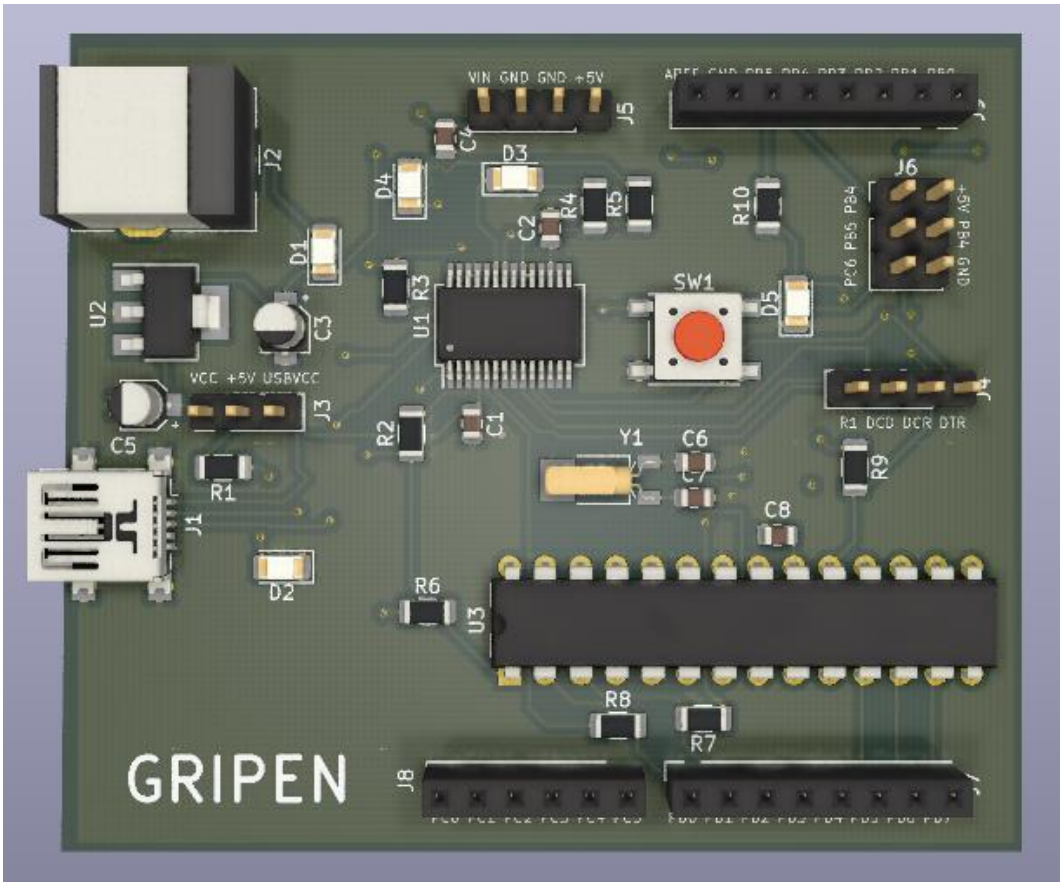
PCB Report

Abstract

This document contains a description of a PCB design. The report should clarify why the circuit board looks the way it does and the thoughts behind the different design choices. There are also pictures of various drawings and 3D visualization of the circuit board.



3D visualization of a pcb-design(Arduino clone), Pcbnew, KiCad, *Daniel Gripenstedt*

Overview

The circuit board is an Arduino clone (Arduino Nuova Generazione (NG) v4.0) which has an 8-bit MCU (ATmega8-16PU [1] and an FT232RL (USB-UART interface) [2]. Program code is sent over with a USB mini-B cable, which can also act as a power supply. There are several LEDs and a reset button. A crystal of 16MHz is placed on the card. All drawings are created with KiCad [3].

Components

Most components are of type SMD as these are cheaper than THT. Contact pins and power supply are of type THT. In total, there are 37 components where the smallest are 0805. Smaller components were not considered necessary as there was sufficient space on the card. All components are located on the top of the card.

Microcontroller

The microcontroller used is an ATmega8-16PU. This MCU has 8Kbytes of On-Chip In-System Reprogrammable Flash memory. Program code is sent over with a USB mini-B cable using the MCU's Boot Loader-Support [1].

Power and power supply

The MCU requires 4.5V – 5.5V voltage [1]. Two polarized capacitors with 100uF have been placed by the cards power supply as they may have a higher capacitance at a lower price than those that are not polarized.

