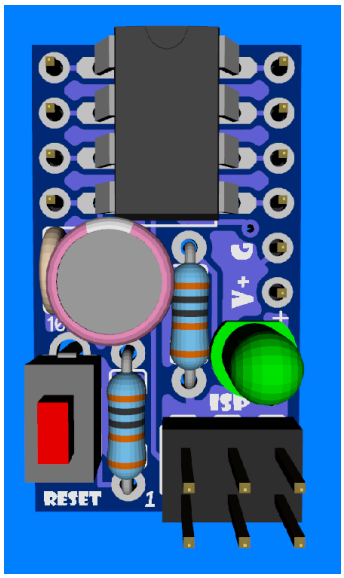
ATtiny85 ISP!

ATtiny25/25/85
Programming and Breakout Board
Build Document and Helpful Tips (rev 1.23.2014)

RULLYWOW INDUSTRIES
www.rullywow.com

Overview



The ATtiny85 ISP! PCB is designed to help you work with Atmel® ATtiny™ DIP-8 series of microcontrollers (ATtiny25, ATtiny45, ATtiny85) easily. The ATtiny85 ISP! is small and inexpensive enough that it can be left in an embedded project. Now you can prototype with the powerful ATtiny™ microcontroller series easily.

Features:

- Arduino™ compatible, using Arduino IDE!
- ISP-6 header for use with standard ISP programmers
- Reset button
- LED power indicator light
- Appropriate filter capacitors
- Designed for breadboard/embedded use

Required components

(1) ATtiny85 ISP! PCB
(1) 220ohm 1/4w resistor
(1) 10k 1/4w resistor
(1) 3mm LED
(1) 100uF 5mm diameter electrolytic capacitor (6.3v)
(1) 100nF (.100 uF) MLCC capacitor (50v)
(1) 2x3pin 2.54mm header (6pin ISP header)
(1) 1x10pin 2.54mm header
(1) DIP-8 Socket
(1) Tact Switch – Through Hole (6 x 3.5 x 4.3mm/5mm) Example: http://www.digikey.com/product-detail/en/TL1107BF130WQ/EG2512-ND/378978
(1) ATtiny85-20PU

Section 1: Assembling the ATtiny85 ISP!

Tools

Required tools:

- Soldering Iron
- Solder
- Wire cutters (flush type)

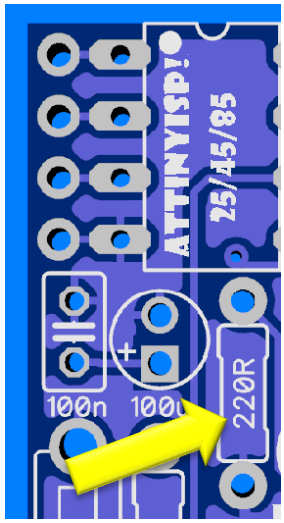
Optional (but helpful) Tools:

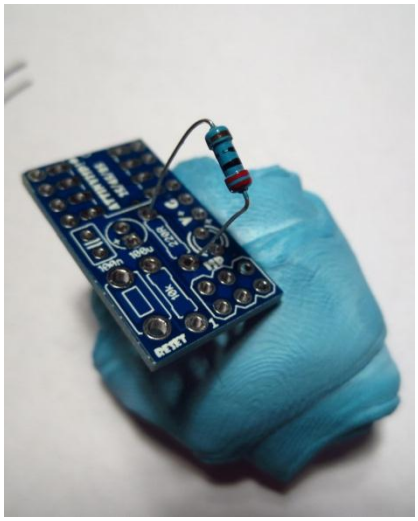
- Desoldering braid/solder sucker/desoldering iron
- Small needlenose pliers
- Soldering flux
- Breadboard
- Blu-Tack (poster putty) or "helping hands" or Panavise® etc.

