Arduino
Variant: [No Variations]

3/1/2020
V1I1

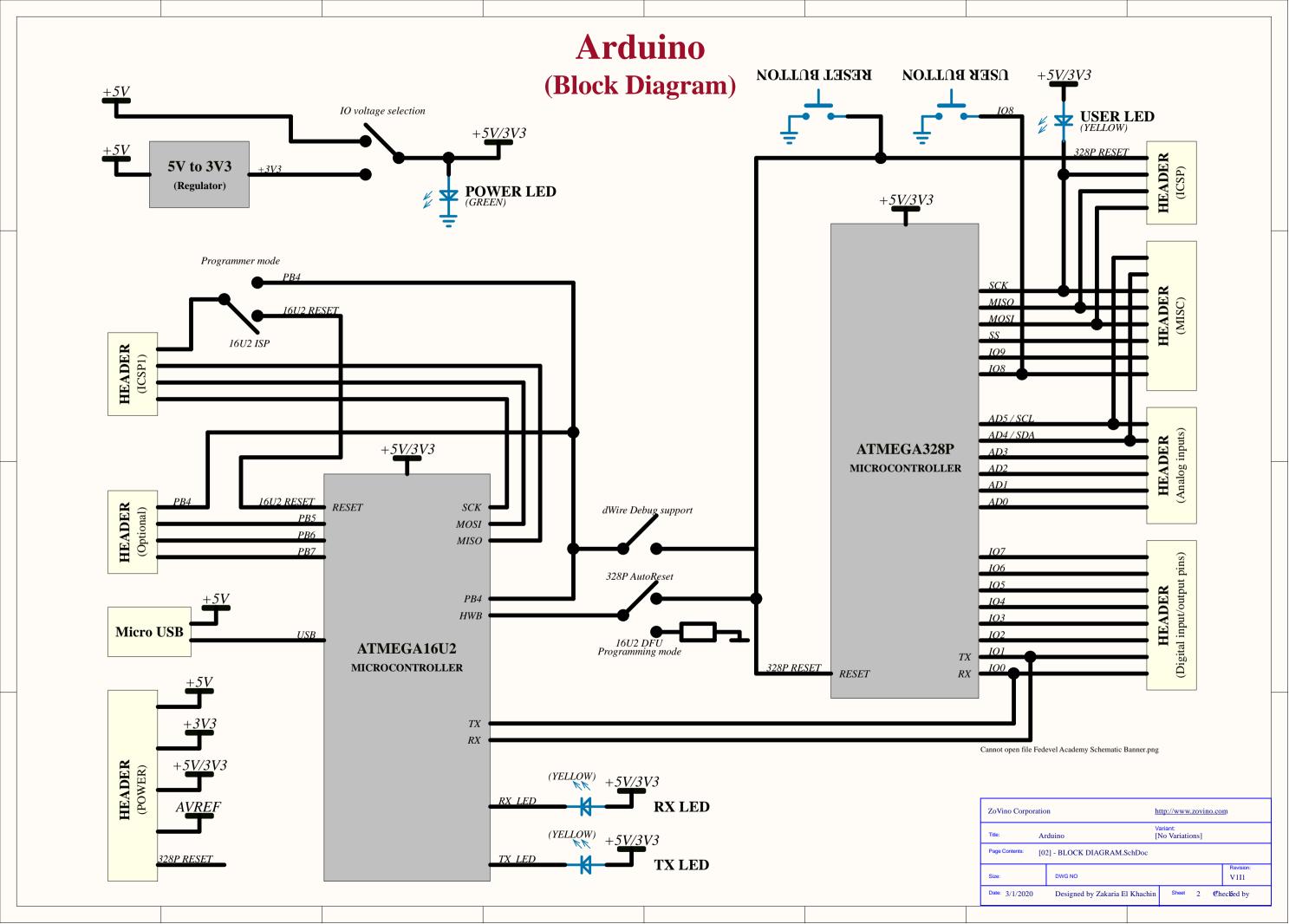RELEASED 02-FEB-2020

