

ALTIUM PCB COURSE (JULY 2024)

Variant: FIXED 5V

**7/24/2024
Trial 1**

RELEASED 24-JULY-2024

Page	Index	Page	Index	Page	Index	Page	Index
1	COVER PAGE	11	21	31
2	BLOCK DIAGRAM	12	22	32
3	28PINS SCHEMATIC	13	23	33
4	REVISION HISTORY	14	24	34
5	15	25	35
6	16	26	36
7	17	27	37
8	18	28	38
9	19	29	39
10	20	30	40

DESIGN CONSIDERATIONS

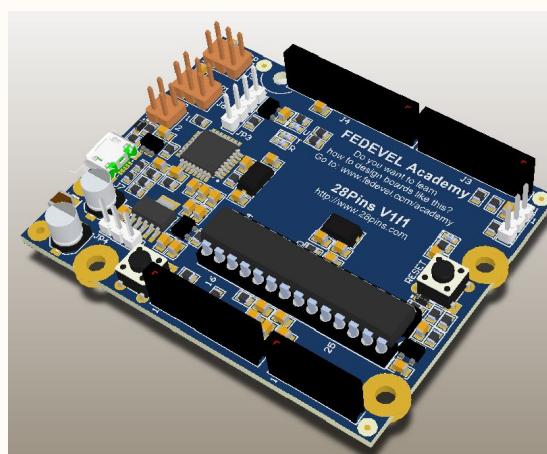
DESIGN NOTE:
Example text for informational design notes.

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.



Would you like to learn how to design boards like this?

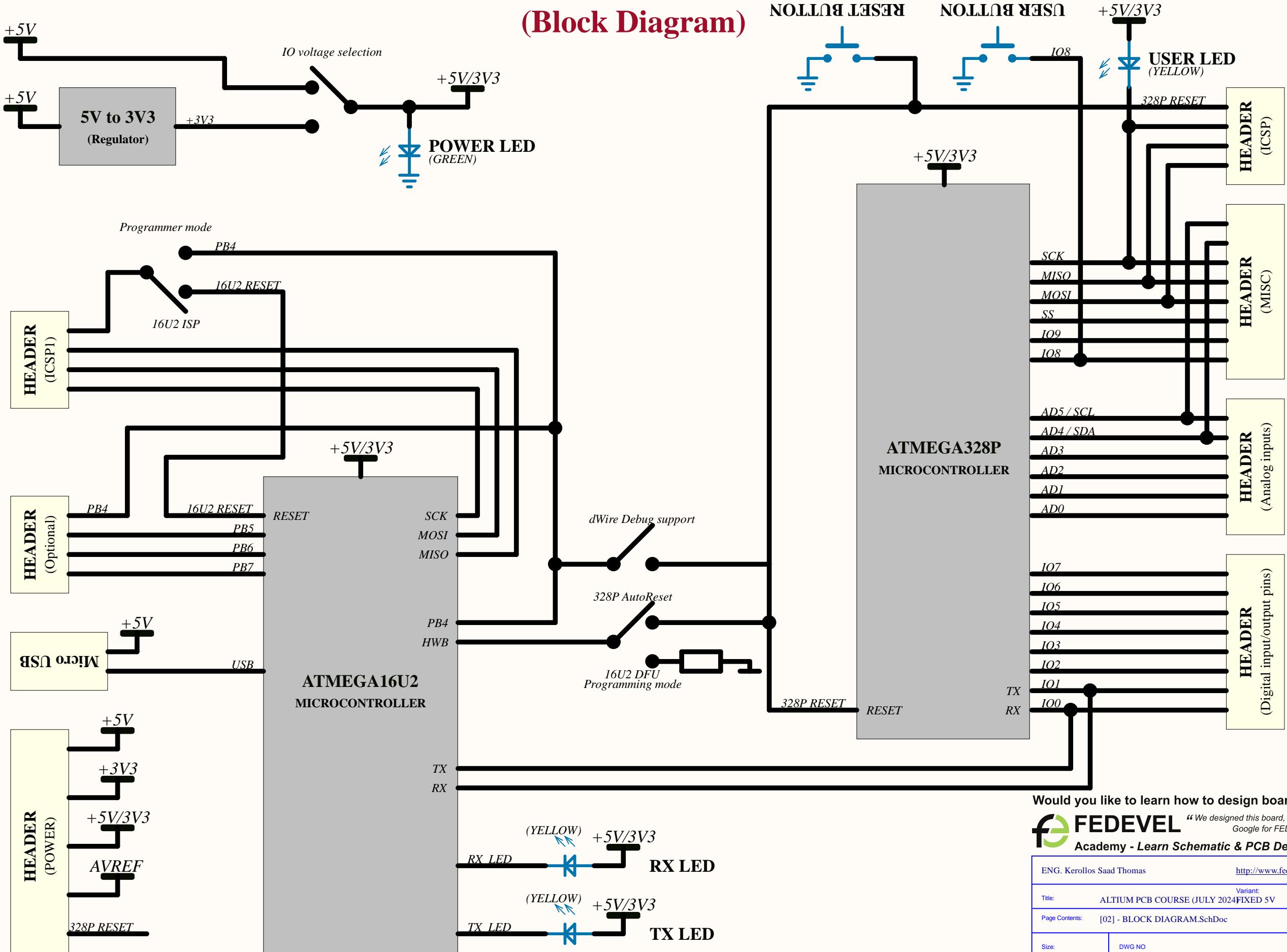
FEDEVEL "We designed this board, so you can learn, Google for FEDEVEL Academy"