Step 1: install 220 ohm resistor

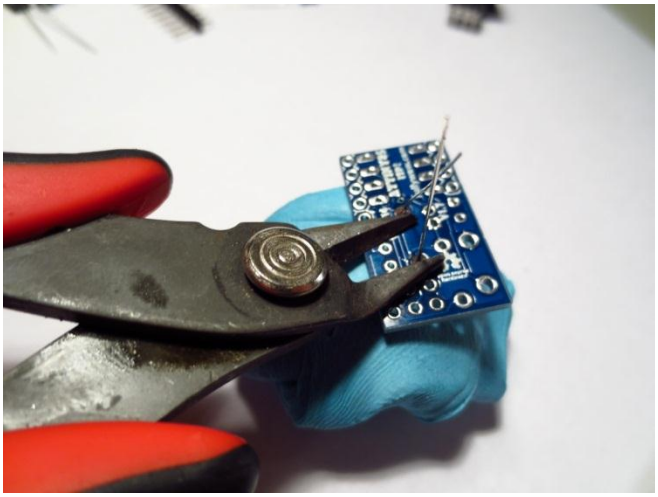
We will populate the PCB starting with the lowest profile components, and working up to the largest. The 220 ohm resistor is used to regulate the voltage that the power indicator LED receives. First, you will need to identify the 220 ohm resistor. You can use a multimeter on the ohm setting to measure, or you can match up the identifier color bands to the picture on the right.

Place the 220 ohm resistor in the PCB in the spot marked "220R", orientation is not important. The "blue stuff" is poster putty which works very nicely for holding components in place while soldering. If you haven't tried this, you should!





Once the resistor is flat against the board, solder it in place from the bottom side and clip the excess leads. Notice the putty is holding the resistor flat against the board while soldering.



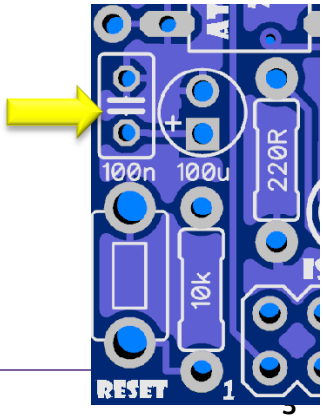
Step 2: install 10K resistor

This is the pull-up resistor for the RESET button. Just as we did in step 1, find the 10K resistor and place it in the PCB where it says "10K". Orientation is not important. Solder from the bottom of the board and clip off the excess leads. The kit's 10K resistor's color bands look like this:

Step 3: install 100nF capacitor

This capacitor provides filtering for the power supply feeding the microcontroller. The 100nF included in the kit is marked "104" and looks like this:

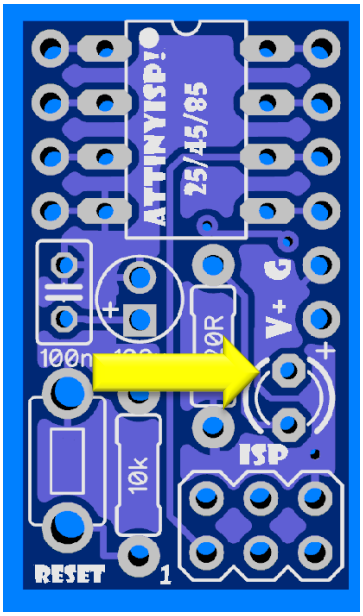
Insert the capacitor into the spot which says "100n" from the top of the PCB, orientation is not important. Solder the leads from the bottom of the PCB and clip off the excess.



Step 4: install LED

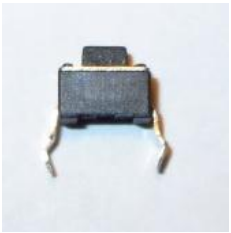
The LED provides an indication there is power going to your ATtiny85 ISP! board. LEDs usually have one lead which is longer than the other. The longer lead is the positive (+) lead.

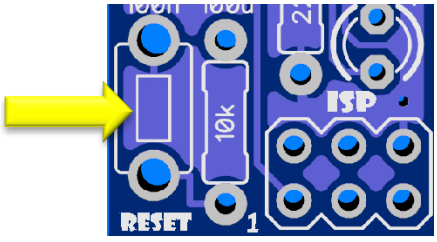
On the PCB, you will find the LED location above the ISP connector indicated by the arrow in the picture. Notice the "+" sign near the top hole. You will insert the LED's longer lead into this top hole and solder from the bottom of the PCB. Once soldered into place, clip off the excess leads. Be sure to get the longer lead in the top hole, or the LED won't work!



Step 5: install reset switch

Grab the rectangular momentary switch from your parts stash, it looks like the picture on the right. Install it into the spot on the bottom left of the PCB above where it says "RESET". Orientation is not important. Solder the switch leads from the bottom of the PCB.