Cannot open file 28Pins3D.png

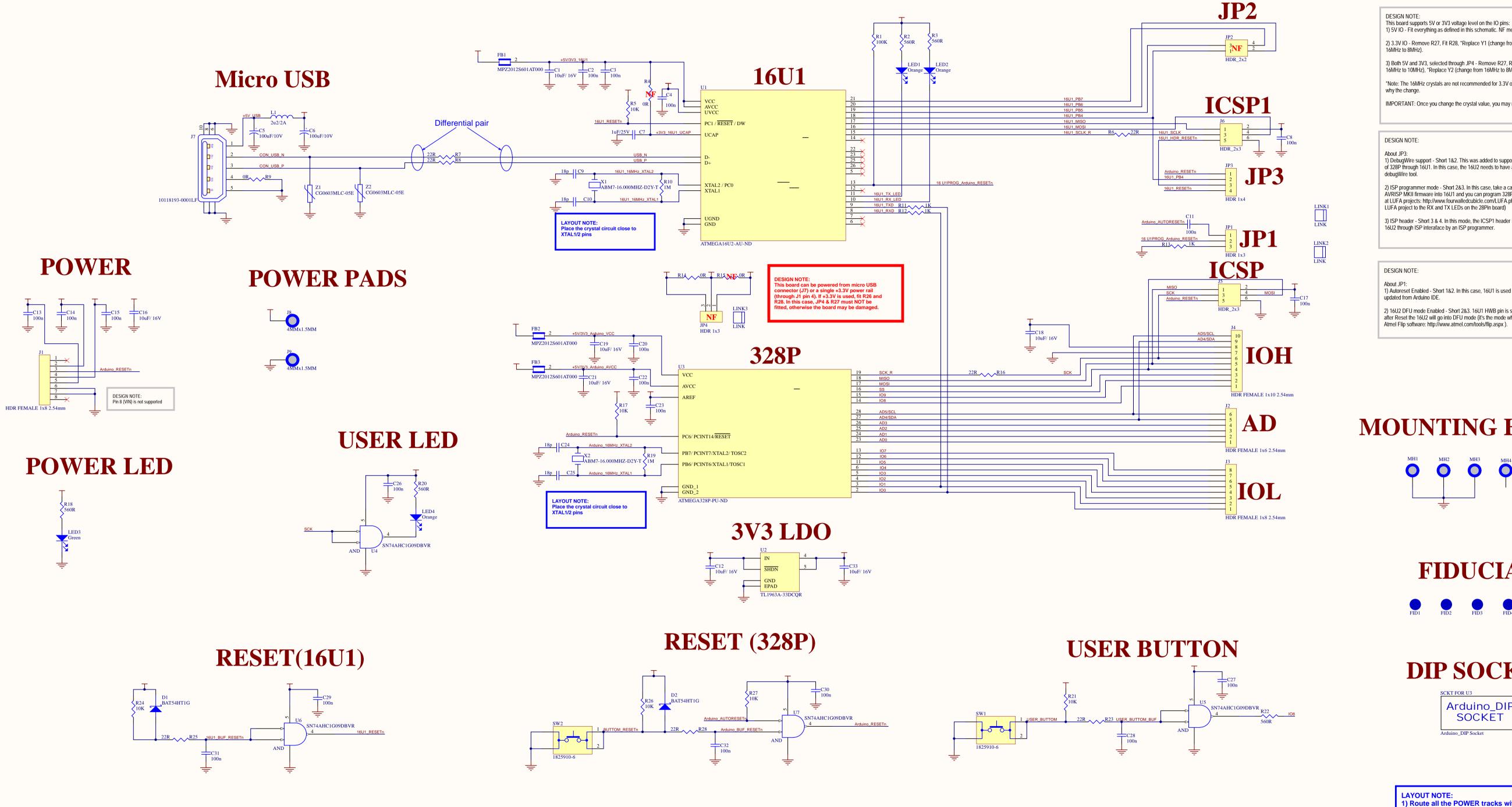DESIGN NOTE:
Example text for debug notes.

