Fabrication of MicroController Board
ELECTRONICS INNOVATION LAB PROJECT

By: Dhiraj Kumar | Vaibhav Jain | Vikas Kamboj
IV Semester
20/04/2018

Overview

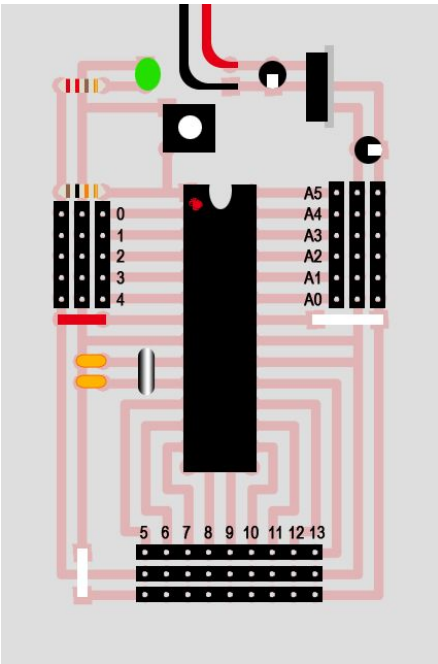
A microcontroller board is a single printed circuit board which contains all the necessary requirements for a basic control task: a microcontroller, I/O ports, a clock generator, power source, RAM and any other necessary ICs. These boards are made with the aim that it will allow a user to quickly start a project without fabricating all the hardware components together every time.

Though, the microcontroller boards allow us to skip the hardware parts, it is necessary for us to have an understanding of how all the components work and participate together. This motivated us to choose this project for this paper.

Materials Required

- AtMega328 microcontroller
- IC holder
- Voltage Regulator (5V)
- Crystal 14.7456 MHz
- Resistor (330E and 10K)
- LED
- Mini Push Button
- Capacitors (10uF/25V and 22pF)
- Male Headers
- Solder Iron and Soldering Wire

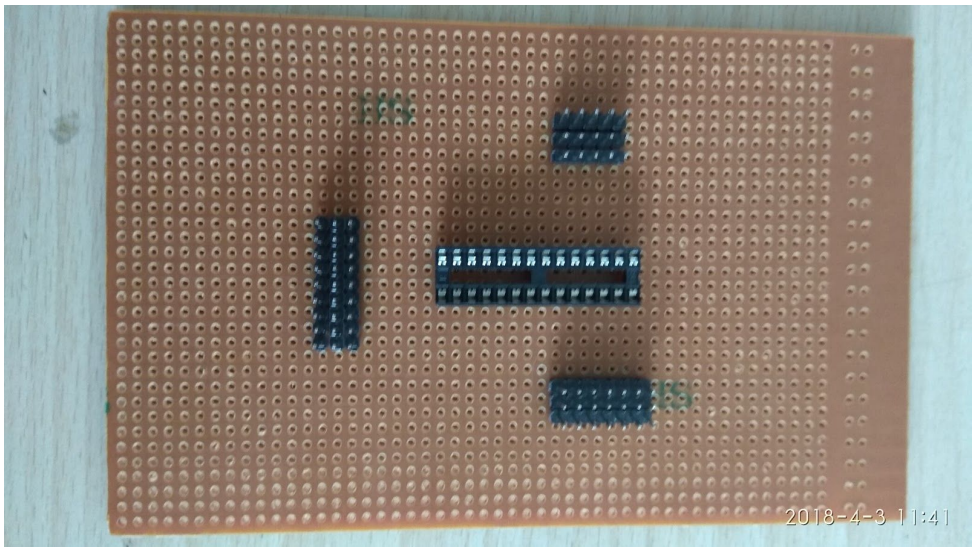
Procedure



This is a schematic diagram of everything.

Follow this diagram to add components to the circuit.

Figure#1



Start by soldering the DIP Socket for the ATmega328.

DO NOT solder in the actual chip.

