My28Pins
Variant: FIXED 5V

RELEASED 03-JUN-2020

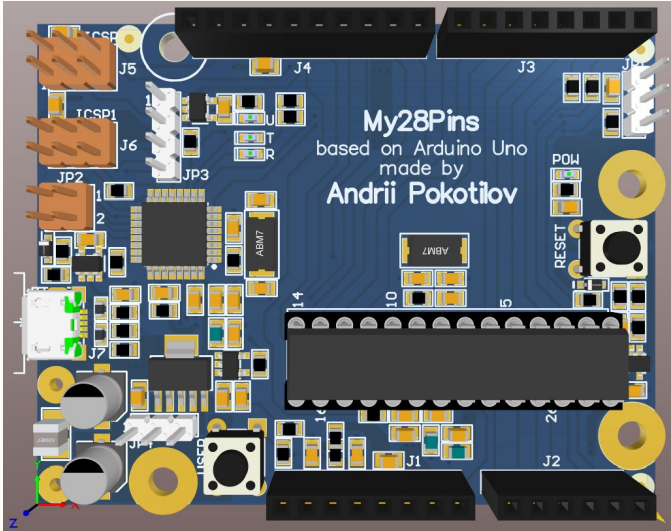**DESIGN CONSIDERATIONS**

DESIGN NOTE:
Example text for debug notes.

DESIGN NOTE:
Example text for informational design notes.