Academy - Learn Schematic & PCB Design ONLINE

ENG. Kerollo Saad Thomas	http://www.fedevol.com/academy
Variant:	
Title:	ALTIUM PCB COURSE (JULY 2024) FIXED 5V
Page Contents:	[01] - COVER PAGE.SchDoc Checked by
Size:	DWG NO
Date:	Revision:
7/24/2024	Trial 1
Designed by www.fedevol.com	Sheet 1 of 5

ALTIUM PCB COURSE (JULY 2024)

(Block Diagram)



Would you like to learn how to design boards like this?

FEDEVEL "We designed this board, so you can learn, Google for FEDEVEL Academy"

Academy - Learn Schematic & PCB Design ONLINE

ENG. Kerrollos Saad Thomas	http://www.fedevol.com/academy
Variant:	
Title:	ALTIUM PCB COURSE (JULY 2024) FIXED 5V
Page Contents:	[02] - BLOCK DIAGRAM.SchDoc
Checked by:	
Size:	DWG NO
Revision:	Trial 1
Date: 7/24/2024	Designed by www.fedevol.com
Sheet 2 of 5	

28PINS - SCHEMATIC (JULY 2024)

My Version of the 28pins project by FEDEVEL

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

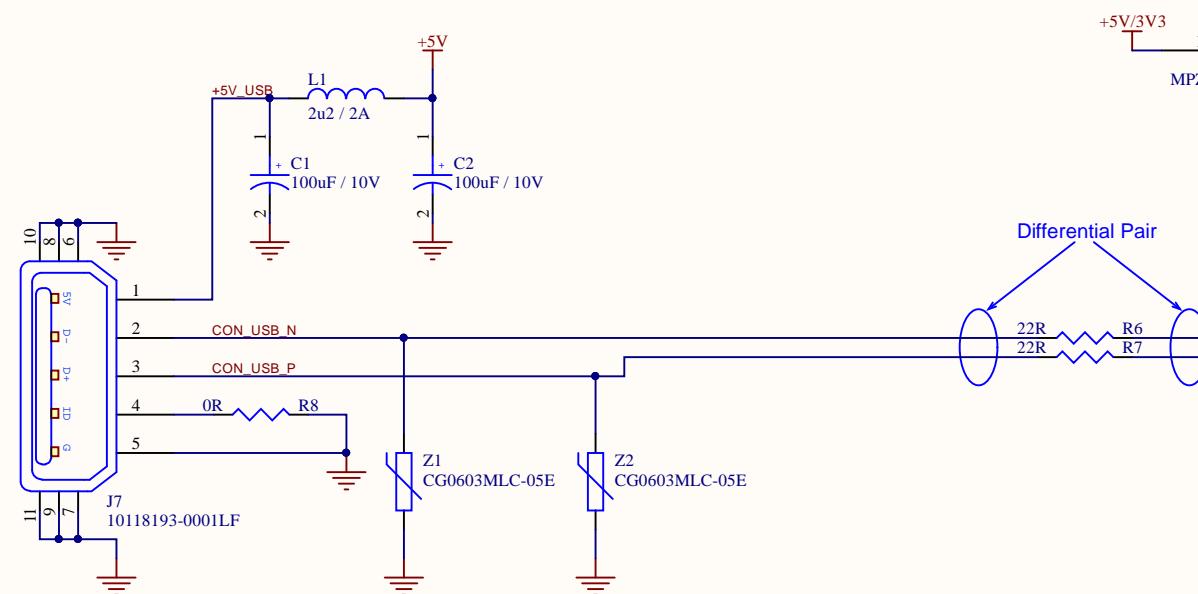
Design Notes

DESIGN NOTE:
This board supports 5V or 3V 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.

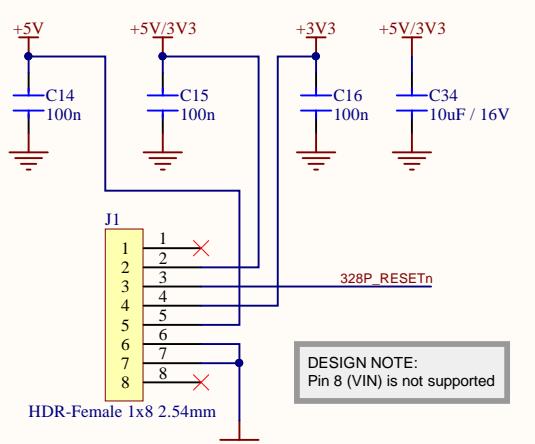
DESIGN NOTE:
About JP3:
1) DebugWire support - Short 1&2. This was added to support possible debugWire debugging (programming) of 328P through 16U2. In this case, the 16U2 needs to have a correct firmware and has to behave as a debugWire tool.
2) ISP programmer mode - Short 2&3. In this case, take a cable and connect 328P together. Upload AVRISP MKII firmware to 16U2 and you can program 328P. Example of AVRISP MKII firmware can be found in LUFA project (<http://www.foundwiredude.com/LUFA.php>). 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 interface by an ISP programmer.

DESIGN NOTE:
About JP1:
1) AutoReset Enabled - Short 2&3. 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 HWIO pin is sampled by 328P during RESET. If it 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>)

