

28PINS - SCHEMATIC

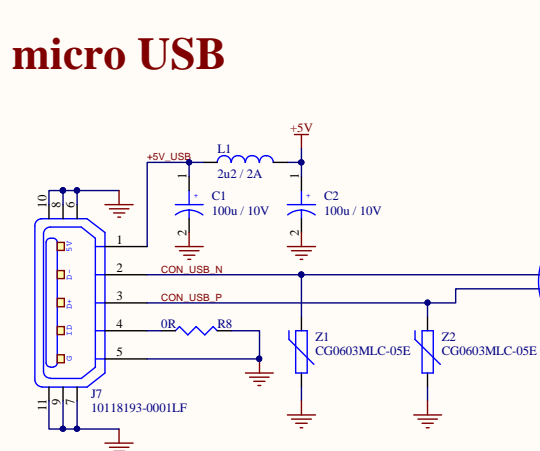
<http://www.28pins.com>

DESIGN NOTE:
 This board supports 5V or 3V3 voltage level on the IO pins:
 1) 5V IO - Fit everything as defined in this schematic. NF means, do not fit this component.
 2) 3.3V IO - Remove R27, Fit R28, *Replace Y1 (change from 16MHz to 10MHz), *Replace Y2 (change from 16MHz to 8MHz).
 3) Both 5V and 3V3, selected through JP4 - Remove R27, Remove R28, Fit JP4, *Replace Y1 (change from 16MHz to 10MHz), *Replace Y2 (change from 16MHz to 8MHz).
 *Note: The 16MHz crystals are not recommended for 3.3V operation. We need to adjust their values, that's why the change.
IMPORTANT: Once you change the crystal value, you may need to re-compile your source code.

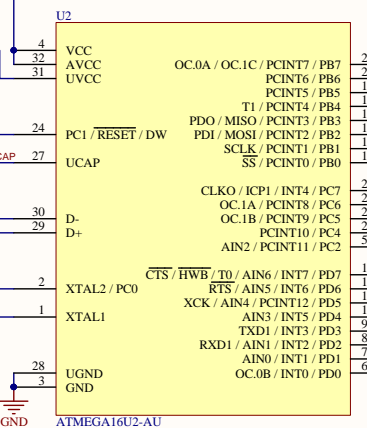
DESIGN NOTE:
 About JP3:
 1) DebugWire support - Short 1&2. This was added to support possible debugWire debugging (programming?) of 328P through 16U2. In this case, the 16U2 needs to have a correct firmware and has to behave as a debugWire tool.
 2) ISP programmer mode - Short 2&3. In this case, take a cable and connect J5 & J6 together. Upload AVRISP MKII firmware into 16U2 and you can program 328P. Example of AVRISP MKII firmware can be found at LUFA projects: <http://www.fourwalledcubicle.com/LUFA.php> (Tip: remap LEDs of the default AVRISP MKII LUFA project to the RX and TX LEDs on the 28Pin board)
 3) ISP header - Short 3 & 4. In this mode, the ICSP1 header is used as a standard ISP header

DESIGN NOTE:
 About JP1:
 1) Autoreset Enabled - Short 1&2. In this case, 16U2 is used to reset 328P when firmware inside 328P is updated from Arduino IDE.
 2) 16U2 DFU mode Enabled - Short 2&3. 16U2 HWB pin is sampled by 16U2 during RESET. If pulled low, then after Reset the 16U2 will go into DFU mode (it's the mode when you can flash 16U2 firmware through USB and Atmel Flip software: <http://www.atmel.com/tools/flip.aspx>).

micro USB



16U2



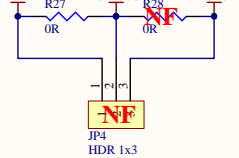
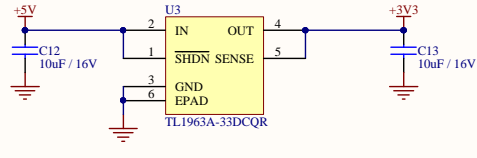
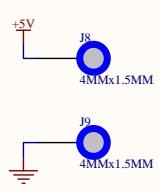
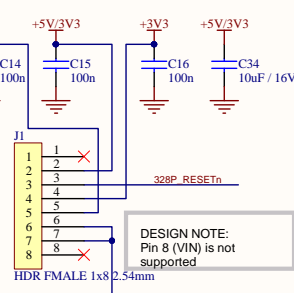
POWER

