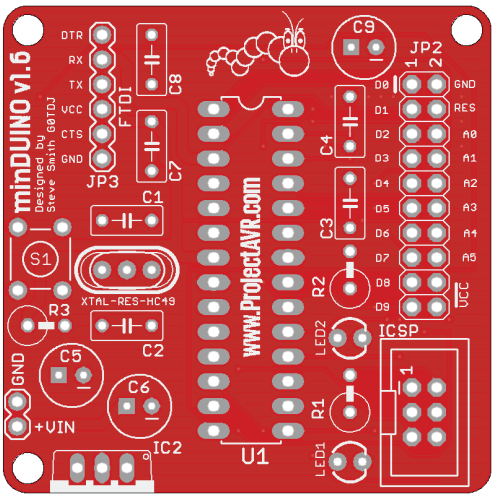
minDUINO v1.6

Assembly Instructions



minDUINO v1.6 PCB

The minDUINO series of PCBs are small footprint Arduino clones. They have been designed to enable learning about AVR Microcontrollers, microcontroller programming and electronic experimentation. Similarly to the Arduino UNO, the minDUINO is based around the very common Atmel ATMEGA328P chip. This chip contains a CPU, Flash ROM and RAM areas. It is, in fact, a very small computer. Many things can be accomplished with this microcontroller and it's family.

The minDUINO PCB is straightforward to assemble. You will need a soldering iron with a small to fine tip, a pair of side cutters (clippers), a pair of pliers (long nose if possible) and a clear work surface.

**N.B. Please be very careful with a soldering iron. When they are powered up, they are very hot. Please make sure others are not in danger of tripping over the cable and that the iron is secured out of the way when not in use.*

The first step in assembling your PCB is to check all the components are present. Check them off on the following list:

minDUINO v1.6 Parts List

Part	Value	Component Type	RS Catalogue No.	Quantity
C1/2	22pf	Ceramic Capacitor	721-5101	2
C3/4/8	100nf	Ceramic Capacitor	789-9242	3
C5/6	100u	Electrolytic Capacitor	703-8106	2
C7	10nf	Ceramic Capacitor	721-5234	1
C9	10u	Electrolytic Capacitor	117-035	1
IC2	LM7805	Linear Regulator	806-2839	1
JP1	2x3 Pin	Unshrouded Pin Header	719-0012	1
JP2	1x6 pin	Shrouded Pin Header	765-5666	1
JP3	1x2 pin	Unshrouded Pin Header	360-6342	1
JP4	2x10 pin	Unshrouded Pin Header	670-3553	1
LED1	RED	3mm LED	708-2753	1
LED2	GREEN	3mm LED	708-2747	1
R1/2	330R	1/4 Watt Resistor	165-0836	2
R3	10K	1/4 Watt Resistor	165-1031	1
S1	Tac	Tactile Switch	758-1929	1
U1	Atmel ATMEGA328P-PU	AVR	696-2260	1
Socket	28 Pin	IC Socket	801-796	1
XTAL	16MHz	Crystal	693-8819	1
Y2	16MHz	Resonator	792-6542	1 *Optional

Install the IC Socket first. Make sure that you align the notch on one end of the socket with the notch indicated on the PCBs white printed symbol. The notch shows which way round you will eventually plug the ATMEGA328 chip in. Place the IC Socket carefully into the 28 holes in the center of the PCB and double check you have aligned the notch correctly. Flip both the PCB and socket over so the board is resting down on the socket. Solder the top left and bottom right connections, and after they have cooled sufficiantly, check the socket is flush with the board. If it is, you can carry on and solder the remaining connections. Solder one after the other in a slow careful manner checking the connections as you go.

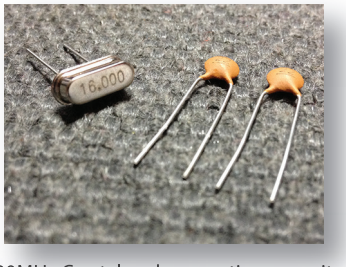
If the socket is not quite flush with the PCB, place the PCB Between your thumb and middle finger. Press your forefinger lightly on the socket and reheat the connections. The socket should become flush with the PCB. Be careful to touch the center of the socket as you do this as to avoid burning your finger. Lay the PCB back down and carry on soldering the other connections after this.

Next install the switch S1. This should clip into the PCB and not fall out when you flip the board over. Solder diagonally top left - bottom right, top right - bottom left. This lets the internal parts of the switch cool down as you solder.