**DESIGN NOTE:
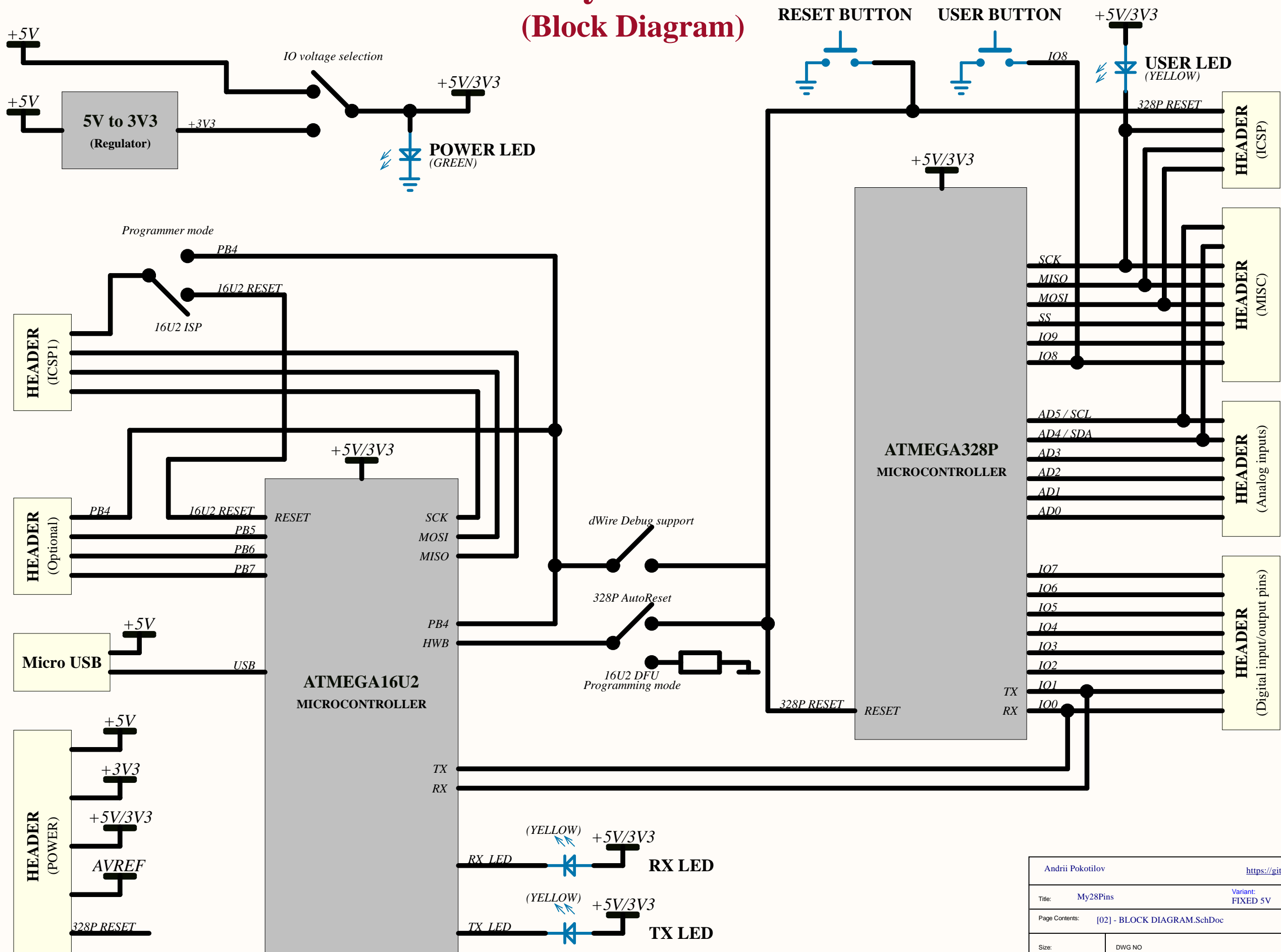Example text for critical design notes.**

DESIGN NOTE:
Example text for cautionary design notes.

**LAYOUT NOTE:
Example text for critical layout guidelines.**

Andrii Pokotilov — https://github.com/Anpokot

Title:	My28Pins	Variant: FIXED 5V	
Page Contents:	[01] - COVER PAGE.SchDoc		Checked by
Size:	DWG NO		Revision: V1I1
Date: 6/5/2020	Designed by Andrii Pokotilov	Sheet 1 of 5	

My28Pins
(Block Diagram)

+5V

+5V

5V to 3V3
(Regulator)
+3V3

IO voltage selection

+5V/3V3

POWER LED
(GREEN)

RESET BUTTON   **USER BUTTON**   **+5V/3V3**

IO8

USER LED
(YELLOW)

328P RESET

HEADER
(ICSP)

Programmer mode

PB4

16U2 RESET

HEADER
(ICSP1)

16U2 ISP

SCK
MISO
MOSI
SS
IO9
IO8

HEADER
(MISC)

HEADER
(Optional)

PB4   *16U2 RESET*   RESET
PB5
PB6
PB7

+5V/3V3

SCK
MOSI
MISO

ATMEGA328P
MICROCONTROLLER

AD5 / SCL
AD4 / SDA
AD3
AD2
AD1
AD0

HEADER
(Analog inputs)

dWire Debug support

Micro USB

+5V

USB

PB4
HWB

328P AutoReset

IO7
IO6
IO5
IO4
IO3
IO2
IO1
IO0

HEADER
(Digital input/output pins)

ATMEGA16U2
MICROCONTROLLER

16U2 DFU
Programming mode

328P RESET   RESET

TX
RX

HEADER
(POWER)

+5V

+3V3

+5V/3V3

AVREF

TX
RX

+5V/3V3

328P RESET

(YELLOW)   **+5V/3V3**

RX_LED   **RX LED**

(YELLOW)   **+5V/3V3**

TX_LED   **TX LED**
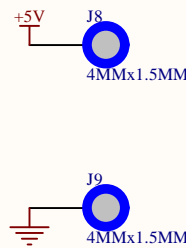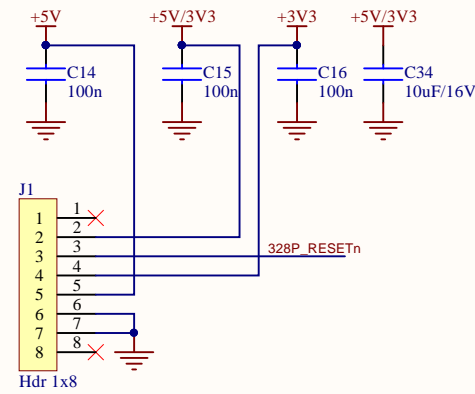
My28Pins - SCHEMATIC

