28Pins
Variant: FIXED 5V

RELEASED 03-JUL-2025

DESIGN NOTE:
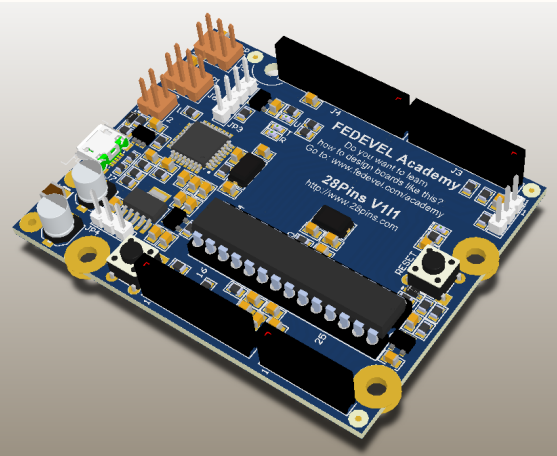Example text for debug notes.

DESIGN CONSIDERATIONS

DESIGN NOTE:
Example text for informational
design notes.

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

DESIGN NOTE:
Example text for cautionary
design notes.

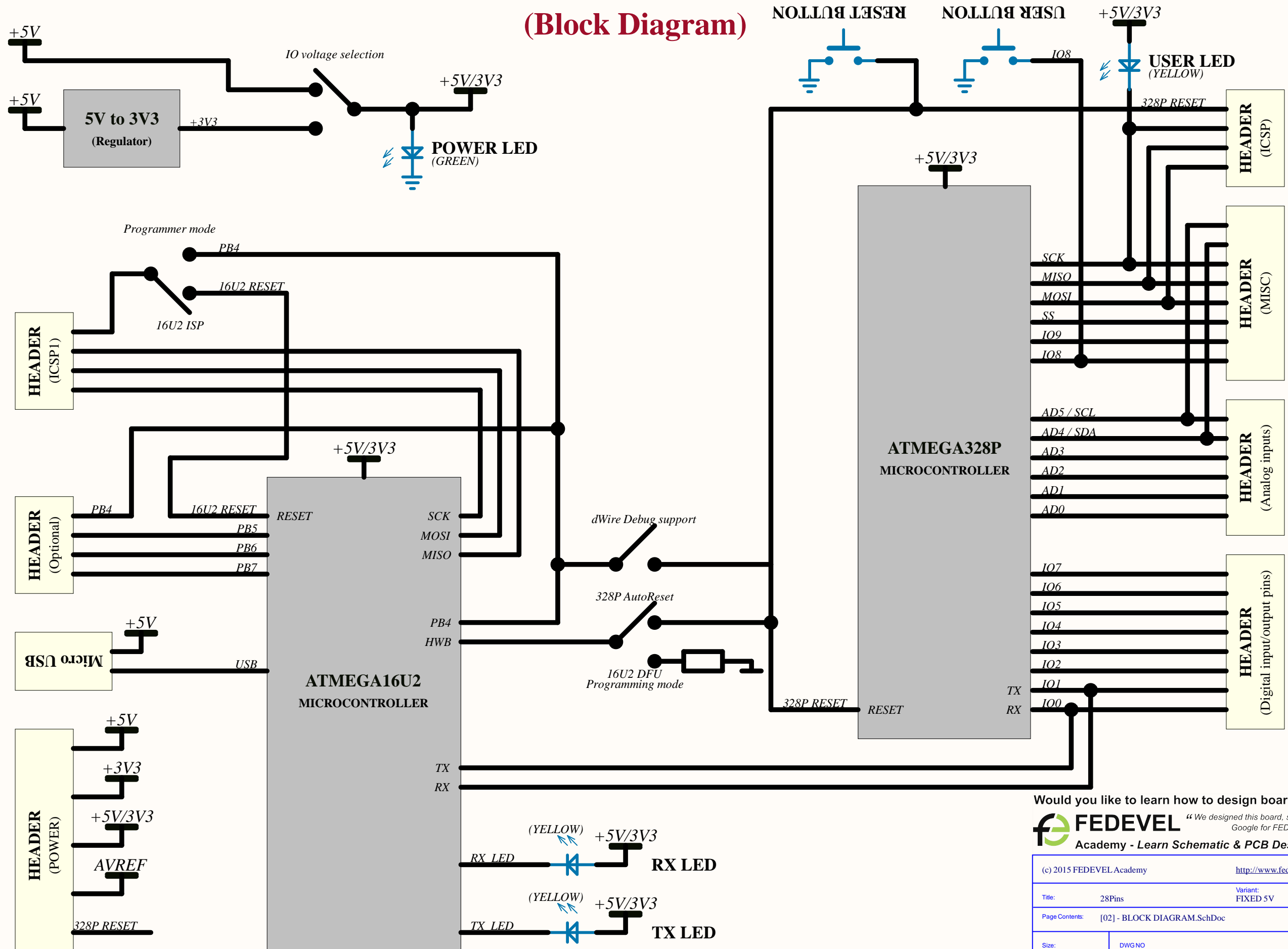**LAYOUT NOTE:
Example text for critical
layout guidelines.**