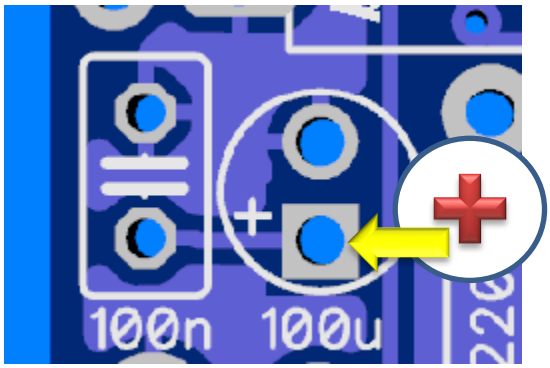




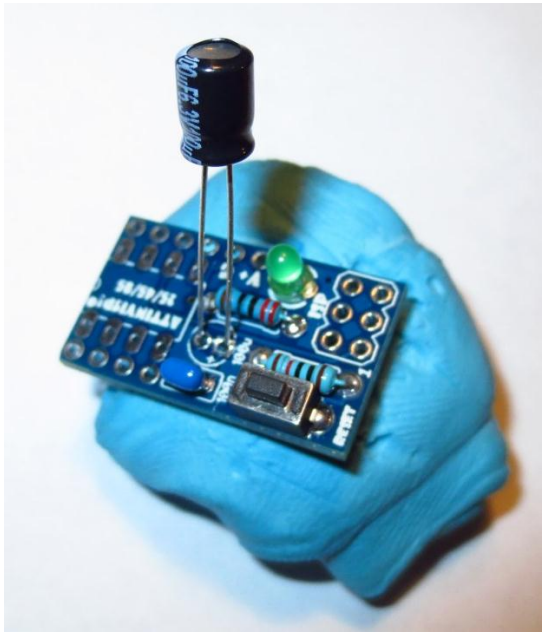
Step 5: install 100uF capacitor

The 100uF capacitor provides additional power filtering and smoothing. The capacitor, much like the LED, has one lead which is longer than the other. The longer lead is the positive (+) side of the capacitor. The negative lead is the shorter lead, which is also identified by the lighter-colored stripe on the can.

Insert the capacitor into the spot on the PCB marked "100u". The longer lead will go into the square shaped pad. Ensure you get the orientation correct. Solder from the bottom of the PCB and clip off the excess leads.

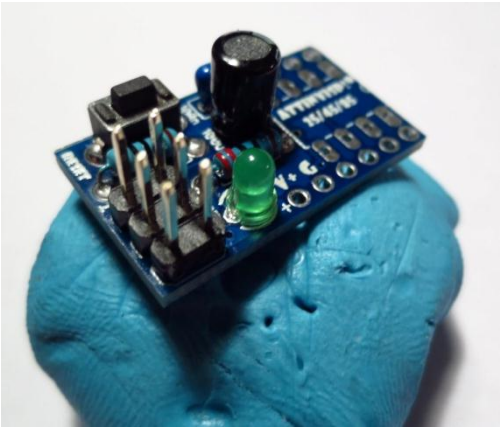


The stripe on the can should face towards the top of the board like this as you are inserting it.



Step 6: install ISP header

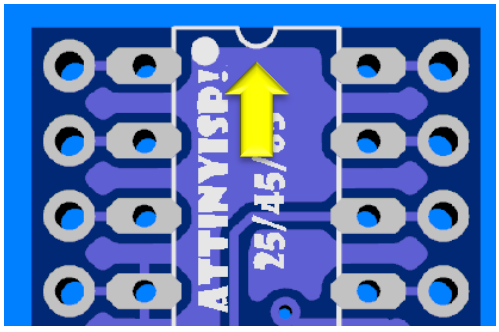
Find the 2x3 dual row header, it looks like the picture on the right. Install it into the spot on the bottom right of the PCB above where it says "ISP". Orientation is not important. Solder the pins from the bottom of the PCB after insertion. It is a good idea to tack one leg into the hole from the bottom and check to see the header is flush against the board before soldering the rest of the pins.



Getting close! You should have something that looks like this so far...

Step 7: install DIP socket

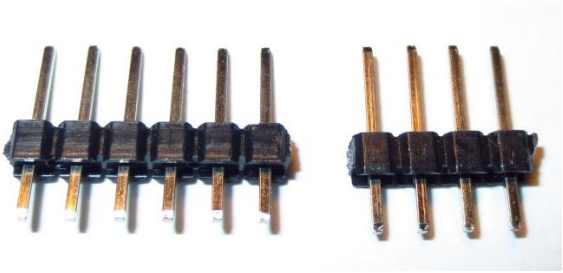
The DIP socket is notched on one end with a "U" cutout. Ensure this cutout matches the orientation of the silkscreen on the PCB board.



