Uno Copy

Variant: FIXED 5V

6/4/2020
V1I1

DESIGN CONSIDERATIONS

DESIGN NOTE:
Example text for informational design notes.

DESIGN NOTE:
Example text for debug notes.

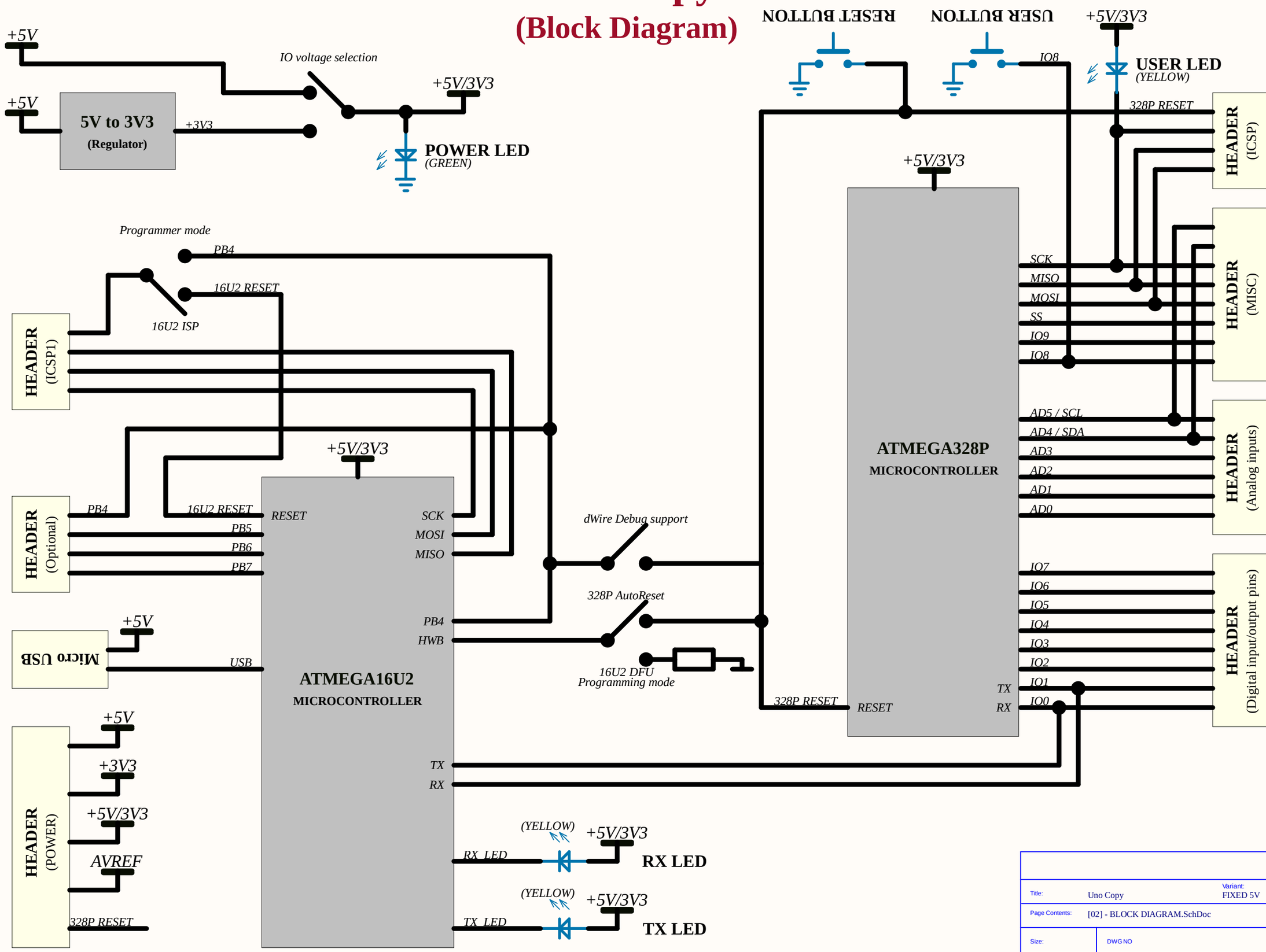
DESIGN NOTE:
Example text for critical design notes.

