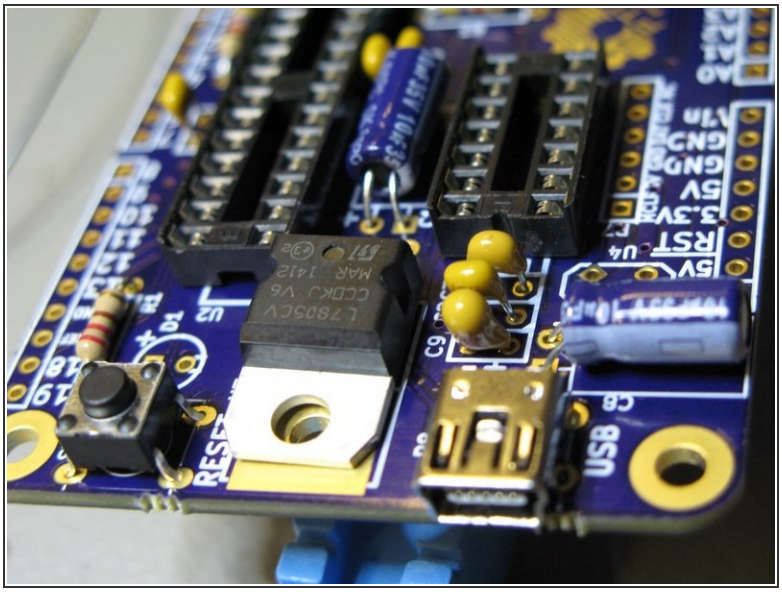
Step 6 — Populate the 5V regulator



- Insert the 5V regulator (U3), and align the hole on the PCB with the hole on the component.
- You can cut the extra leads

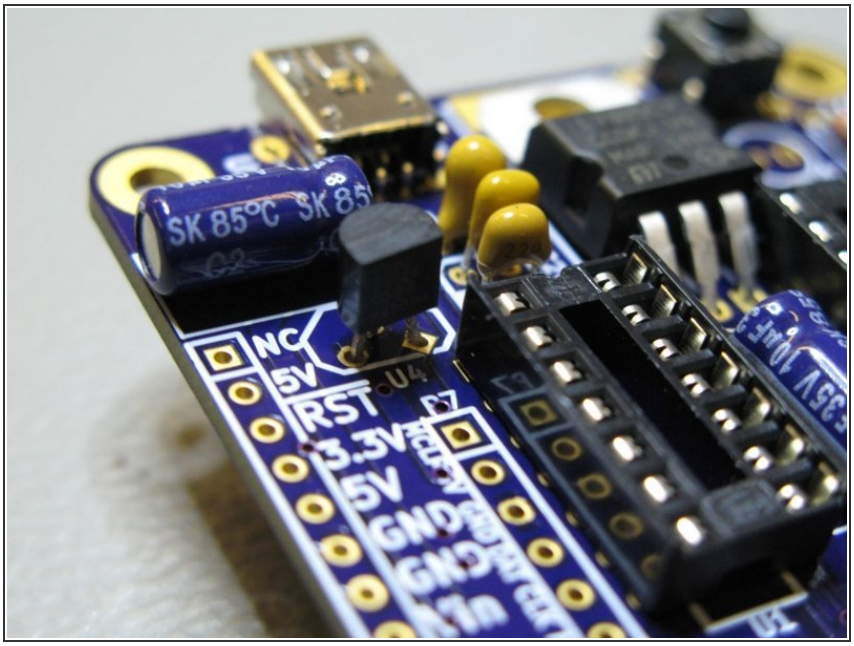
Step 7 — Populate the electrolytic capacitors