Voltage can be supplied directly with 5V from USB mini-B or via a voltage divider and the card's "power jack" but it is also possible to connect an external power source via the VIN stick connected before the voltage divider. Several decoupling capacitors have been used to protect the circuits from current spikes.

Power options:

1. USB mini-B cable connected to, for example, a computer.
2. Power Jack with, for example, an AC to DC converter.
3. VIN stick with, for example, a 9V battery.

PCB Layout

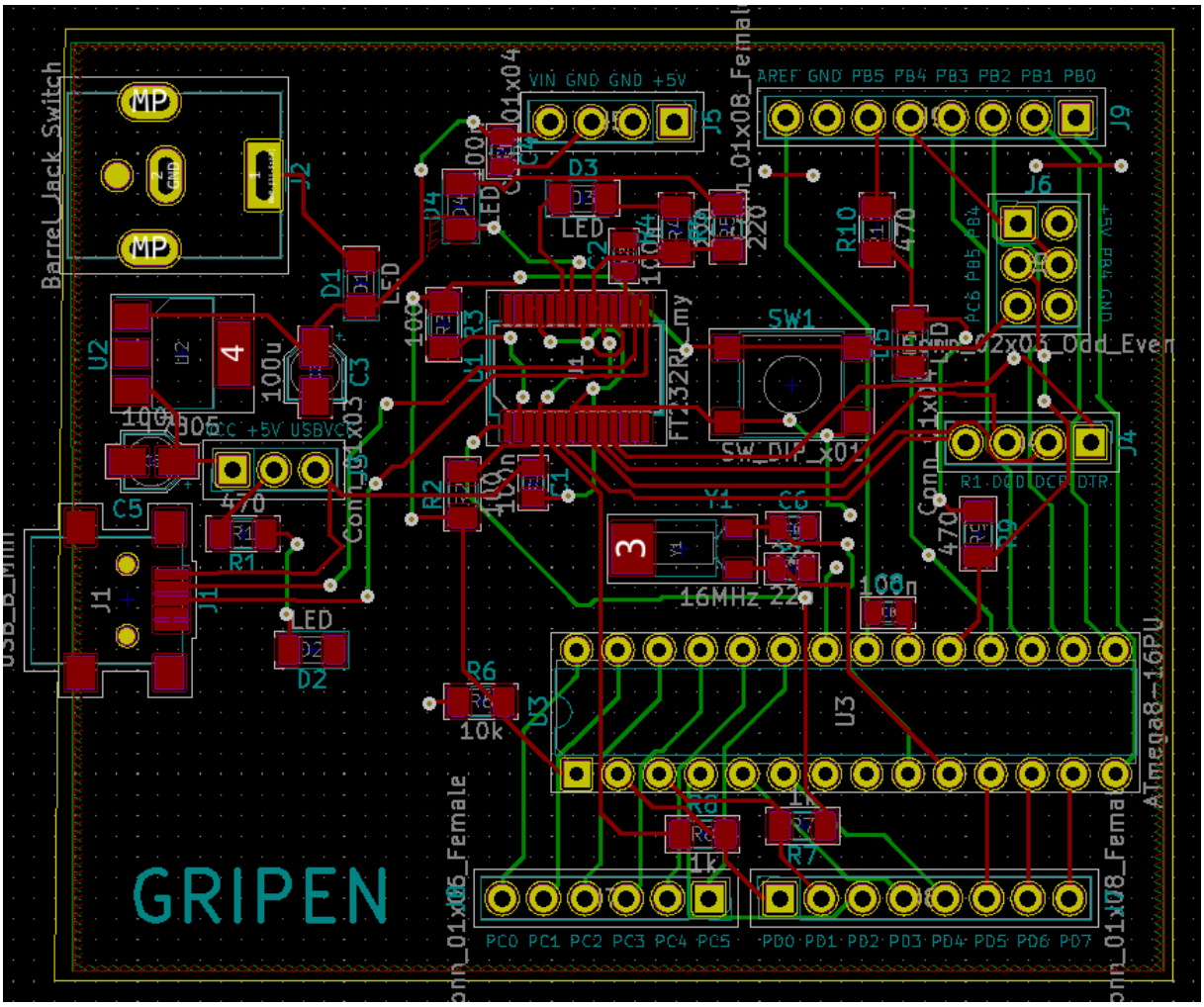
This design is of a two-layer card with a ground plane and a voltage plane. The crystal Y1 is located near the MCU and the loading capacitance have been placed close to the crystal. This is good because it is a sensitive circuit. The load capacitances are positioned so that their fields are counterproductive and the signals from the crystal travels close to each other so that the magnetic fields cancel each other out. No other wires have been pulled parallel or above which may interfere. The ground signal from the crystal to the MCU is short and close to the other two signals, which gives

a small loop area. The ground plane got disconnected by the crystal but was solved by connecting a bridge between the connected and disconnected ground planes.