POWER PADS

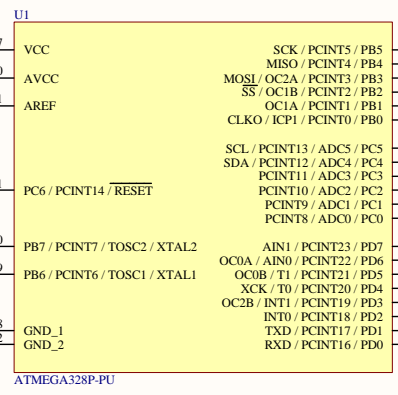
3V3 LDO

Power Selection

DESIGN NOTE:
 This board can be powered from micro USB connector (J7) or a single +3.3V power rail (through J1 pin 4). If +3.3V is used, fit R26 and R28. In this case, JP4 & R27 must NOT be fitted, otherwise the board may be damaged.

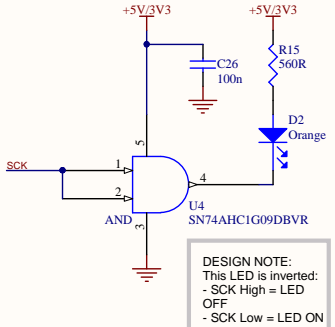
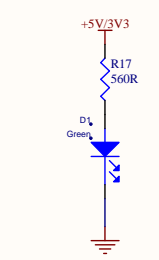


328P



POWER LED

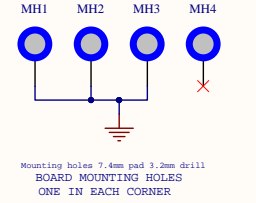
USER LED



DIP SOCKET



MOUNTING HOLES



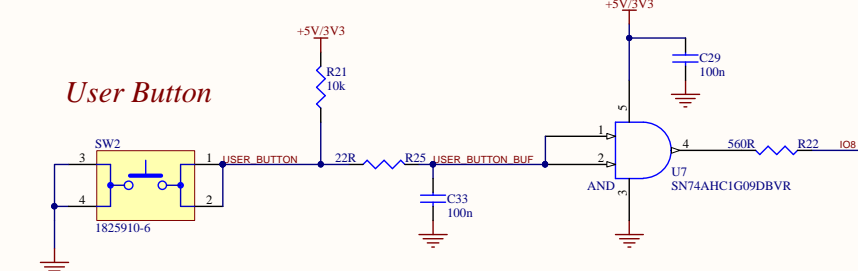
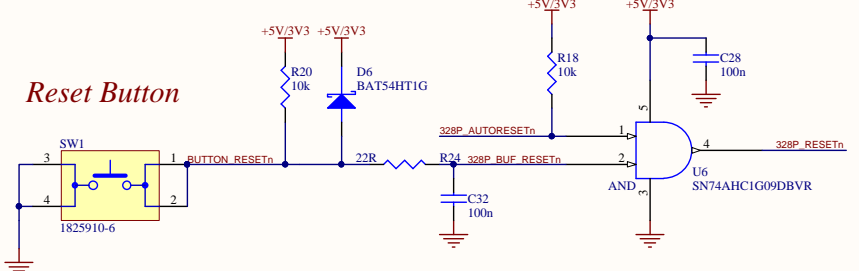
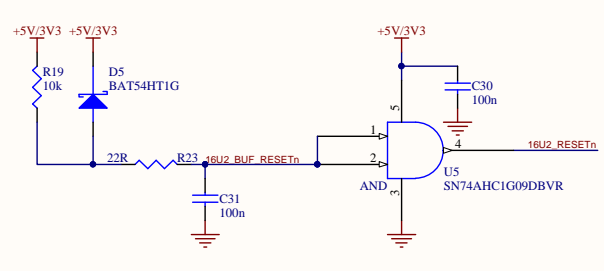
FIDUCIALS



RESET (16U2)

RESET (328P)

USER BUTTON



LAYOUT NOTE:
 1) Route all the POWER tracks with minimum track width 0.4mm.
 2) Route all the other tracks by 0.4mm and change them by the end of the design to 0.2mm. To change all of them at once, use this filter "(not InNet(+)) and not InNet('GND')) and IsTrack and (OnLayer('L1') or OnLayer('L2'))" and then set 0.2mm width in PCB Inspector panel.

FEDEVEL Academy
 Learn Schematic and PCB Design - ONLINE

(c) 2015 FEDEVEL Academy <http://www.fedevel.com/academy>

Title:	28Pins	Variant:	[No Variations]
Page Contents:	[03] - 28PINS SCHEMATIC.SchDoc	Checked by:	
Size:	DWG NO	Revision:	VIII
Date:	3.4.2015	Sheet	3 of 5