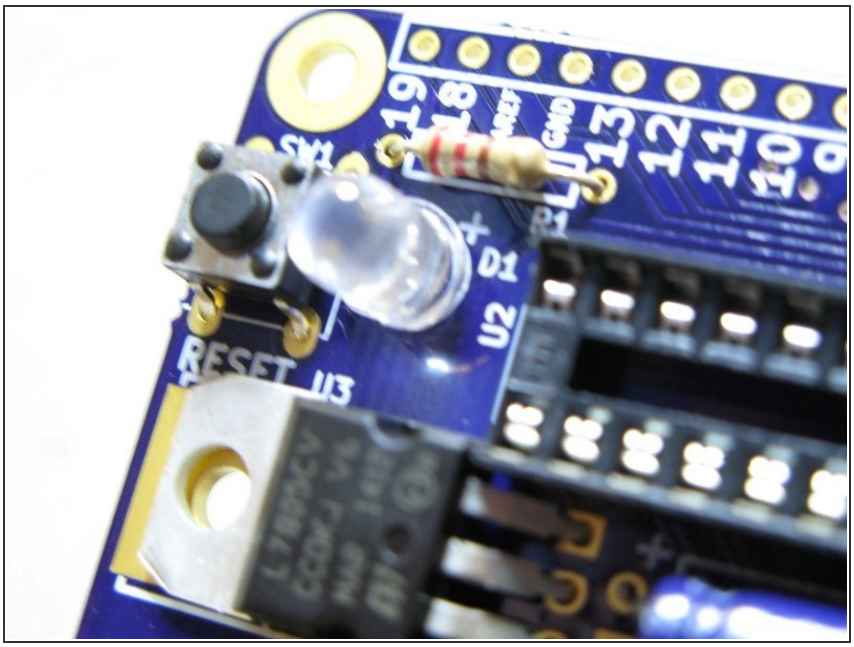
- Put the 2 electrolytic capacitors (C1 and C8) in place. Bend them so they lie on the PCB as shown in the picture.
- On the PCB there is a +sign, and one side of the capacitor has a stripe with a -sign. The -sign should, of course, be on the opposite side to the +sign.

Step 8 — Populate the 3.3V regulator



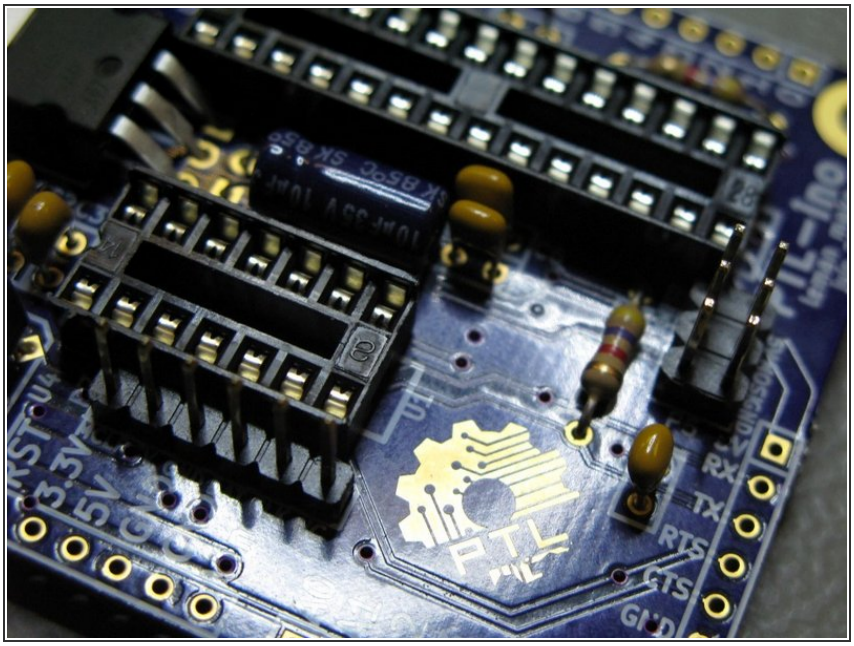
- Insert the LM1700 3.3V regulator (U4). The marking on the PCB matches the shape of the component; so that the flat part of the regulator matches the straight line on the board.
- You will have to bend the middle lead a little bit in order for it to fit.
- You can cut the extra leads when done with soldering.