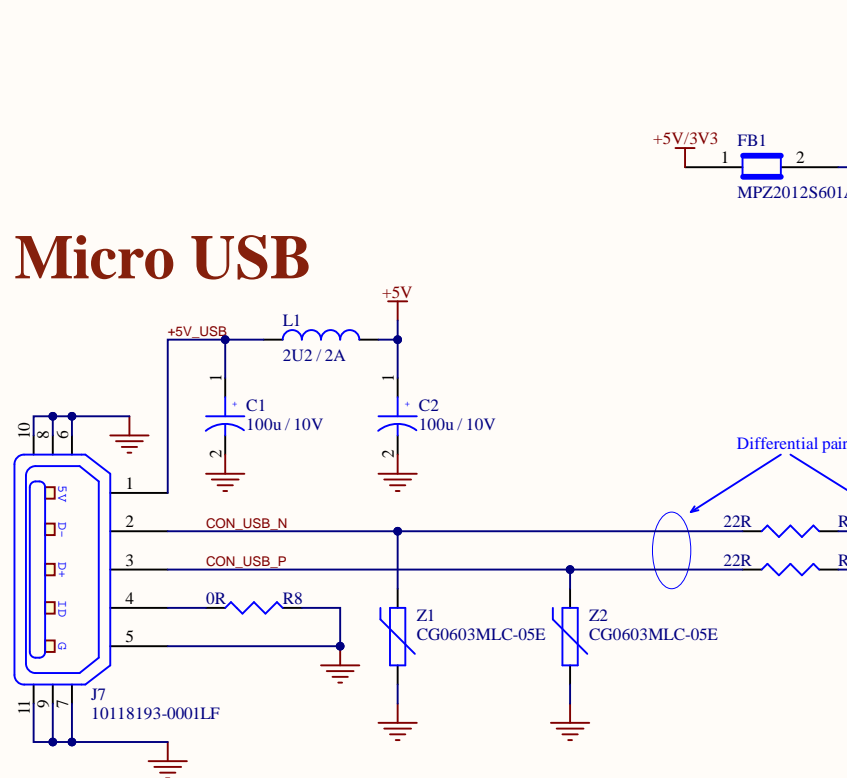**Would you like to learn how to design boards like this?**

FEDEVEL
" We designed this board, so you can learn,
Google for FEDEVEL Academy "
Academy - *Learn Schematic & PCB Design ONLINE*
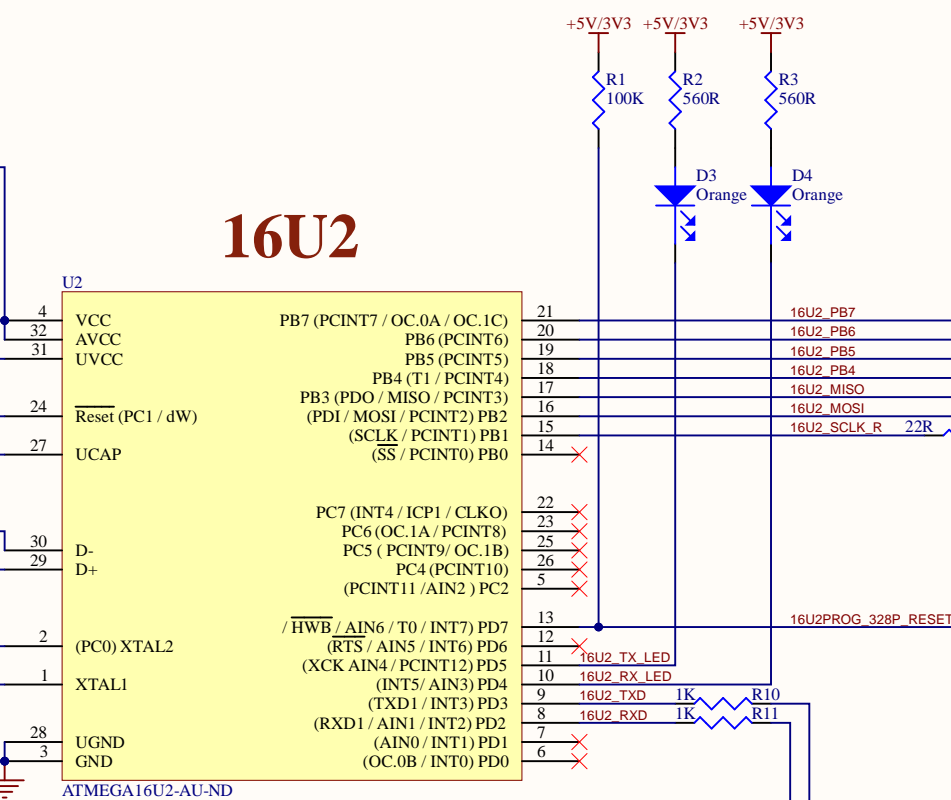
Title: 28Pins	Variant: FIXED 5V	
Page Contents: [01] - COVER PAGE.SchDoc	Checked by	
Size:	DWG NO	Revision: V1I1
Date: 7/2/2025	Designed by www.fedevel.com	Sheet 1 of 5