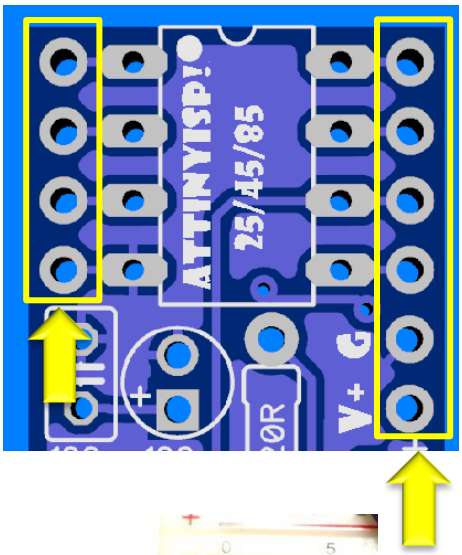
Insert the DIP socket into the board from the top and solder all 8 legs from the bottom of the board. As with the ISP header, a good tip is to tack one leg in first and ensure the component is sitting flat against the PCB. Once you are satisfied, you can solder the rest of the legs in.

Step 8: install header pins

Find the straight header pin section. With a pair of pliers, you will carefully break it so you end up with two pieces. One piece will need to be 4 pins long, and the other will be 6 pins long.

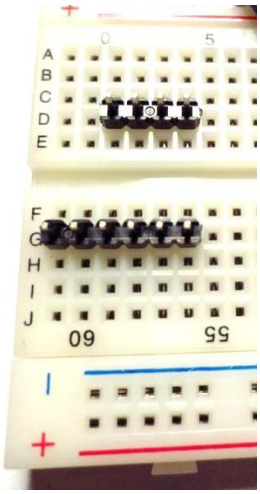


These headers will be the pins which are inserted into your breadboard. The pin locations are on the left and right sides of the PCB.

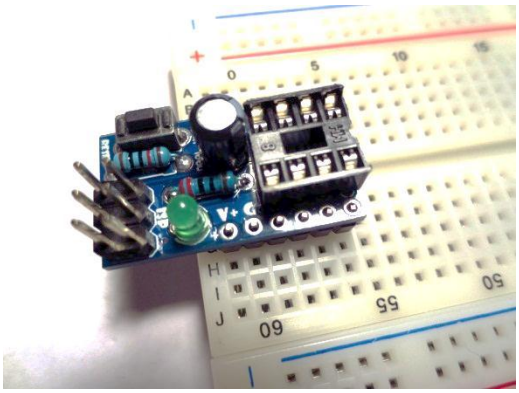


While you can solder the header pins while trying to be careful that they are at a right angle to the PCB, it is highly recommended to use a breadboard which helps keep the pins alligned while soldering.

Insert the header pins into a breadboard like this:



This will allow you to put the PCB assembly on top of the pins and solder each of the 10 pins from the TOP of the board.