Step 9 — Populate the LED



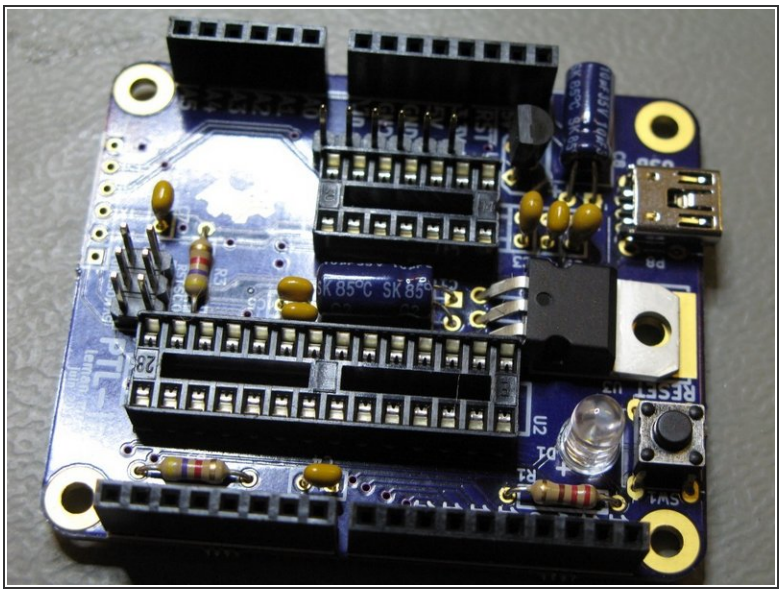
- Put the LED (D1) in place. The longer lead goes where the + marking is on the PCB.
- You can cut the extra leads after soldering.

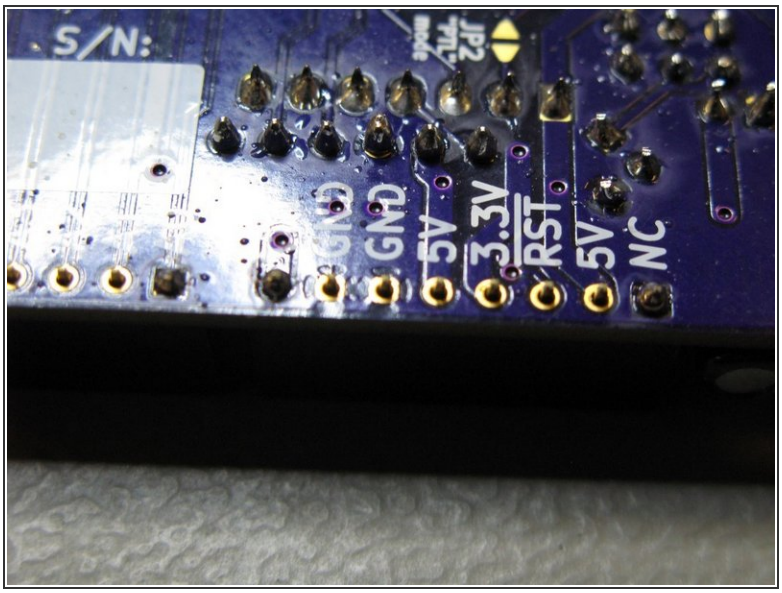
Step 10 — Populate the male headers



- Put the 1x6 **male header** and 2x3 **male header** in their holes and solder them. Masking tape can help to keep them in place.
- Check the picture for the correct position. There is no orientation for headers.