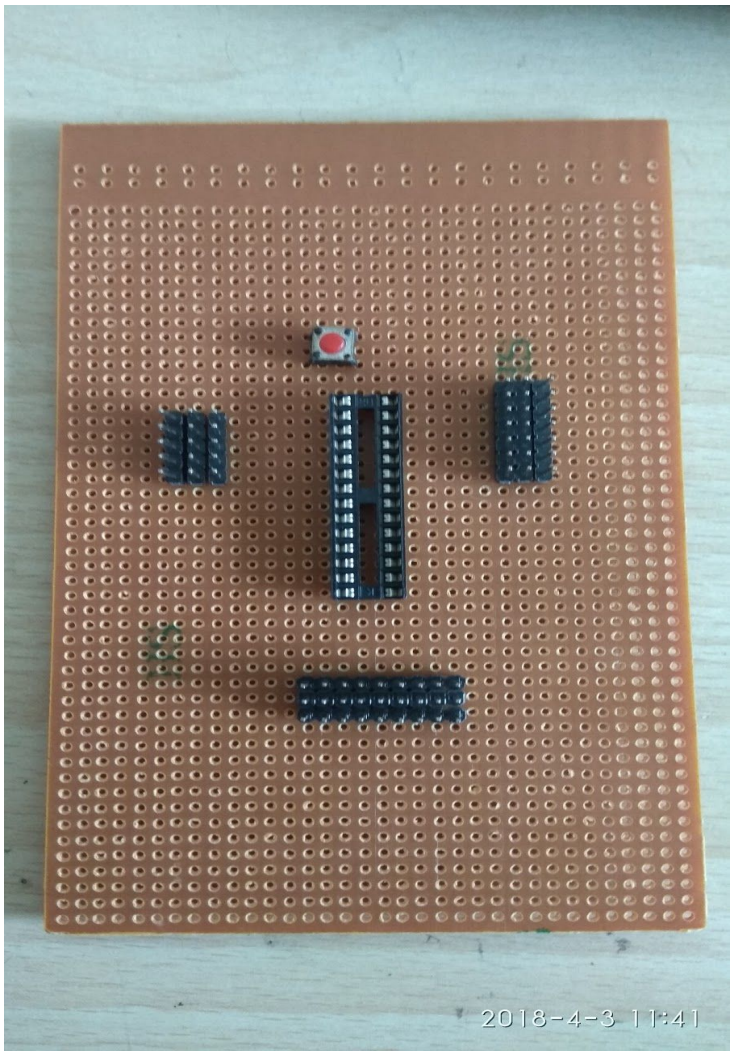
It is always a good idea to use these Sockets, because you can remove your chip if you want to use it in another board or project or whatever. And de-soldering them is a royal pain.

Note the area where there is a little oval cut out. This is to let you know how the chip should be oriented. When you finally put your ATmega chip into the socket, you want to place it with the oval facing up.

The top left of the chip is the reset. If you don't put the chip in right, nothing will work.

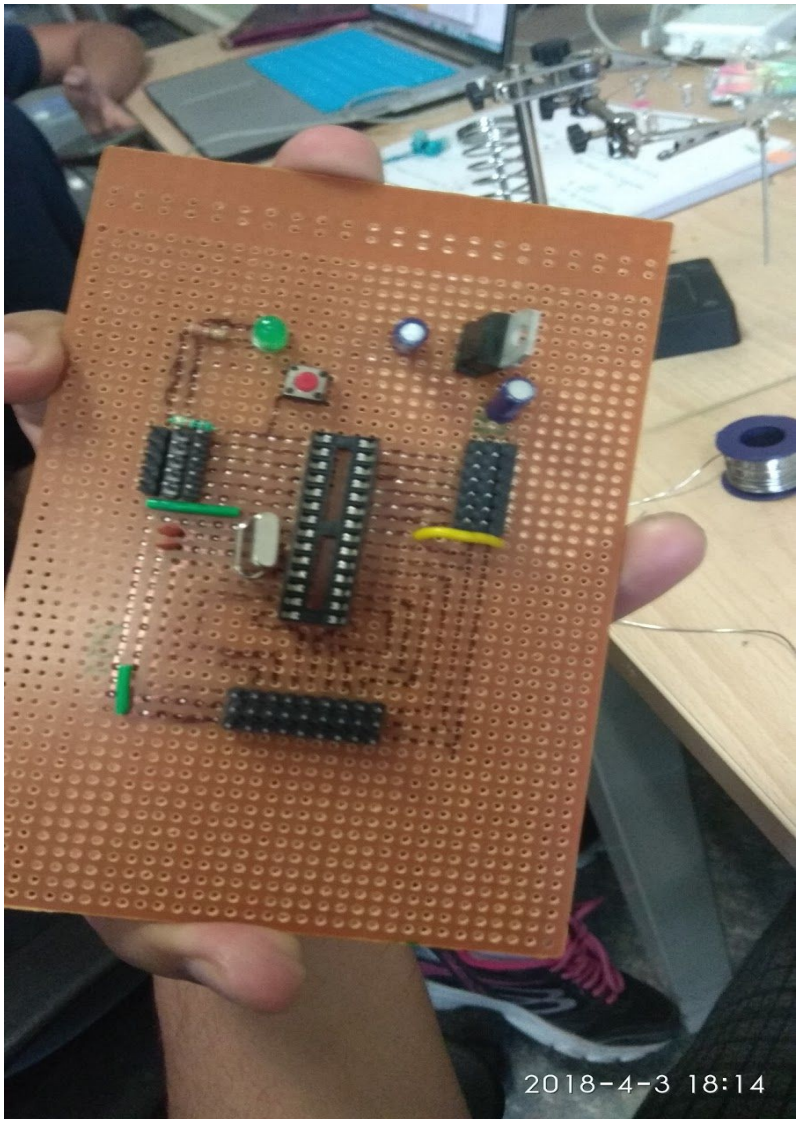
Add the little jumper wires.