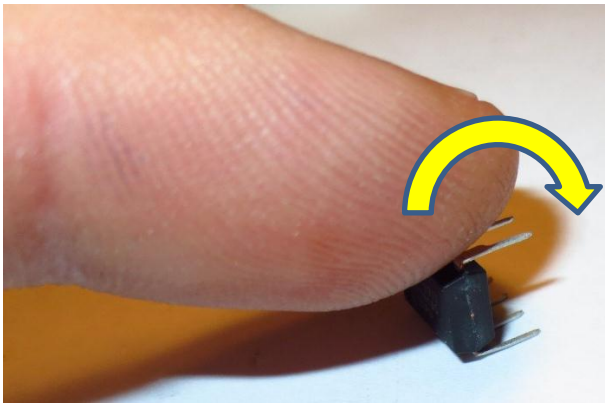


Step 9: install ATtiny85 microcontroller

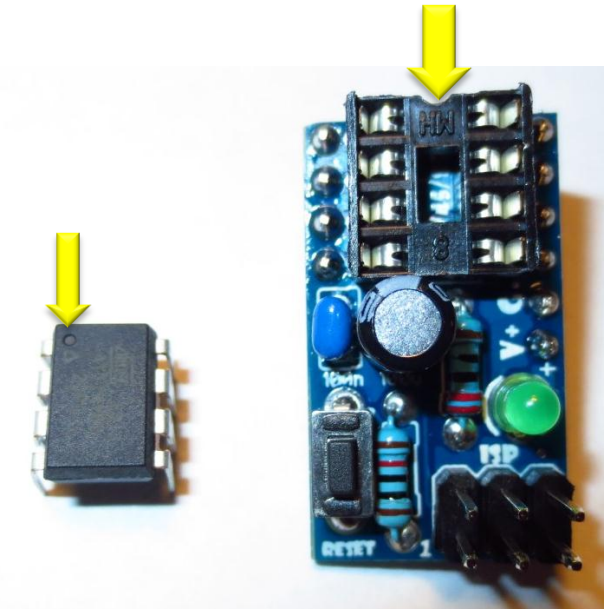
Great job! The final step is to take the ATtiny85 uC and insert it into the DIP socket.

The chip as it comes from the factory, will most likely come with its pins spread slightly in a "v-shape" like this:

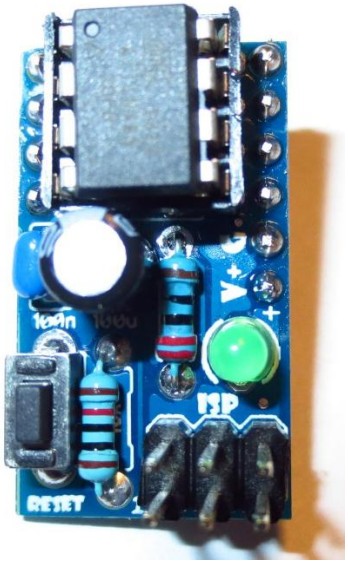
You can carefully use the flat surface of a table to "roll" the chip and bend the leads straight. Be careful not to overdo it! Try to bend each side of pins into a 90-degree position so the chip will fit nicely into the DIP socket.



Once the leads are straight, you are ready to install the chip into the socket. Notice the small dot on the chip's corner indicates the #1 pin. You will need to ensure this dot is installed on same edge the "U" notch in located in the socket. In this case, the dot on the chip will be nearest the top of the PCB. Carefully insert the chip in the socket – try not to bend any pins.



Step 10: Finished!