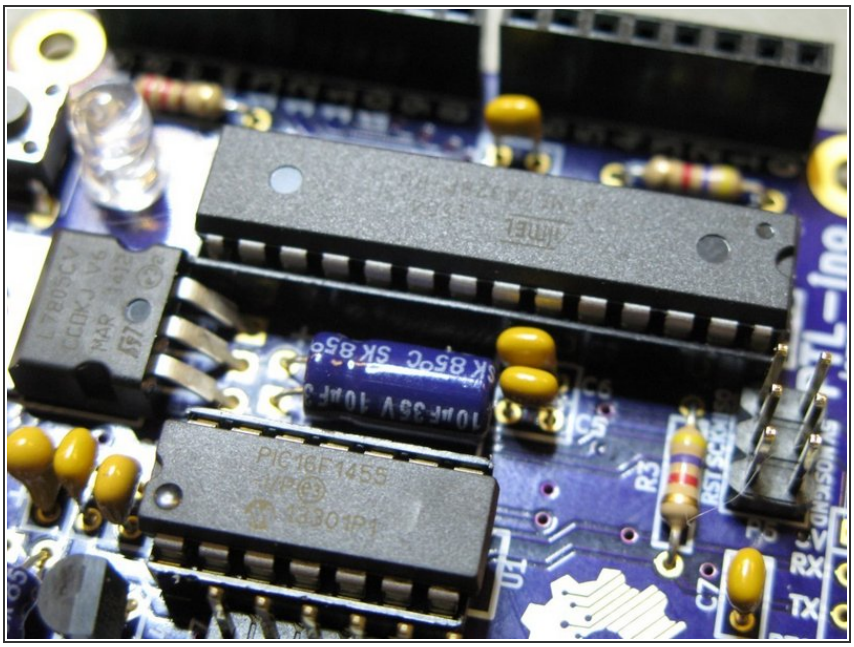
Step 11 — Populate the female headers





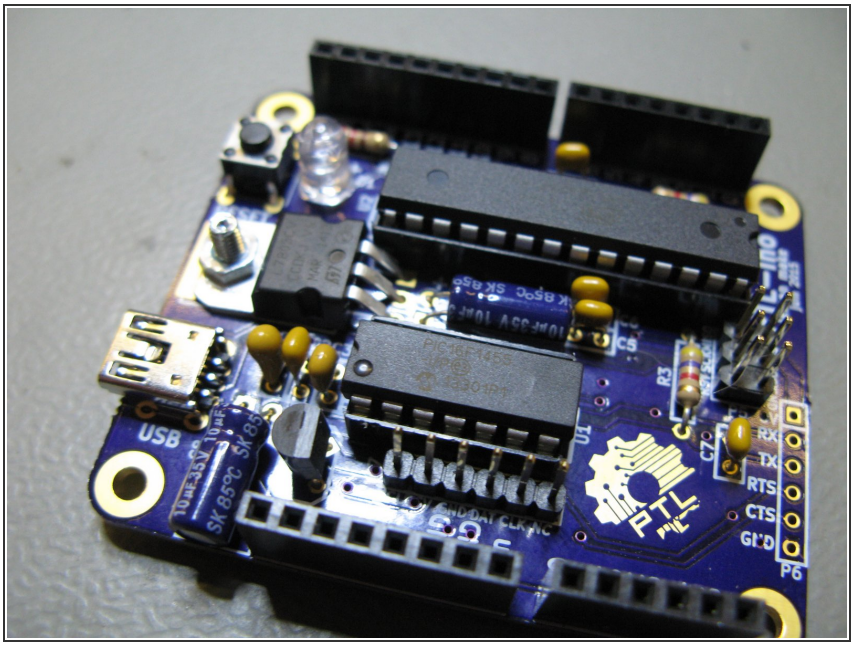
- Put the **female headers** in place. When you solder them, first solder just the 2 extremities.
- After soldering the 2 outer pins of the 4 headers, check that the headers are perfectly perpendicular to the PCB. If not, simply heat one of the soldered pads and push the header flat to the PCB.
- Having the header perfectly perpendicular will make it easy to add "Shields" to your board, otherwise adding a shield will require you to bend the leads