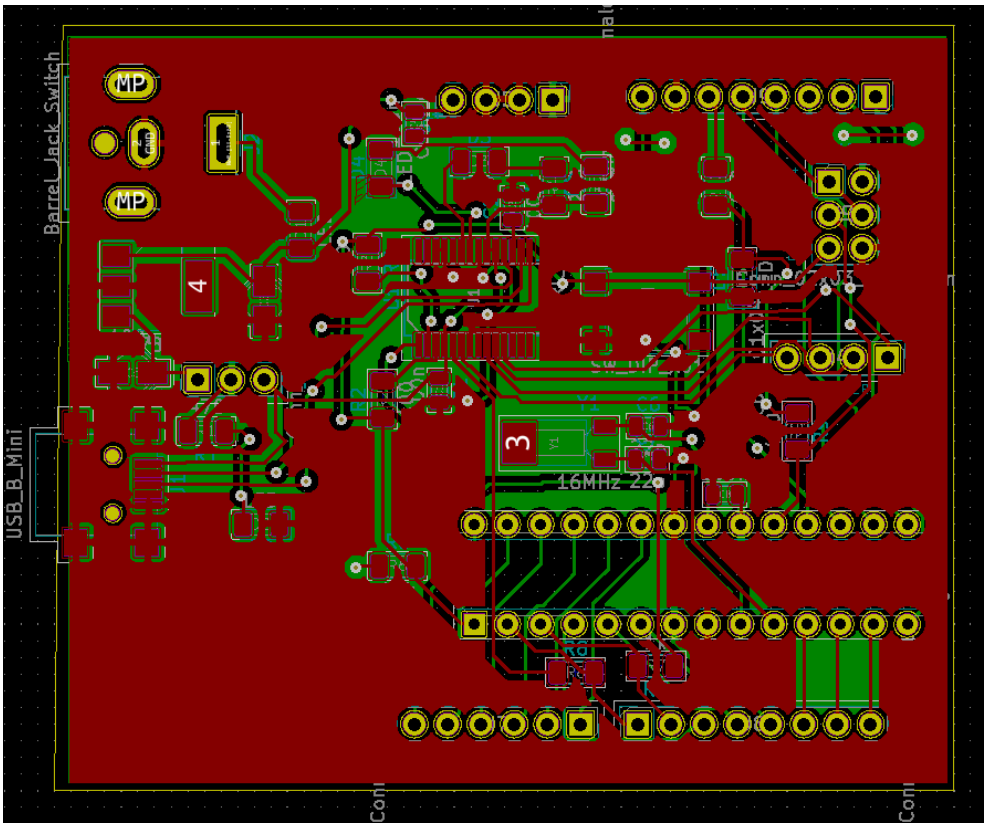
Decoupling capacitors are located near the circuits they decouple to minimize impedance and maximize decoupling efficiency.

Wires and ground/voltage planes are located a bit in from the edge of the card to prevent interference (the card may otherwise act as a patch antenna).

