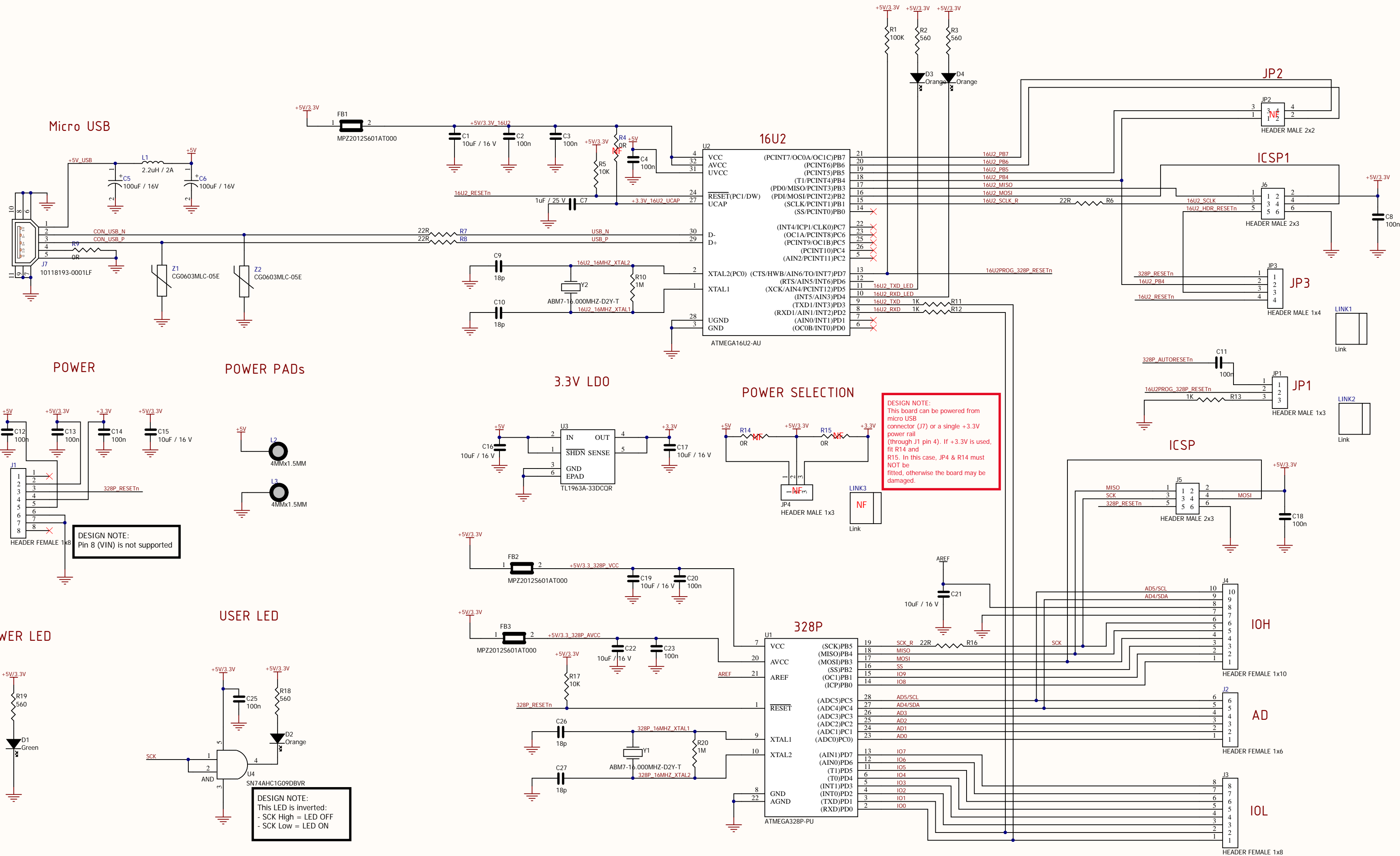


28PINS - SCHEMATIC



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, R27 F1, *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 J3:

- 1) DebugWire support - Short 1&2. This was added to support possible debugWire debugging (programming?) of 3299 through 16U2. In this case, the 16U2 needs to have a correct firmware and have 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 MKII1 firmware into 16U2 and you can program 329P. Example of AVRISP MKII1 firmware can be found at LUFA projects: <http://www.fourwalledcubicle.com/LUFA.php> (strip LEDS of the default AVRISP MKII1 LUFA project to the RX TX LEDs on the 29pin 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 (the mode when you can flash 16U2 firmware through USB and Atmel Flip software: <http://www.atmel.com/tools/flip.aspx>).

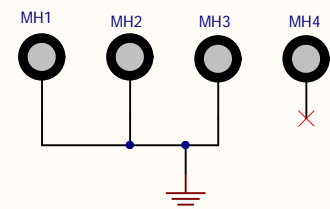


SCKT FOR U1

28-DIP SOCKET

28-DIP Socket

MOUNTING HOLES



BOARD MOUNTING HOLES Mounting holes 7.4mm pad 3.2mm drill MH1 MH2 MH3 ONE IN EACH CORNER

FIDUCIALS



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 INet("+") and not INet("GND")) and IsTrack and (OnLayer('L1') or OnLayer('L2'))" and then set 0.2mm width in PCB Inspector panel.

| | | | |
|-----------------|---------------------------------|----------------|------------|
| None | | | |
| Title: | 28Pins Variant: [No Variations] | | |
| Page Contents: | 28Pins_Schematic_VIII.SchDoc | | |
| Size: | DWG NO | Revision: VIII | |
| Date: 3/31/2020 | Designed By Abdulmaguid Eissa | Sheet 1 | Checked by |