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.
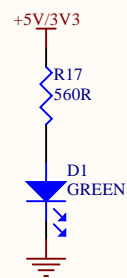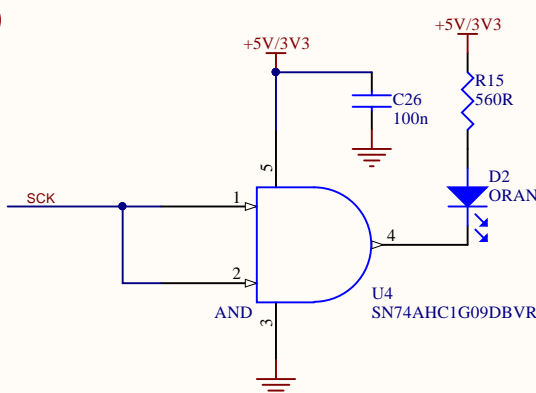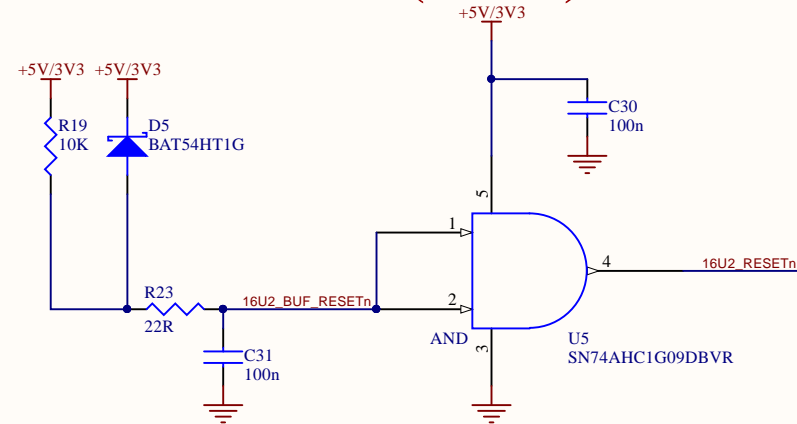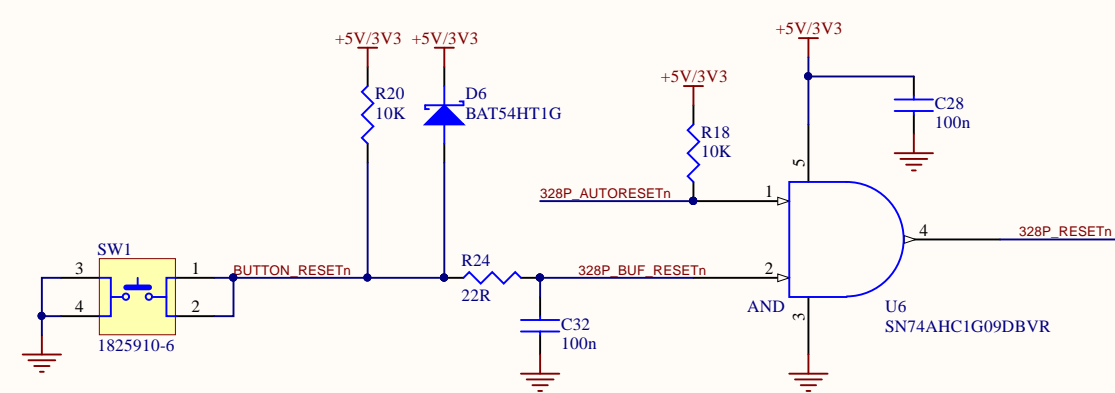
Micro USB
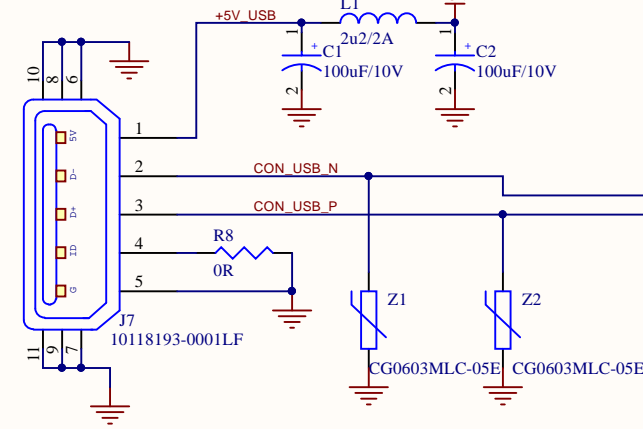
POWER

POWER PADS

POWER LED

USER LED

RESET (16U2)

RESET (328P)