DESIGN NOTE:
Example text for cautionary design notes.

LAYOUT NOTE:
Example text for critical layout guidelines.

Title:	Uno Copy	Variant:	FIXED 5V
Page Contents:	[01] - COVER PAGE.SchDoc		Checked by
Size:	DWG NO		Revision: V1I1
Date:	6/4/2020	Sheet	1 of 5

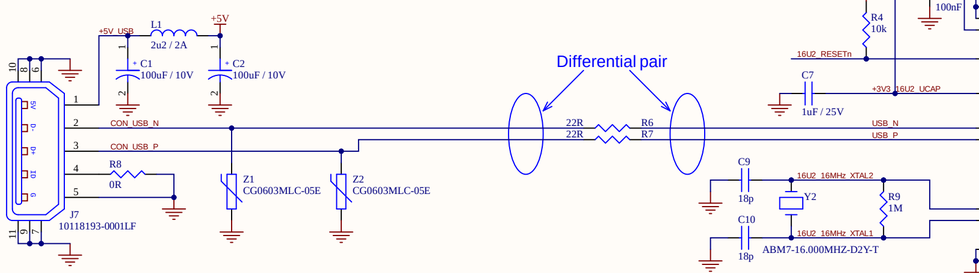
Uno Copy
(Block Diagram)



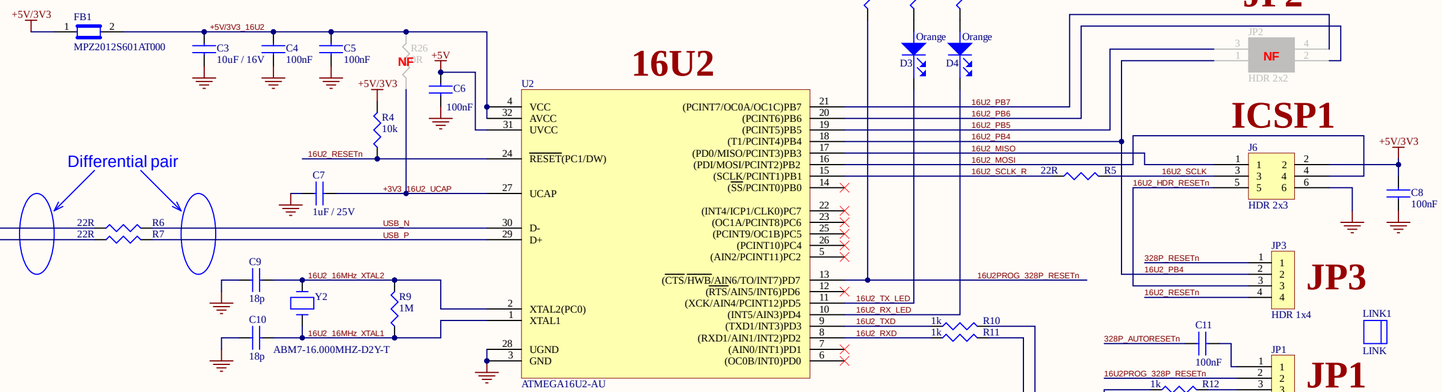
Title:	Uno Copy	Variant: FIXED 5V	
Page Contents:	[02] - BLOCK DIAGRAM.SchDoc	Checked by	
Size:	DWG NO	Revision: V1I1	
Date: 6/4/2020	Sheet 2 of 5		

