



Components:

1 X 28 pin IC socket, 1 X 16Mhz Xtal, 2 X 22pF ceramic capacitors, 1 X 220 Ohm resistor, 1 X 10K ohm resistor, 1 X green 3.2mm LED, some female headers and the PCB.

Construction:

Construction is started by inserting the two capacitors and the 16MHz crystal, solder these in place and trim the leads short. Next we insert the LED and take note of the orientation, add the 220 ohm resistor and the 10K ohm resistor. Insert the IC socket and tack two opposite pins to hold it tightly in place while you solder the remaining pins.

A small modification to the TTL pins on the Bootloader is required, once plugged in to the KMDuino we will not be able to program the new device unless we extend the pins through both sides of the board. It is simple to add a strip of male header pins on top once the female headers are soldered below. Finally you can add the remaining female headers at the bottom of the board. These are made up of one section of 6 holes, one section of 9 holes and one section of 7 holes.

At this stage, check your soldering to ensure that there are no solder bridges or bad joints. Once you are sure then your board is complete and ready for use.

To use the board, just insert it on top of the KMDuino, insert the blank Atmega chip, connect to your computer through the TTL socket as normal (remember we added TTL pins on top of the Boot Loader which feed through to the TTL below). Open the Arduino software, select tools > board menu for the board you want to burn the boot loader for (we will use Arduino UNO) and then select Burn Bootloader > Arduino as an ISP.

Congratulations, you have successfully created your own Bootloader Shield and programmed a blank Atmega 328 chip with the bootloader.

PCB Files: <http://www.zs6kmd.za.net/bootloader.rar>

Happy Building

73 de Kevin ZS6KMD