28Pins
(Block Diagram)

RESET BUTTON

USER BUTTON

+5V/3V3

IO8

USER LED
(YELLOW)

328P RESET

IO voltage selection

+5V

+5V

5V to 3V3
(Regulator)

+3V3

+5V/3V3

POWER LED
(GREEN)

+5V/3V3

HEADER
(ICSP)

HEADER
(MISC)

SCK
MISO
MOSI
SS
IO9
IO8

AD5 / SCL
AD4 / SDA
AD3
AD2
AD1
AD0

HEADER
(Analog inputs)

ATMEGA328P
MICROCONTROLLER

Programmer mode

PB4

16U2 RESET

16U2 ISP

HEADER
(ICSP1)

+5V/3V3

dWire Debug support

328P AutoReset

HEADER
(Optional)

PB4
PB5
PB6
PB7

16U2 RESET
RESET

SCK
MOSI
MISO

PB4
HWB

16U2 DFU
Programming mode

328P RESET

RESET

IO7
IO6
IO5
IO4
IO3
IO2
IO1
IO0

TX
RX

HEADER
(Digital input/output pins)

+5V

Micro USB

USB

ATMEGA16U2
MICROCONTROLLER

+5V
+3V3
+5V/3V3

AVREF

HEADER
(POWER)

328P RESET

TX
RX

(YELLOW)
+5V/3V3

RX_LED

RX LED

(YELLOW)
+5V/3V3

TX_LED

TX LED

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

28PINS - SCHEMATIC

Micro USB

16U2
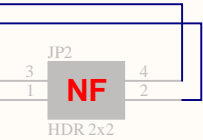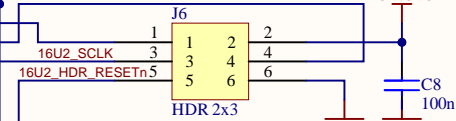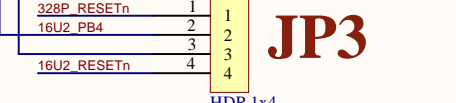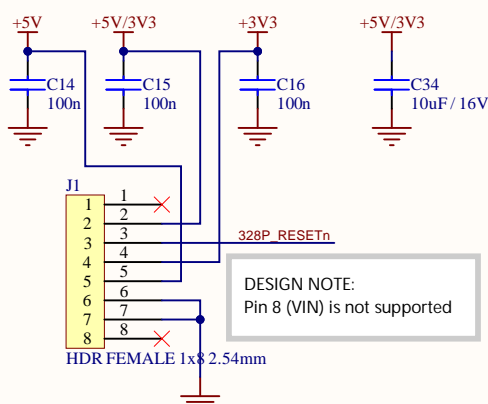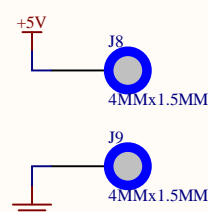
POWER

POWER PADS

3V3 LDO
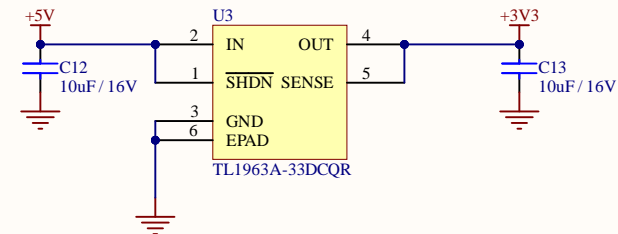
Power Selection
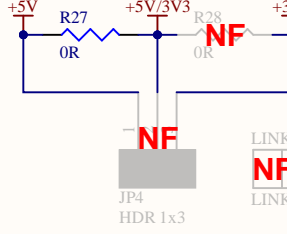
328P