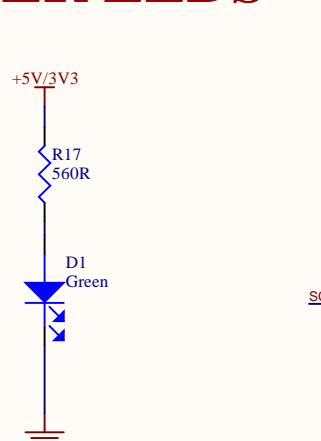
Micro USB



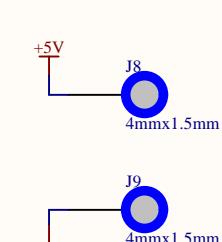
POWER



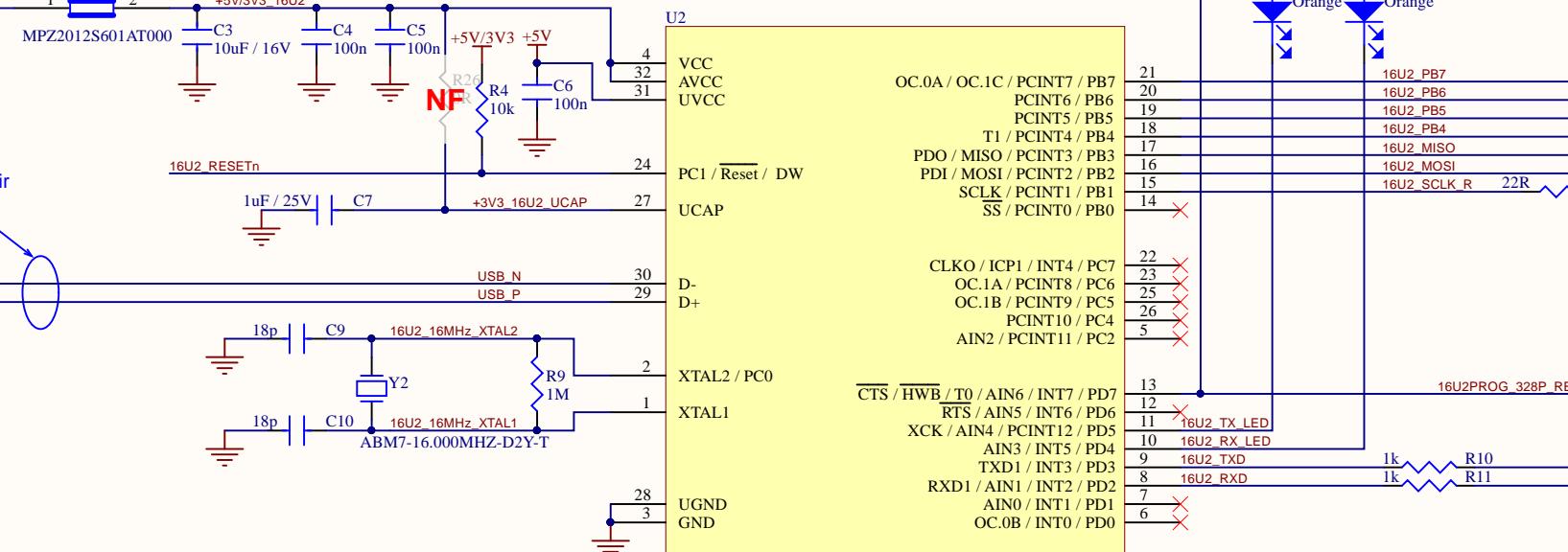
POWER LEDS



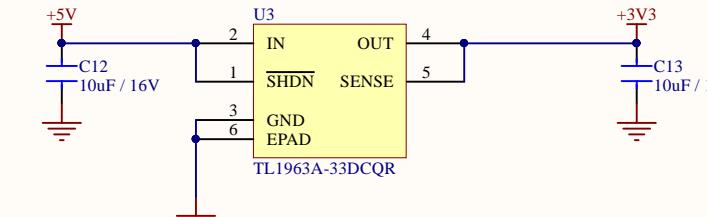
POWER PADS



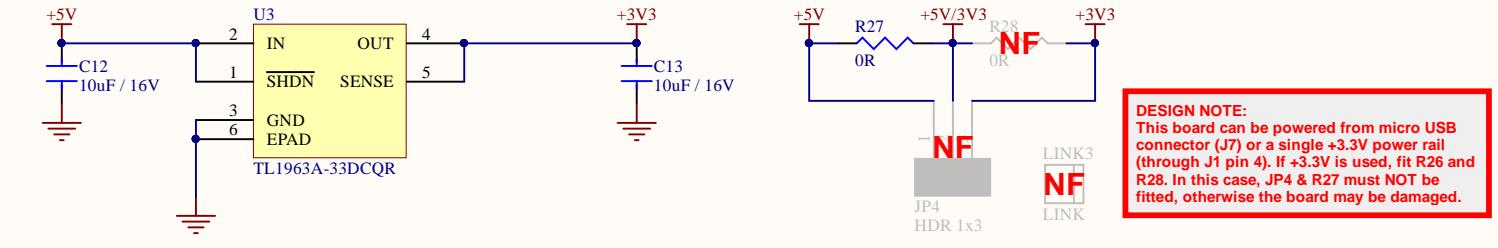
16U2



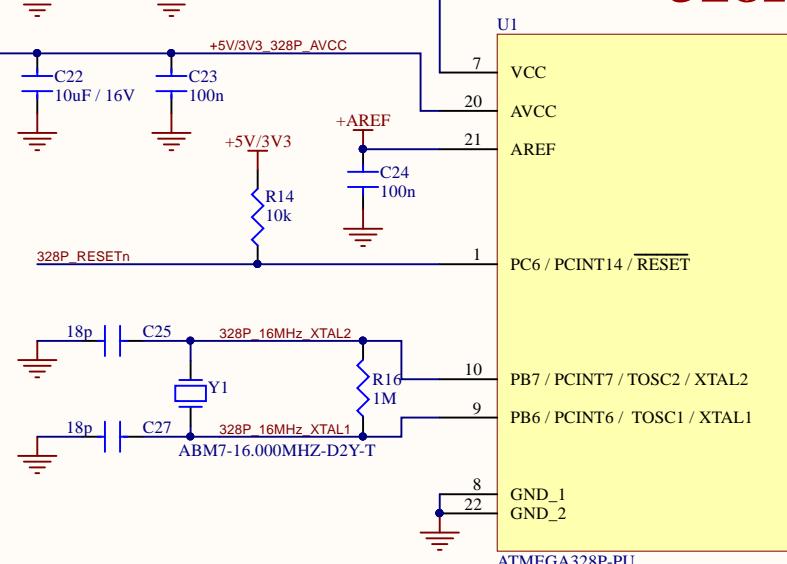
3V3 LDO



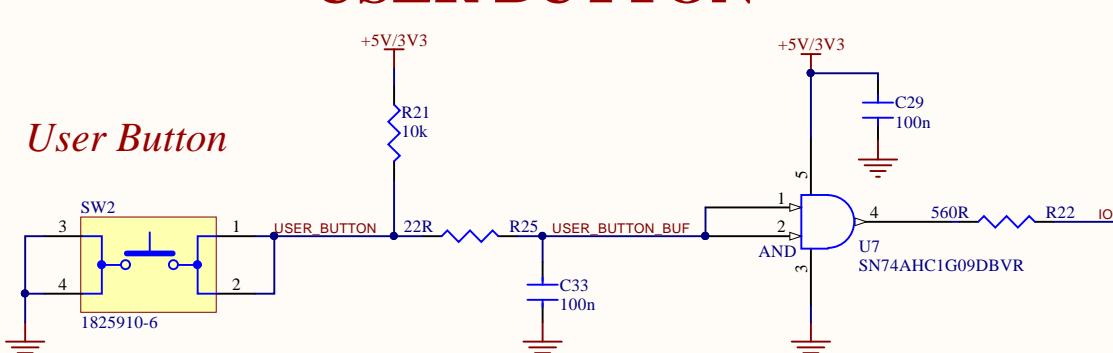
Power Selection



328P



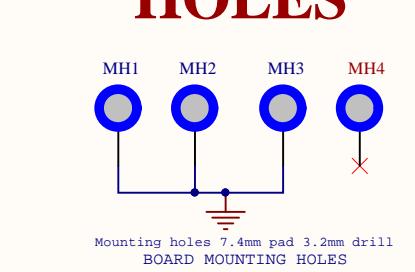
USER BUTTON



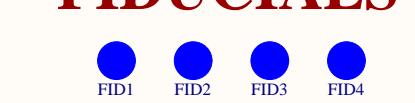
DIP SOCKET