Arduino UNO - SCHEMATIC

Micro USB



16U2



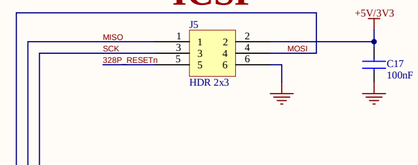
JP2

ICSP1

JP3

JP1

ICSP

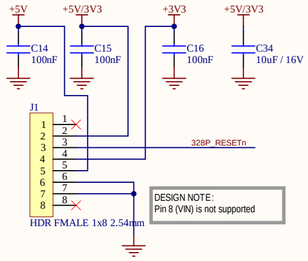


DIP SOCKET

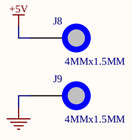
SCKT FOR U1

28-DIP Socket

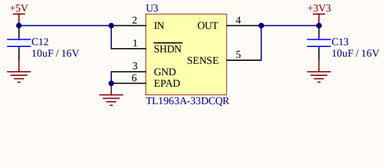
POWER



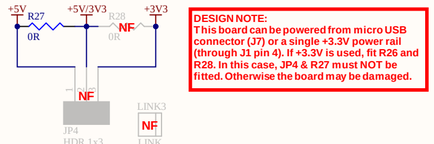
POWER PADS



3V3 LDO



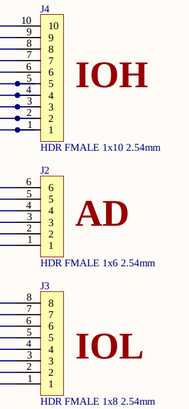
Power Selection



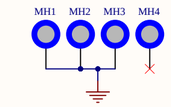
DESIGN NOTE:
This board can be powered from micro USB connector (J7) or a single +3.3V power rail (through J1 pin 4). If +3.3V is used, fit R26 and R28. In this case, JP4 & R27 must NOT be fitted. Otherwise the board may be damaged.

328P

IOH



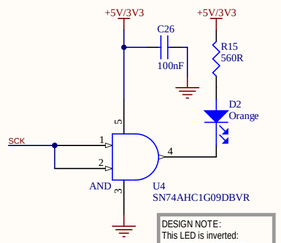
MOUNTING HOLES



FIDUCIALS

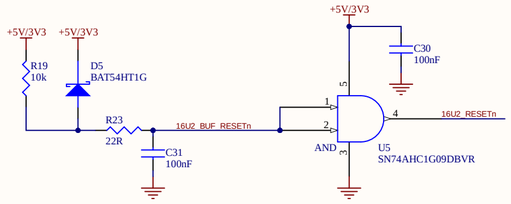
POWER LED

USER LED

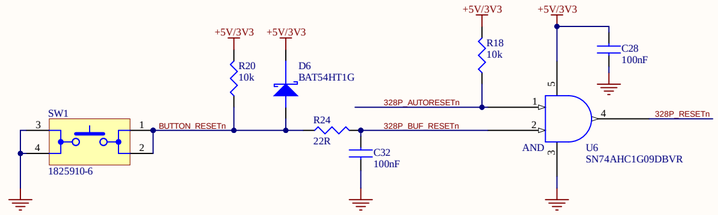


DESIGN NOTE:
This LED is inverted:
- SCK High = LED OFF
- SCK Low = LED ON

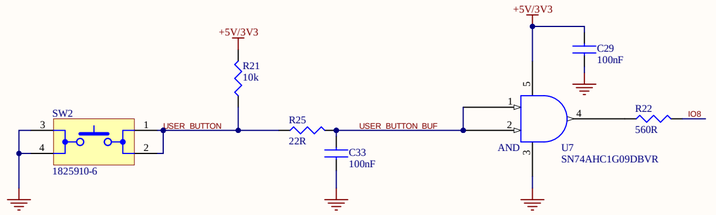
RESET (16U2)



RESET (328P)



USER BUTTON



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.

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. It 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, 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.

Title: Uno Copy		Variant: FIXED 5V
Page Contents: [03] - SCHEMATIC.SchDoc		Checked by
Size:	DWG NO	Revision: V1l1
Date: 6/4/2020		Sheet 3 of 5

REVISION HISTORY

Title:	Uno Copy	Variant: FIXED 5V
Page Contents:	[04] - REVISION HISTORY.SchDoc	Checked by
Size:	DWG NO	Revision: V1I1
Date:	6/4/2020	Sheet 4 of 5

Designator
[01] - COVER PAGE.SchDoc

Designator
[02] - BLOCK DIAGRAM.SchDoc

Designator
[03] - 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.

Title:	Uno Copy	Variant: FIXED 5V	
Page Contents:	UNO_copy V1I1 Project.SchDoc		Checked by
Size:	DWG NO		Revision: V1I1
Date: 6/4/2020		Sheet 5 of 5	