The ceramic capacitors come next (ignore C1 & C2 for now). Identify C7. It should be marked with the number 103. Capacitors are measured in units of Farad. The 103 represents the number 10 and three zeros, 10,000 (meaning 10 nf - ten nano-farads). Lots of component values are marked in a similar fashion. Place C7 with the number facing to the right of the board. Flip the board and the capacitor over, like you did with the IC Socket and solder it in. Once the solder cools down, clip the lead ends off as close as you can get to the PCB with your clippers *(The reason the number is faced to the right is so that if someone wishes to read the component values after the PCB is assembled, they can do so without having to turn the board around multiple times, and also it looks neat!)*.

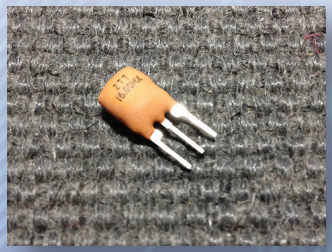
Follow on with C8, C3 and C4 (all marked 104) in the same way, remembering to face the number to the right of the board.

The ATMEGA328P microcontroller, like all microcontrollers, requires a clock to run to. The ATMEGA328 has an internal device called an R/C oscillator which can be programmed to run the chip. However, it is a basic oscillator and isn't very accurate. A much more accurate oscillator is configured externally. In the minDUINO's case, it is configured to use a crystal and two supporting capacitors. Fit the crystal, C1 & C2 next.*



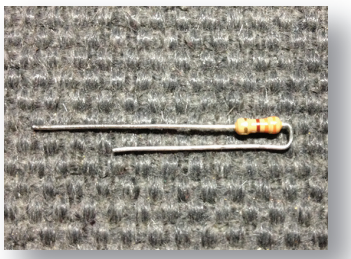
16.00MHz Crystal and supporting capacitors

Time to tackle the LEDs and their associated resistors. LED1 is the power LED. I suggest you use the red LED for this purpose but there is no reason not to use the green one. If you look at the white print on the PCB you will see that one side of the LED symbol is round, the other flattened off. You will find the component is the same. The negative lead (The shorter of the two on the component) matches the flattened off side. Make sure you place the LEDs in the correct way round otherwise they will not work and may get damaged. Once you have soldered in the LED, find the 330R Resistor. The colour code for these is ORANGE-ORANGE-BROWN-GOLD. You should have two of these, one for each LED. Bend the lead opposite to the tolerance band (GOLD) over at 90 degrees. Try not to bend it right next to the resistors body because this will weaken it. Then bend the remaining lead a further 90 degrees.



16.00MHz Resonator with built in capacitors

**Provision has been made in the kit to optionally fit a ceramic resonator instead of a crystal and capacitors for the clock. In this case, do not fit C1 & C2 since the resonator has capacitors built in.*



330R Resistor with correctly bent lead

Follow the white printed symbol on the PCB to place the resistor correctly. If the component is fitted the wrong way round in this case, it would still work without any issues. After you have fitted LED1 and R1, go ahead and fit LED2 and R2 the same way.

Since you have had practice with R1 & 2, you can now fit R3 in exactly the same way. The colour code for this one is BROWN-BLACK-ORANGE-GOLD.

Time for the larger Electrolytic capacitors now. These are black cylindrical components and must be fitted the correct way round. If they aren't fitted correctly, there is a danger that they could explode and cause damage, including the possibility of you being indured. Be extra careful! The electrolytic capacitors are clearly marked with a '-' (minus sign) in a strip down the side of the body. This is the negative lead. Also, the longer lead is the positive. On the PCB, the positive lead goes into the square hole and the negative into the round hole. Fit C9 first. It's value is 10uf (micro-farad). As usual, fit flush to the PCB. Now you can fit the two taller 100uf electrolytic capacitors in the places for C5 and C6.

The last active component to fit is the 5v Linear Regulator. This is a three leaded device that has a tab with a hole in it. This is to fit an optional heatsink, a piece of metal to dissipate heat. The minduino's regulator shouldn't need a heatsink since it doesn't use enough current to get hot. Fit the regulator with the tab's silver side to the outside of the PCB as the white printed symbol shows. It won't go down completely flush with the board because the leads have wide sections in them but this is normal.

Now, find JP3 which is a 6-pin socket in a single line. This is to allow the use of an optional FTDI device for programming. As with the IC Socket, solder the two end connectors first and check it is flush with the board. If not, deal with it appropriately. Complete by soldering the rest of its connections.