Figure#2



Add the female headers and reset button.

Figure#3



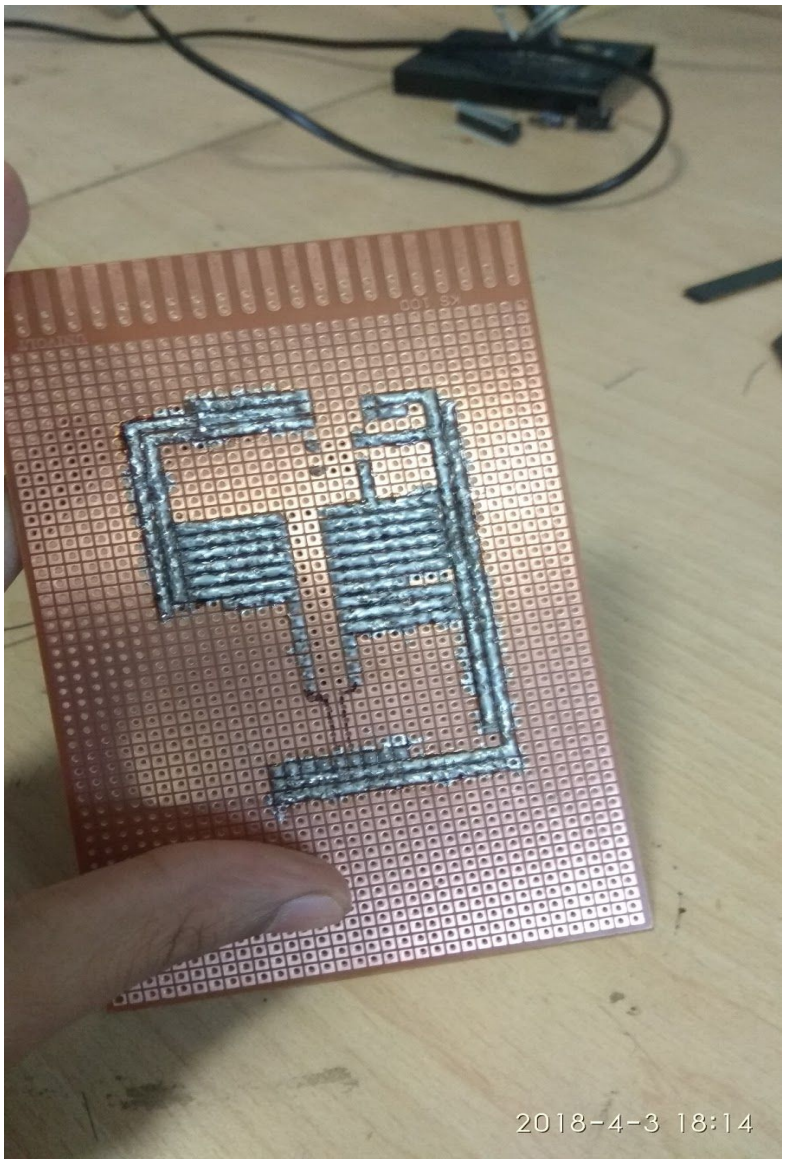
Add the capacitors and the voltage regulator.

Important: The capacitors have a little white bar on one side. This is to let you know where to attach it to the ground.

Attach the side with the white bar to where the square pad (ground) is and the other to the power. Do this for both of the capacitors.

The voltage regulator needs to be soldered with the back of the "chair"/ the tall metal part facing out.

Figure#4



Add the 22pF ceramic capacitors (2 of them) and the 16mHz crystal.