USER BUTTON

16U2

ATMEGA16U2-AU-ND

3V3 LDO

TL1963A-33DCQR

328P

ATMEGA328P-PU
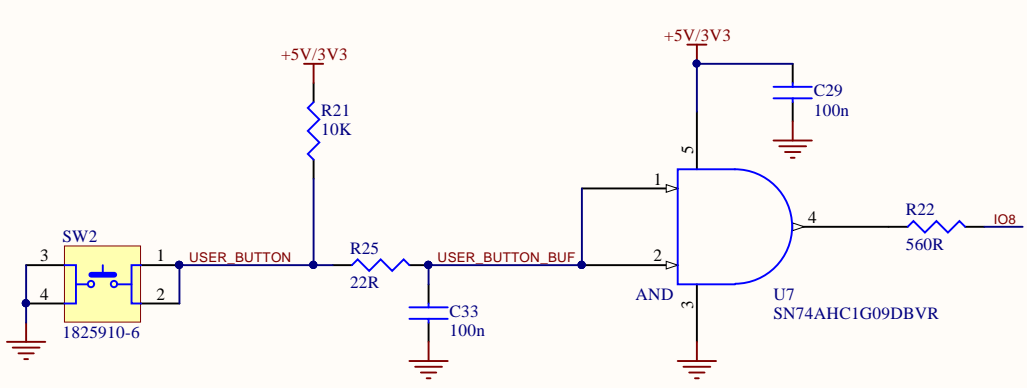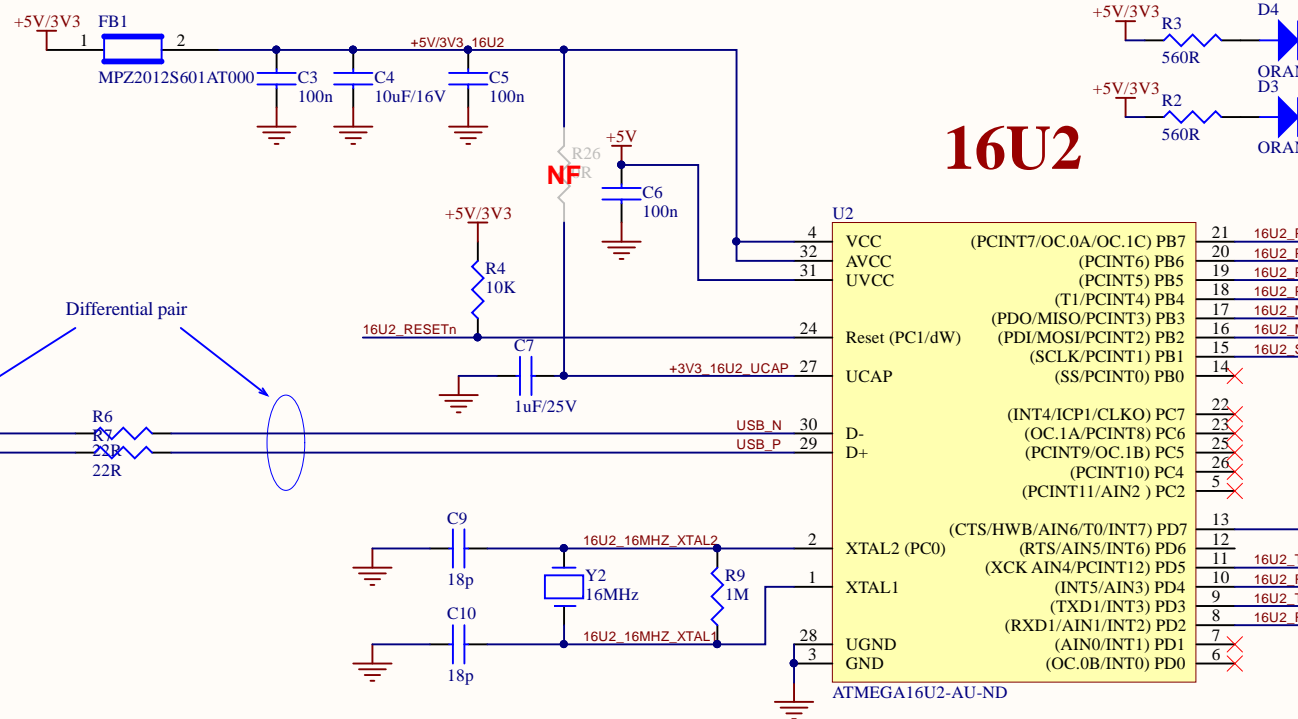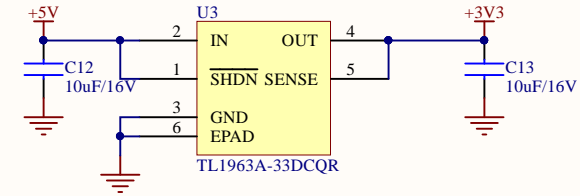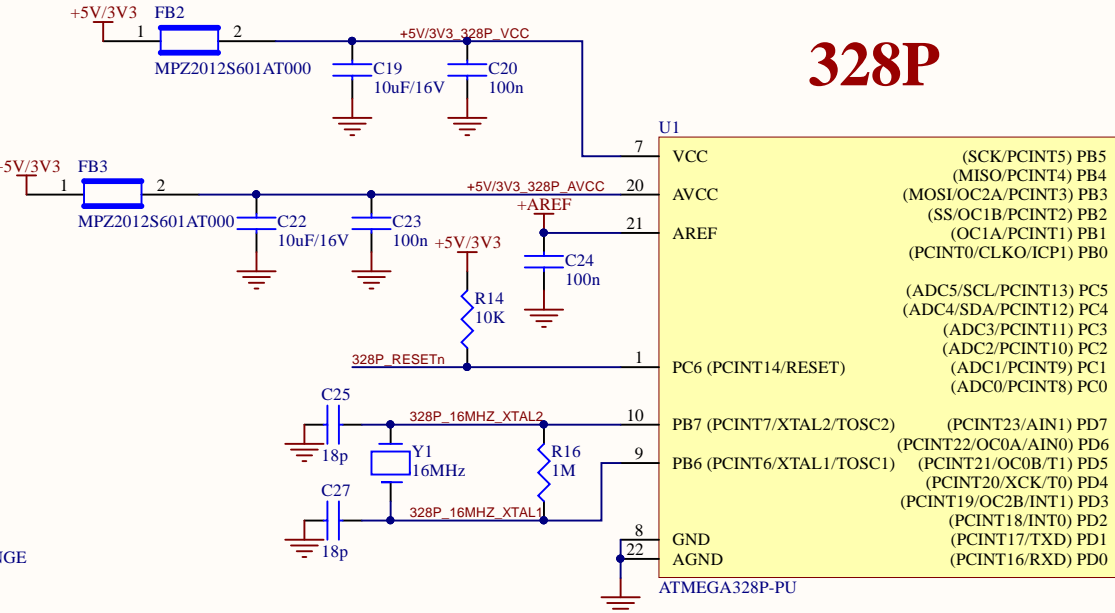
JP2