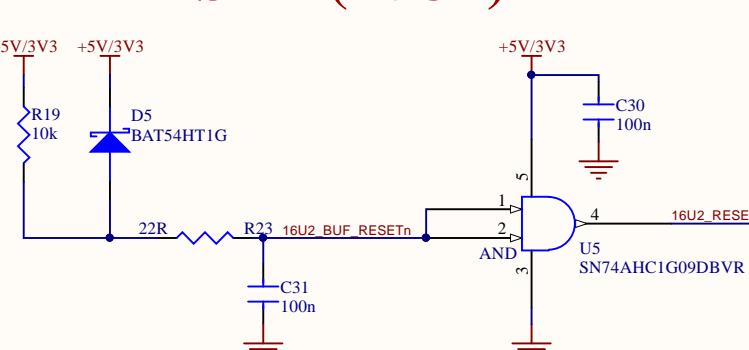
MOUNTING HOLES



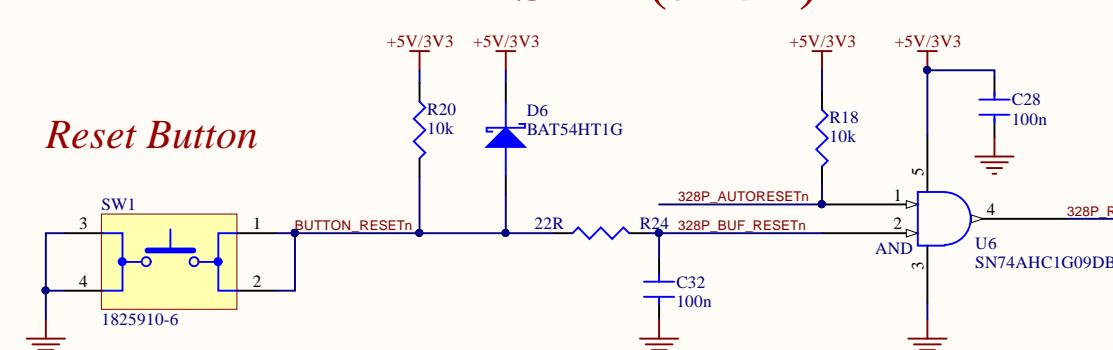
FIDUCIALS



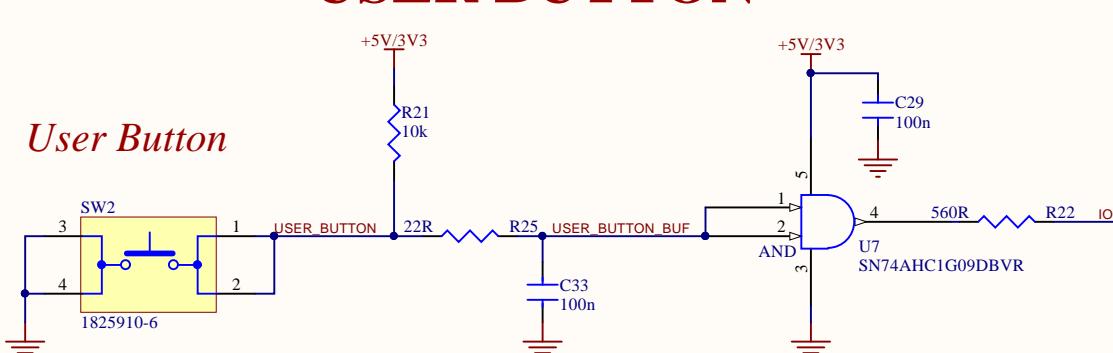
RESET (16U2)



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.

ENG. Kerilos Saad Thomas

Title: ALTIUM PCB COURSE (JULY 2024) Variant: FIXED 5V

Page Contents: [03] - 28PINS SCHEMATIC.SchDoc Checked by <http://www.fedevol.com>

Size: A2 DWG NO: Trial 1

Date: 7/24/2024 Designed by www.fedevol.com Sheet 3 of 5

REVISION HISTORY

A A

B B

C C

D D