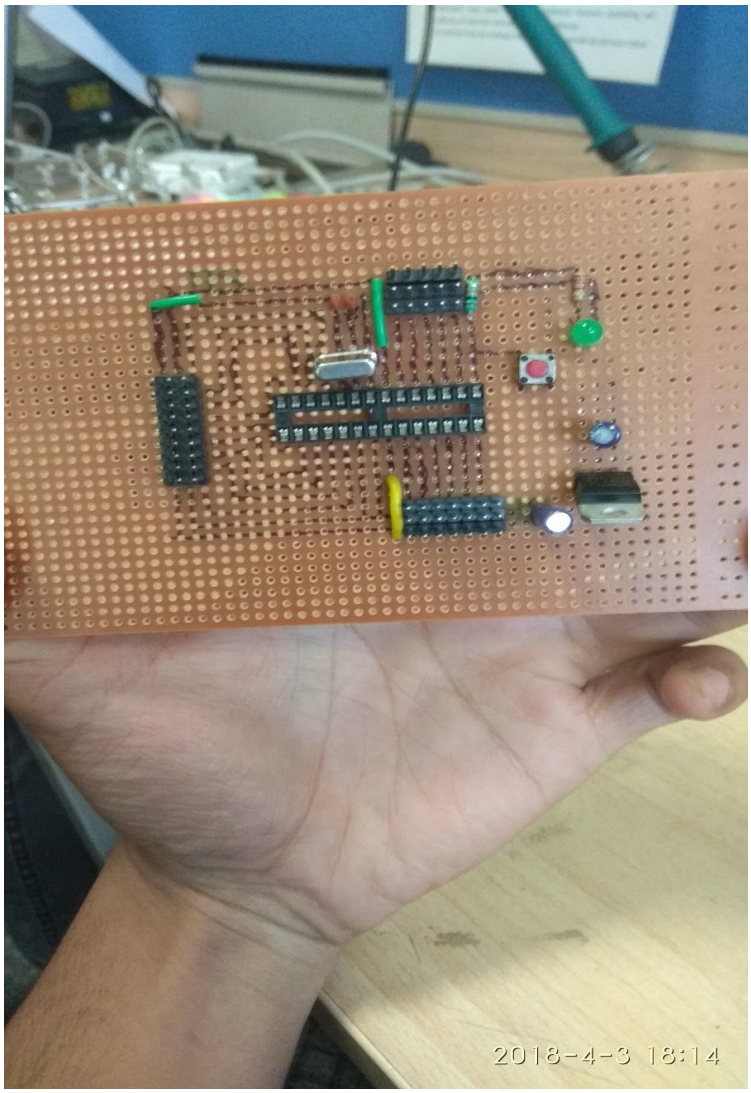
It does not matter how the ceramic capacitors or the crystal are oriented.

Add the resistors.

The resistors are 10K Ohm and 220 Ohm.

The 10K Ohm resistor is banded brown, black, orange and gold and will provide resistance to the button. The 220 Ohm resistor is banded red, red, brown and gold and will provide resistance to the LED.