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.

ZoVino Corporation — http://www.zovino.com

Title:	Arduino	Variant: [No Variations]
Page Contents:	[01] - COVER PAGE.SchDoc	Reviewed by Robert Feranec
Size:	DWG NO	Revision: V1I1
Date: 3/1/2020	Designed by Zakaria El Khachin	Sheet 1 of 5

Arduino
(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

16U2 ISP

HEADER
(ICSP1)

SCK
MISO
MOSI
SS
IO9
IO8

HEADER
(MISC)

+5V/3V3

ATMEGA328P
MICROCONTROLLER

AD5 / SCL
AD4 / SDA
AD3
AD2
AD1
AD0

HEADER
(Analog inputs)

HEADER
(Optional)

PB4
PB5
PB6
PB7

16U2 RESET

+5V/3V3

RESET

SCK
MOSI
MISO

dWire Debug support

328P AutoReset

IO7
IO6
IO5
IO4
IO3
IO2
IO1
IO0

HEADER
(Digital input/output pins)

Micro USB

+5V

USB

PB4
HWB

16U2 DFU
Programming mode

ATMEGA16U2
MICROCONTROLLER

328P RESET

RESET

TX
RX

+5V

+3V3

+5V/3V3

AVREF

328P RESET

HEADER
(POWER)

TX
RX

(YELLOW)

+5V/3V3

RX_LED

RX LED

(YELLOW)

+5V/3V3

TX_LED

TX LED

Cannot open file Fedevel Academy Schematic Banner.png

ARDUINO SCHEMATIC

Micro USB

16U1

JP2

ICSP1

JP3

JP1

ICSP

POWER

POWER PADS

328P

IOH

AD

IOL

MOUNTING HOLES

POWER LED

USER LED

3V3 LDO

FIDUCIALS

RESET(16U1)

RESET (328P)

USER BUTTON

DIP SOCKET

Arduino_DIP SOCKET
Arduino_DIP Socket
SCKT FOR U3

Differential pair

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 16U1. In this case, the 16U2 needs to have a correct firmware and has to behave as a debugWire tool.
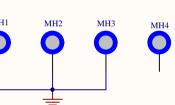
2) ISP programmer mode - Short 2&3. In this case, take a cable and connect J5 & J6 together. Upload AVRISP MKII firmware into 16U1 and you can program 328P. Example of AVRISP MKII firmware can be found at LUFA projects: http://www.fourwalledcubicle.com/LUFA.php (Tip: remap LEDs of the default AVRISP 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, 16U1 is used to reset 328P when firmware inside 328P is updated from Arduino IDE.

2) 16U2 DFU mode Enabled - Short 2&3. 16U1 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.