Step 12 — Populate the micro controllers



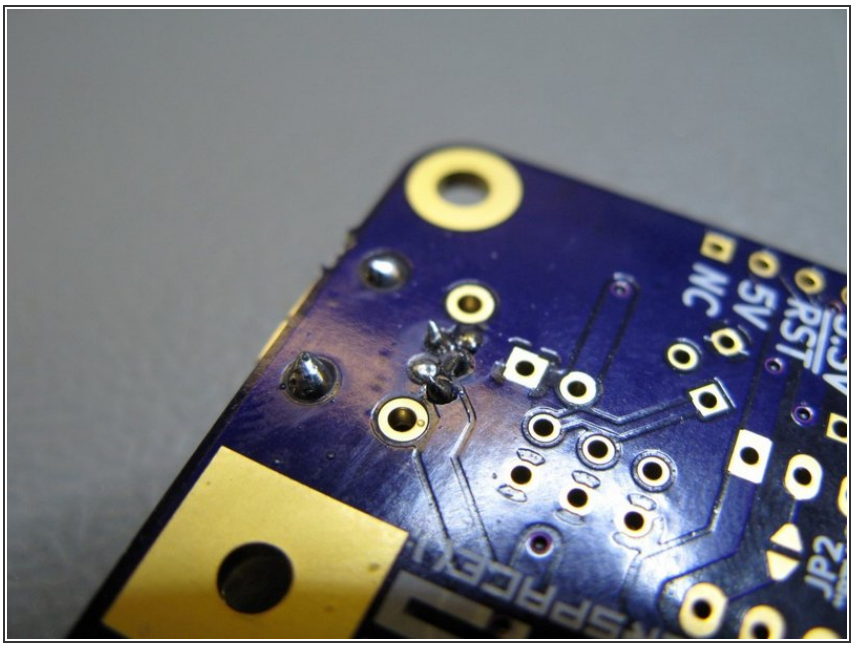
- To insert the micro controller, you have to gently bend the pins so they are really straight. The best option is to slowly push the chip on one side, bending the pins on the table.
- The micro controller has an orientation. Find the small dot and marking on one side, it should match with the marking on the PCB and the IC socket.

Step 13 — Screw the 5V regulator to the PCB



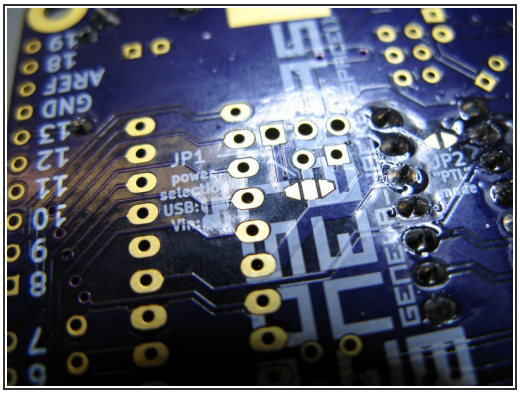
- Screw the 5V regulator to the PCB using the M3 screw and nut.
- The screw and nut are used to make a better contact with the board to dissipate heat. It is not used in the electronic circuit.

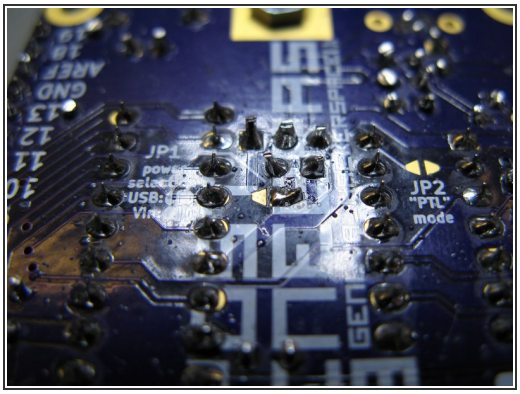
Step 14 — Populate the mini-USB connector

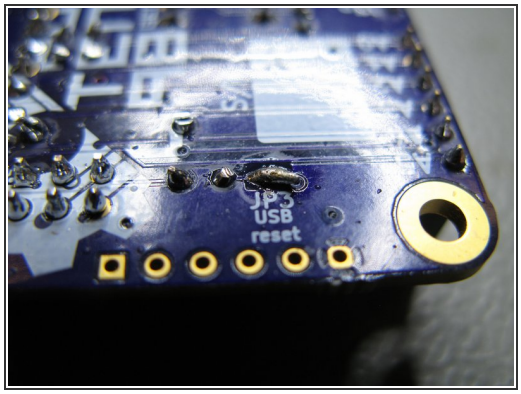


- insert the mini-USB connector and start soldering the 2 (or 4) leads of the connector shield.
- Solder the 5 pins for the USB power and signal

Step 15 — Solder the jumper/bridge on the back-side







- Solder the jumper JP1. Read the marking on the PCB to decide if powering should come from USB or Vin pin.
- Solder jumper JP3 to have the auto-reset feature needed to program with the Arduino IDE:

You can get a complete ready to assemble kit from Post Tenebras Lab.