ICSP1

JP3

JP1

ICSP

IOH

AD

IOL

DIP SOCKET

SOCKET FOR U1

28-DIP SOCKET

28-Dip Socket

MOUNTING HOLES

Mounting holes 7.4mm pad 3.2mm drill
BOARD MOUNTING HOLES
ONE IN EACH CORNER

FIDUCIALS

FIDICIALS 4x TOP

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, thats 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 to program 16U2 through ISP interaface by an ISP programmer.

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).

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.

Andrii Pokotilov — https://github.com/Anpokot

Title: My28Pins
Variant: FIXED 5V
Page Contents: [03] - 28PINS SCHEMATIC.SchDoc
Checked by
Revision: V1I1
Date: 6/5/2020
Designed by Andrii Pokotilov
Sheet 3 of 5

REVISION HISTORY

Designator
[01] - COVER PAGE.SchDoc

Designator
[02] - BLOCK DIAGRAM.SchDoc

Designator
[03] - 28PINS SCHEMATIC.SchDoc

Designator
[04] - REVISION HISTORY.SchDoc

NOTES

Mark Not Fitted Components as
NF

DRAFT - Very early stage of schematic, ignore details.

PRELIMINARY - Close to final schematic.

CHECKED - There should not be any mistakes. Tell the engineer if you find one.

RELEASED - A board with this schematic has been sent to production.

Andrii Pokotilov · https://github.com/Anpokot

Title: My28Pins	Variant: FIXED 5V
Page Contents: 28Pins_Project_V1I1 Project.SchDoc	Checked by
Size:	DWG NO	Revision: V1I1
Date: 6/5/2020	Designed by Andrii Pokotilov	Sheet 5 of 5