Would you like to learn how to design boards like this?



Academy - Learn Schematic & PCB Design ONLINE

ENG. Kerollos Saad Thomas	http://www.fedevol.com/academy
Variant:	
Title:	ALTIUM PCB COURSE (JULY 2024) FIXED 5V
Page Contents:	[04] - REVISION HISTORY.SchDoc
Checked by:	
Size:	DWG NO
Revision:	Trial 1
Date:	7/24/2024
Sheet	4 of 5

Designator
[01] - COVER PAGE.SchDoc

Designator
[02] - BLOCK DIAGRAM.SchDoc

Designator
[03] - 28PINS SCHEMATIC.SchDoc

Designator
[04] - REVISION HISTORY.SchDoc

A

A

B

B

C

C

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.

Would you like to learn how to design boards like this?

 **FEDEVEL** "We designed this board, so you can learn,
Google for FEDEVEL Academy"

Academy - Learn Schematic & PCB Design ONLINE

ENG. Kerollos Saad Thomas	http://www.fedevol.com/academy
Variant:	
Title:	ALTIUM PCB COURSE (JULY 2024) FIXED 5V
Page Contents:	28Pins_Project_VIII Project.SchDoc
Checked by:	
Size:	DWG NO
Revision:	Trial 1
Date: 7/24/2024	Designed by www.fedevol.com
Sheet 5 of 5	