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.

Would you like to learn how to design boards like this?
FEDEVEL Academy - Learn Schematic & PCB Design ONLINE
We designed this board, so you can learn.
Google for FEDEVEL Academy

http://www.fedevel.com/academy

Title:	Variant: 5V/3V3	
Page Contents: 28Pins_Project_V1I1 Project.SchDoc		Checked by
Size:	DWG NO	Revision:
Date: 27-12-2024	Designed by www.fedevel.com	Sheet 1 of 5

27-12-2024

Variant: 5V/3V3

27 DEC 2024

DESIGN CONSIDERATIONS

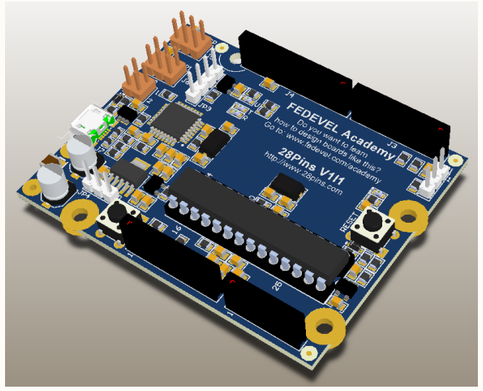
DESIGN NOTE:
Example text for informational design notes.

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

http://www.fedevel.com/academy

Title:	Variant: 5V/3V3	
Page Contents: [01] - COVER PAGE.SchDoc		Checked by
Size:	DWG NO	Revision:
Date: 27-12-2024	Designed by www.fedevel.com	Sheet 2 of 5

THIS DOCUMENT AND THE DATA DISCLOSED HEREIN OR HEREWITH IS THE PROPERTY OF ALTIUM LIMITED AND MAY BE FREELY DISTRIBUTED IN WHOLE. NO RIGHTS ARE RESERVED OR EXPRESS OR IMPLIED WARANTEE GIVEN.

REVISION	DESCRIPTION	DATE	APPROVED

28PINS - SCHEMATIC

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 to10MHz), *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 AVRISPMKII firmware into 16U2 and you can program 328P. (Tip: remap LEDs of the default AVRISPMKII LUFA project to the RX and TX LEDs on the 28Pin board)

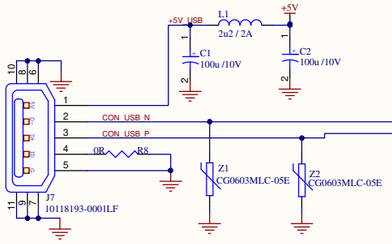
3) ISP header - Short 3 & 4. In this mode, the ICSP1 header is used as a standard ISP header to program 16U2 through ISP interaface by an ISP programmer.

DESIGN NOTE:

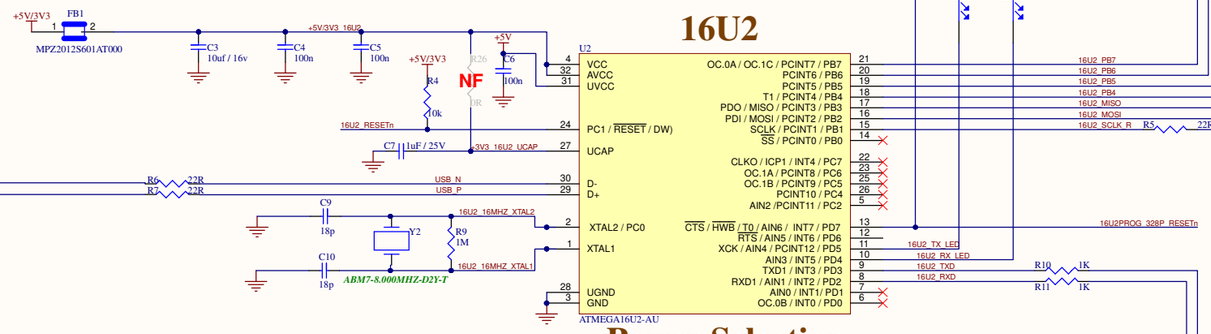
About JP1:
1) Autoreset Enabled - Short 1&2. In this case, 16U2 is used to reset 328P when firmware inside 328P is updated from Arduino IDE.

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.