POWER LED

USER LED

RESET (16U2)

RESET (328P)

USER BUTTON

DIP SOCKET

MOUNTING HOLES

FIDUCIALS

ICSP1

JP2

JP3

JP1

ICSP

IOH

AD

IOL

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 interface 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).
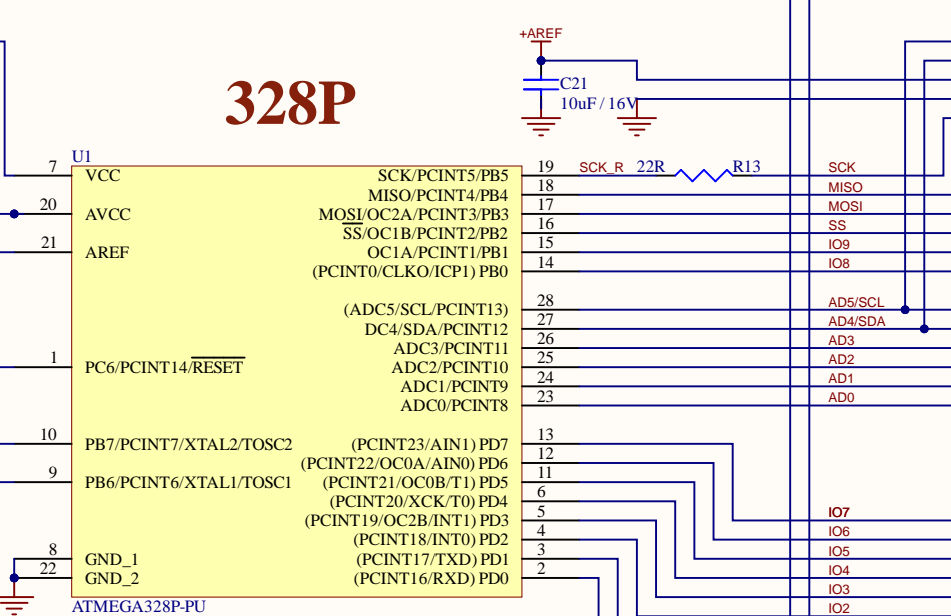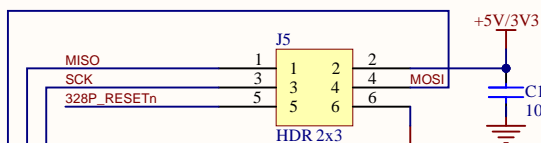
DESIGN NOTE:
This board can be powered from micro USB connector (J7) or a single +3.3V power rail (through J1 pin 4). If +3.3V is used, fit R26 and R28. In this case, JP4 & R27 must NOT be fitted, otherwise the board may be damaged.
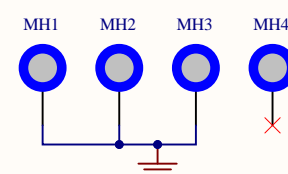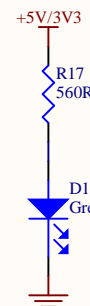
DESIGN NOTE:
Pin 8 (VIN) is not supported

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

Reset Button

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.

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

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

Academy - Learn Schematic & PCB Design ONLINE

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

Title:	28Pins	Variant: FIXED 5V

Page Contents: [03] - 28PINS SCHEMATIC.SchDoc   Checked by

Size:	DWG NO	Revision: V111

Date: 7/2/2025   Designed by www.fedevel.com   Sheet 3 of 5

28-DIP SOCKET

SCKT For U

28-DIP Socket

BOARD MOUNTING HOLES
ONE IN EACH CORNER

Mounting holes 7.4mm pad 3.2mm drill

MH1 MH2 MH3 MH4

FID1 FID2 FID3 FID4

FIDICIALS 4x TOP

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*

Title:	28Pins
Variant:	FIXED 5V
Page Contents:	[04] - REVISION HISTORY.SchDoc Checked by
Size:	DWG NO
Revision:	V1I1
Date:	7/2/2025

Sheet 4 of 5

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.

Title: 28Pins

Variant:
FIXED 5V

Page Contents: 28Pins_Project_V1I1 Project.SchDoc   Checked by

Size:

DWG NO

Revision:
V1I1

Date: 7/2/2025   Designed by www.fedevel.com   Sheet 5 of 5