Figure#5

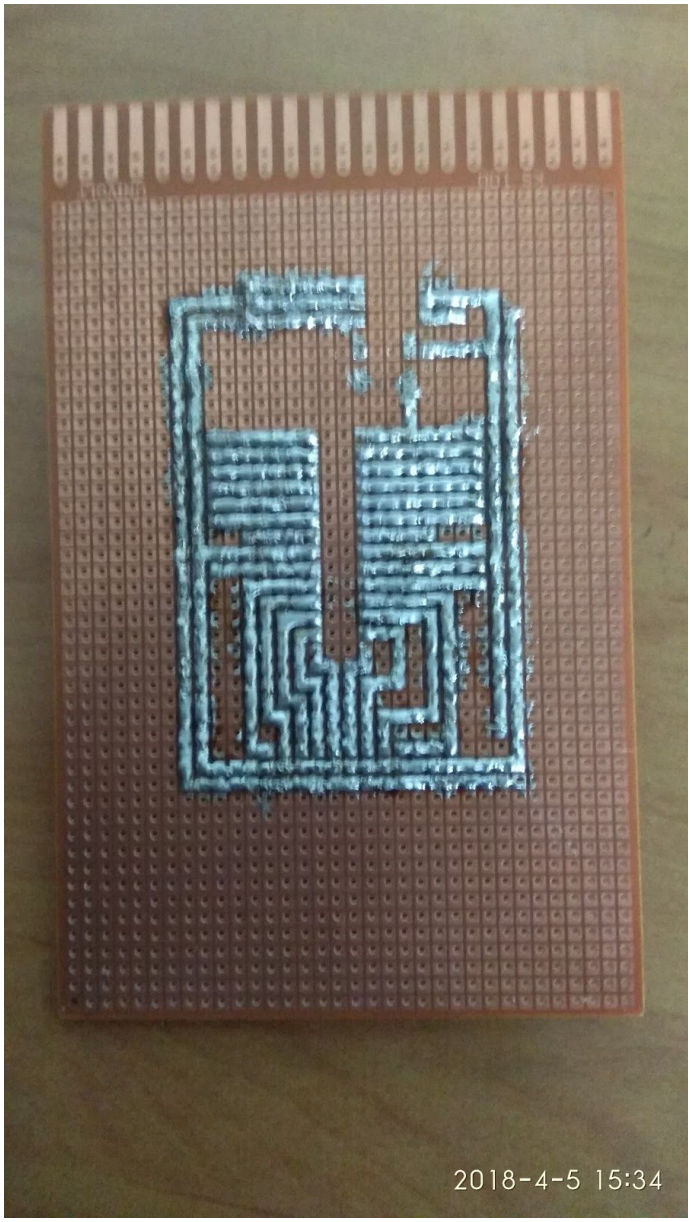


Add the LED and button.

The LED will light up and tell us that the board is getting power.

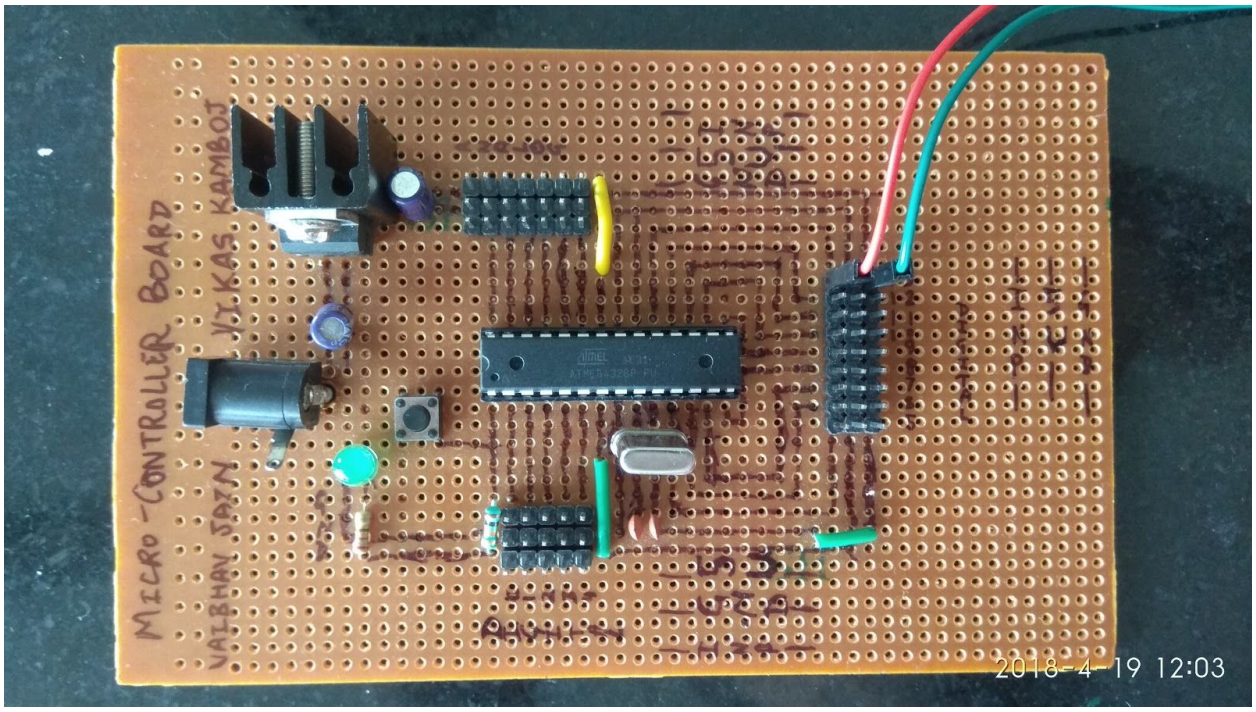
The button will allow us to reset the board.

Figure#6



Solder the remaining circuit.

Figure#7: Final Result



Here is the final self-made microController Board.