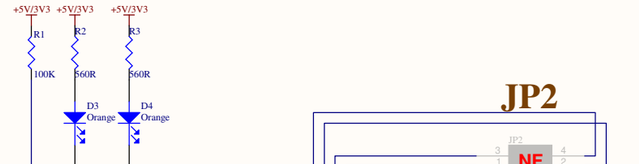
Micro USB



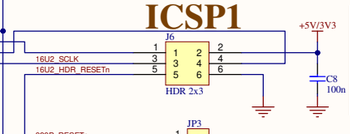
16U2



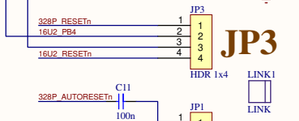
JP2



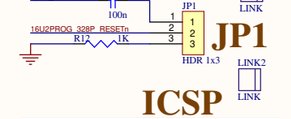
ICSP1



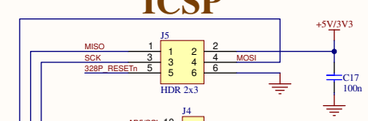
JP3



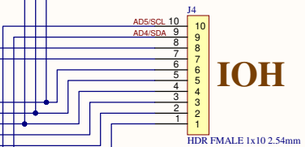
JP1



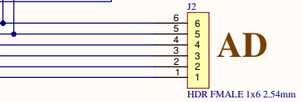
ICSP



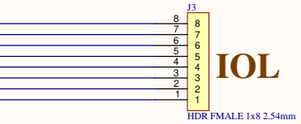
IOH



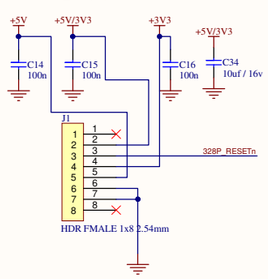
AD



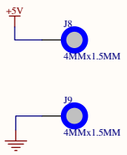
IOL



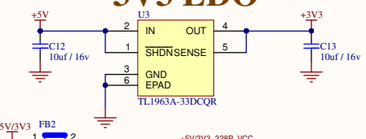
POWER



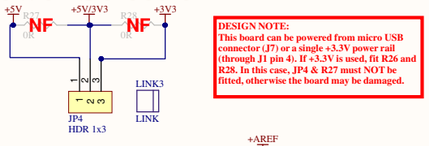
POWER PADS



3V3 LDO



Power Selection

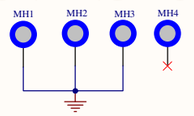


DESIGN NOTE:
This board can be powered from micro USB connector (J7) or a single +3.3V power rail (through J1 pin 4). If +3.3V is used, fit R26 and R28. In this case, JP4 & R27 must NOT be fitted, otherwise the board may be damaged.

328P

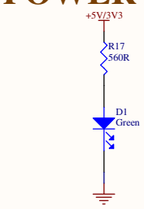
DIP SOCKET

MOUNTING HOLES

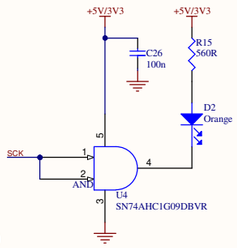


FIDUCIALS

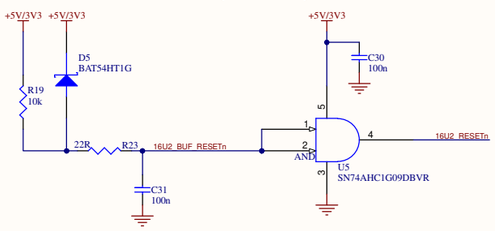
POWER LED



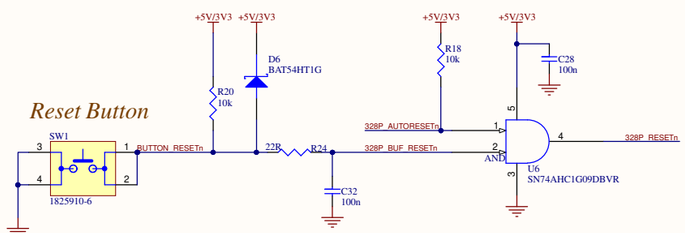
USER LED



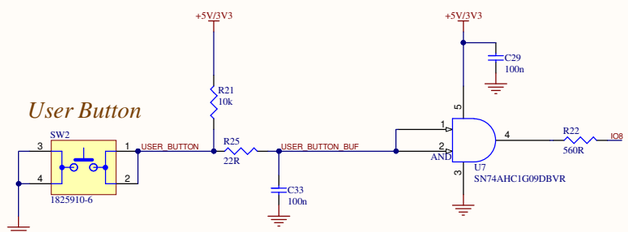
RESET (16U2)



RESET (328P)



USER BUTTON



LAYOUT NOTE:

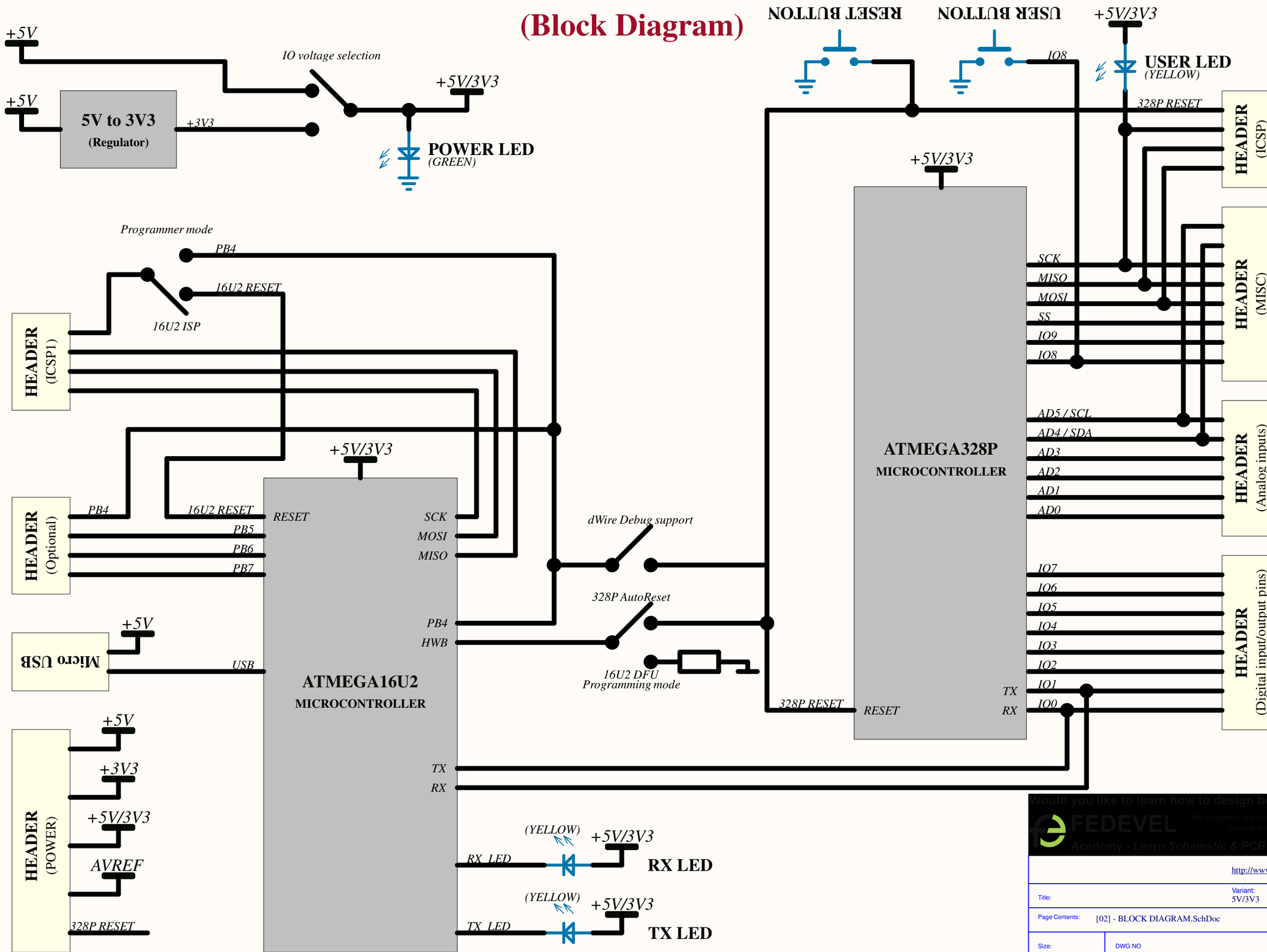
1) Route all the POWER tracks with minimum track width 0.4mm.

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.

APPROVALS	DATE
ENG: .	
DSN: .	
CHK: .	
REFERENCE DOCUMENTS	
BOM:	
ASSY DWG:	
FAB DWG:	
PCB DWG:	

PROJECT: Altium
PROJECT REVISION: Not in version control
DOCUMENT REVISION: Not in version control
DESIGN ITEM:
TITLE: 28Pins
SIZE: A2
CAGE CODE:
DWG NO.
REV
SCALE:
FILE NAME: [03] - 28PINS SCHEMATIC.SchDoc
SHEET 3 OF

(Block Diagram)



Would you like to learn how to design boards like this?
FEDEVEL Academy - Learn Schematic & PCB Design ONLINE
We designed this board, so you can learn.
Google for FEDEVEL Academy

	http://www.fedevel.com/academy
Title:	Variant: 5V/3V3
Page Contents:	[02] - BLOCK DIAGRAM.SchDoc
Size:	DWG NO / Revision:
Date: 27-12-2024	Designed by www.fedevel.com / Sheet 4 / Checked by

REVISION HISTORY

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

	http://www.fedevel.com/academy	
Title:	Variant: 5V/3V3	
Page Contents: [04] - REVISION HISTORY.SchDoc		Checked by
Size:	DWG NO	Revision:
Date: 27-12-2024	Sheet 5 of 5	