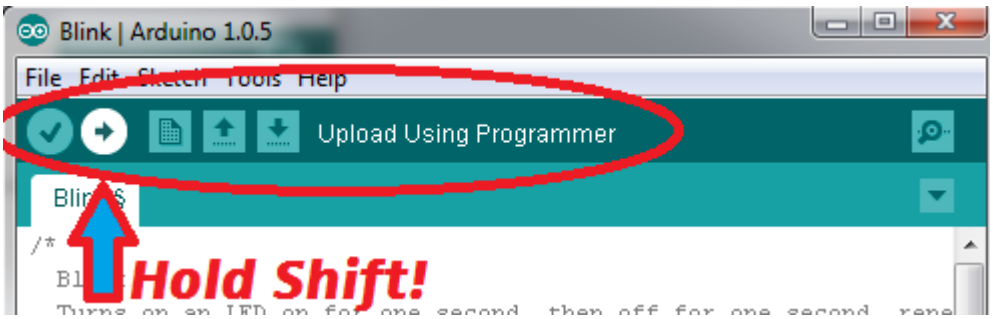


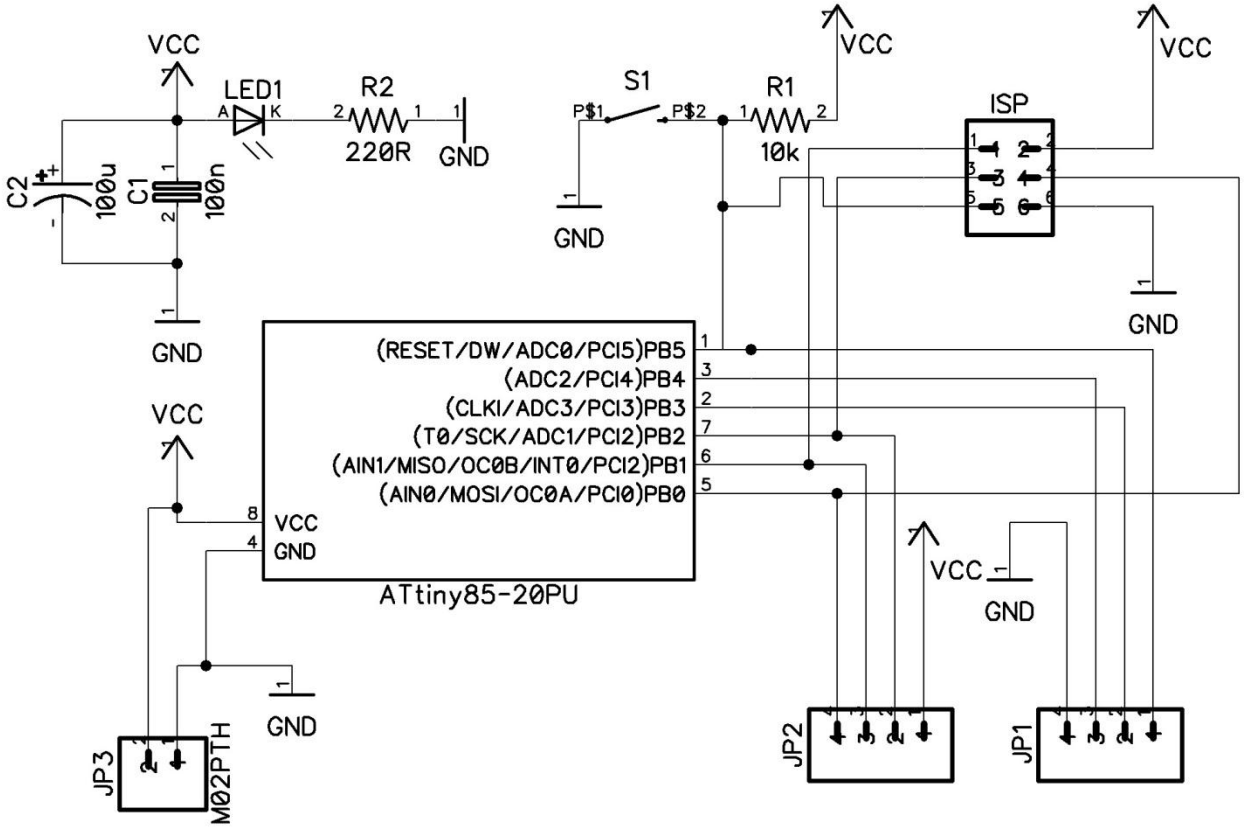
Very nice work! Assembly is complete, and you should have a ATtiny85 ISP! which looks like this.

Before firing it up for the first time, check to see if there are any solder bridges (globs of solder) which could be shorting out components and traces on the PCB.

Helpful Tips:

- highlowtech.org is a great resource for learning how to interface the ATtiny™ with Arduino IDE. Really awesome info.
- The ATtiny85 ISP! will be powered by the ISP cable when plugged in. This means if you use a USB programmer, you can power your breadboard via your USB port (up to 500mA).
- If powering the chip/board from an external source, be careful to ensure the polarity is correct, otherwise you may destroy the chip (poof!) The max voltage is 5VDC.
- The V+ and G pins simply carry the 5v positive and negative to the right side of the PCB. This makes it easy to wire up to your breadboard rail or other circuits.
- After downloading the required ATtiny files to your Arduino IDE, you will need to first use the "burn bootloader" function. This actually doesn't upload a "bootloader" but rather sets the fuses and clock speed of the chip before you can upload sketches
- You may need to unplug the ATtiny85 ISP! board from your circuit if you get an error when programming it. Simply pop the PCB out of your breadboard, upload a sketch, and pop it back in. Connected circuits may share the MISO/MOSI/SCK lines.
- When programming with the Arduino IDE, you will need to hold the "Shift" key down while clicking the upload arrow. This "uploads using programmer." This is an easy one to miss!




VCC
LED1
A
K
R2
220R
GND
C2
100u
C1
100n
GND
S1
P$1
P$2
R1
10k
GND
VCC
ISP
GND
VCC
(RESET/DW/ADC0/PCI5)PB5
(ADC2/PCI4)PB4
(CLKI/ADC3/PCI3)PB3
(T0/SCK/ADC1/PCI2)PB2
(AIN1/MISO/OC0B/INT0/PCI2)PB1
(AIN0/MOSI/OC0A/PCI0)PB0
VCC
GND
ATtiny85-20PU
VCC
GND
VCC
GND
GND
JP3
M02PTH
JP2
JP1