LAYOUT NOTE:
Place the crystal circuit close to XTAL1/2 pins

LAYOUT NOTE:
Place the crystal circuit close to XTAL1/2 pins
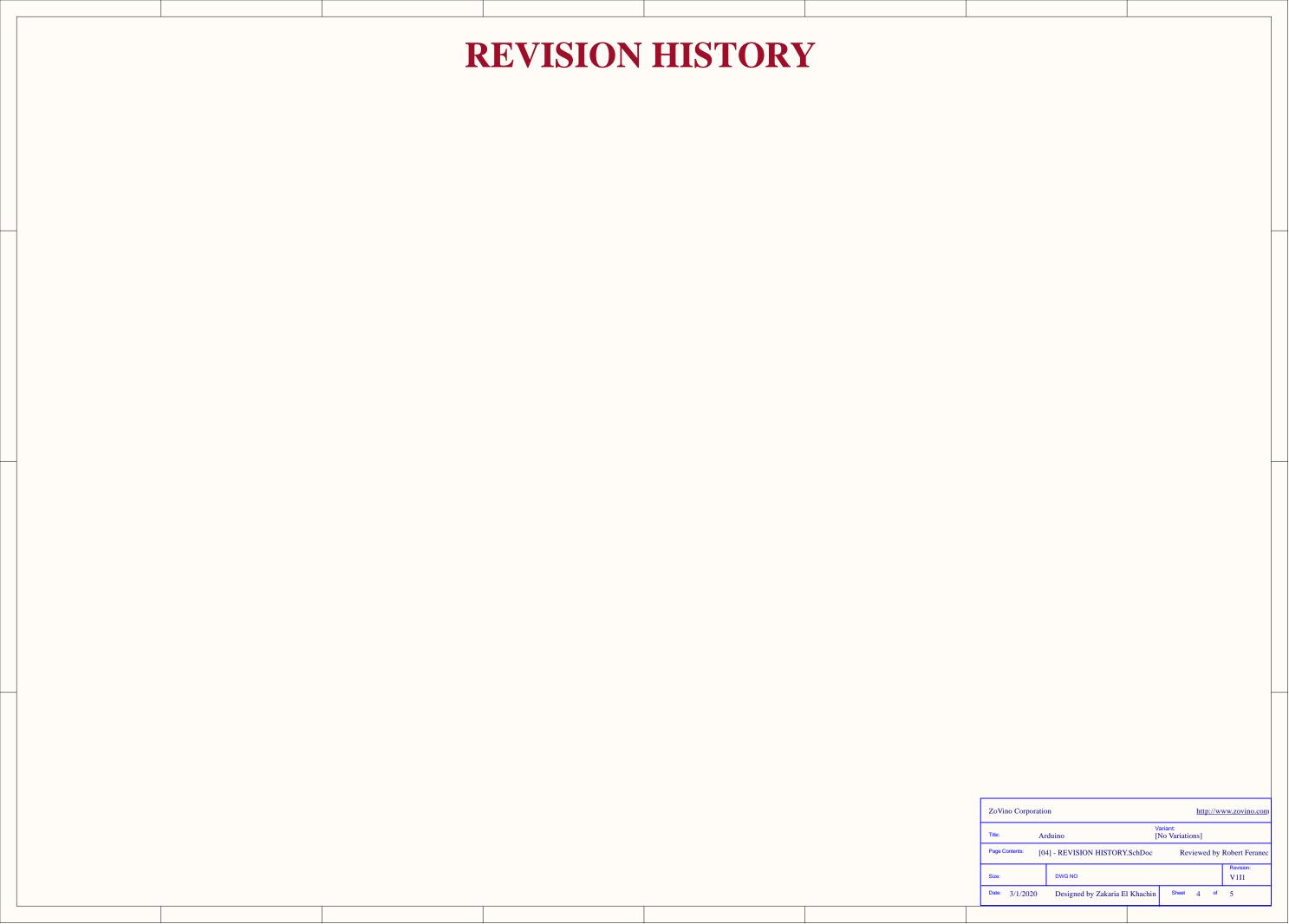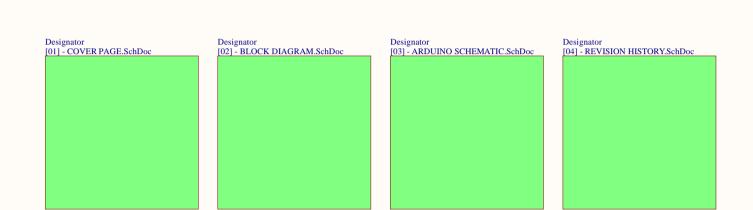
DESIGN NOTE:
Pin 8 (VIN) is not supported

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 (OnLayer('L1') or OnLayer ('L2'))" and then set 0.2mm width in PCB Inspector panel.

U1 ATMEGA16U2-AU-ND

U3 ATMEGA328P-PU-ND

U2 TL1963A-33DCQR

HDR FEMALE 1x8 2.54mm

HDR FEMALE 1x10 2.54mm

HDR FEMALE 1x6 2.54mm

HDR FEMALE 1x8 2.54mm

REVISION HISTORY

Designator
[01] - COVER PAGE.SchDoc

Designator
[02] - BLOCK DIAGRAM.SchDoc

Designator
[03] - ARDUINO 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.

Cannot open file Fedevel Academy Schematic Banner.png