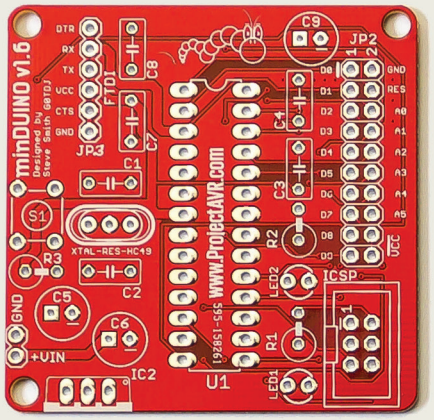
Next, fit the 6-pin connector ICSP (In Circuit Serial Programming). This allows direct programming of the AVR Chip.

Lastly fit the two pin header for the power input near to the regulator. This can be a bit fiddly so take your time.

It is worth having a quick re-run through these instructions and check you haven't forgotten anything or fitted a component the wrong way round before you power it up. The AVR chip is programmed with a simple blink program for testing so when you have re-checked, carefully bend the chips pins in slightly by holding the body of the chip and pressing the pins on a straight surface just a little. This allows the chip to fit into the socket easier. Insert the ATMEGA328 chip into the socket, not forgetting to orient it with the notch of the IC socket matching the marking on the chip. You may then connect power to the GND and +VIN lines via the connections near the regulator. All being well, you should see your test LED pulsing. If not, remove the power quickly and double check all connections. Check that you have connected the power the right way round (Positive to +VIN and Negative to GND).

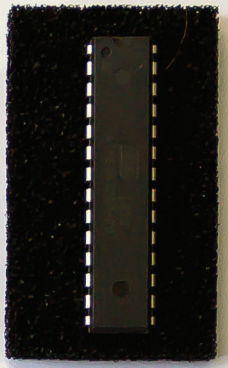
All design information and files for this project, and others are available on **www.projectavr.com**

Component Identification



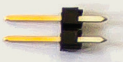
minDUINO v1.6 PCB

28-pin IC Socket



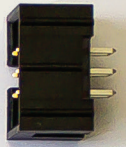
ATMEGA328P

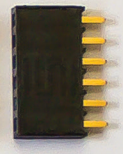
20-pin Header



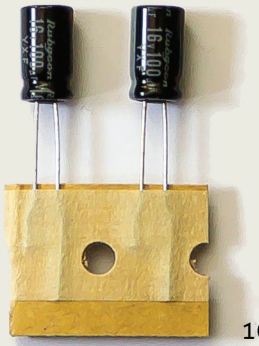
2-pin Power Header

6-pin ICSP Header





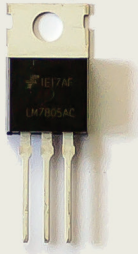
6-pin FTDI Header

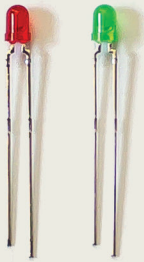


2x100µf Electrolytic Capacitors

10µf Electrolytic Capacitor

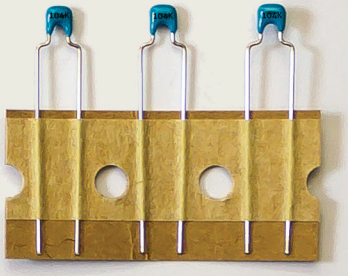
7805 5V Linear Regulator





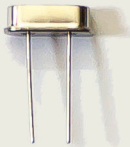
Red & Green LEDs

Tac Switch

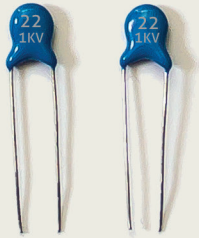


3x100nf (104K) Ceramic Capacitors

16MHz Ceramic Resonator

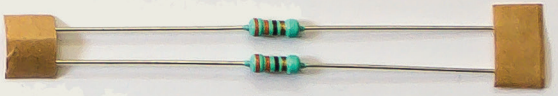


16MHz Crystal



2x22pf Ceramic Capacitors

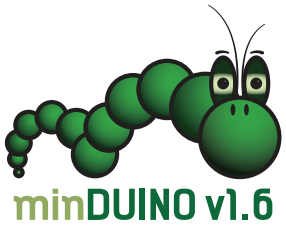
10nf (103) Ceramic Capacitor



2x330R Resistors (ORANGE-ORANGE-BLACK)

10k Resistor (BROWN-BLACK-ORANGE)

**Component types may be substituted with equivalents*



minDUINO v1.6

designed by
Steve Smith G0TDJ