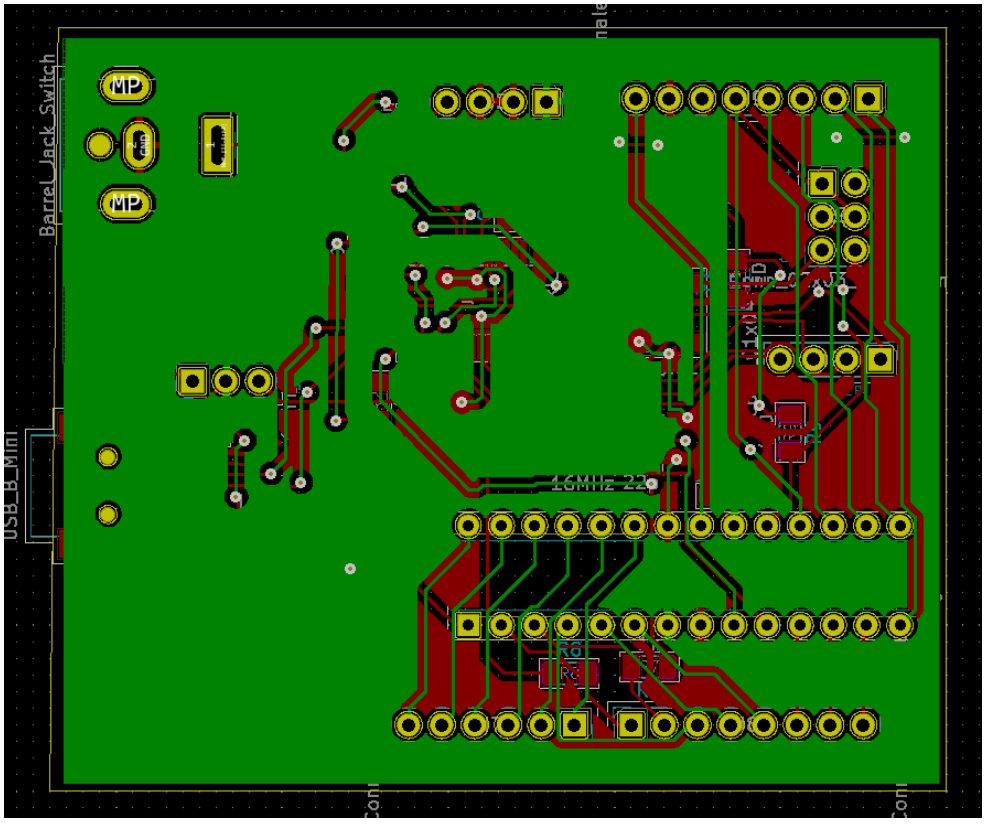
The circuit is partitioned so that power/voltage supply is placed on one part of the card, the MCU with resistors, crystal and capacitors on another and the FT232RL with resistors, diodes and capacitors on another part of the card.



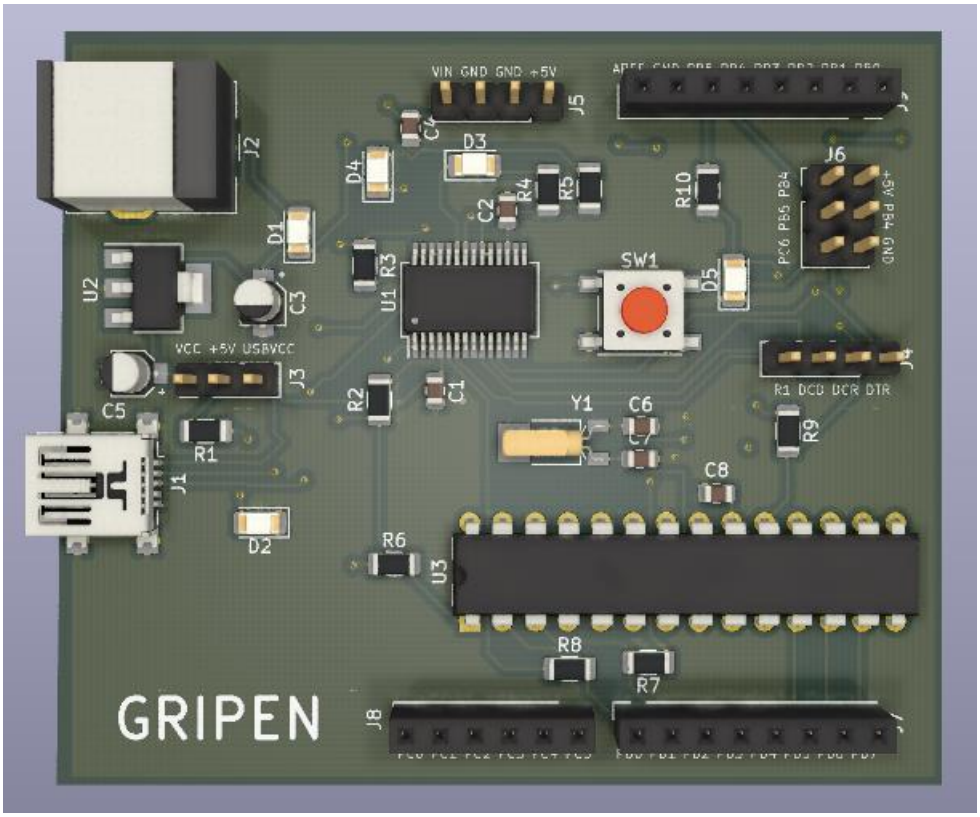
Front of the circuitboard, Pcbnew, KiCad, *Daniel Gripenstedt*



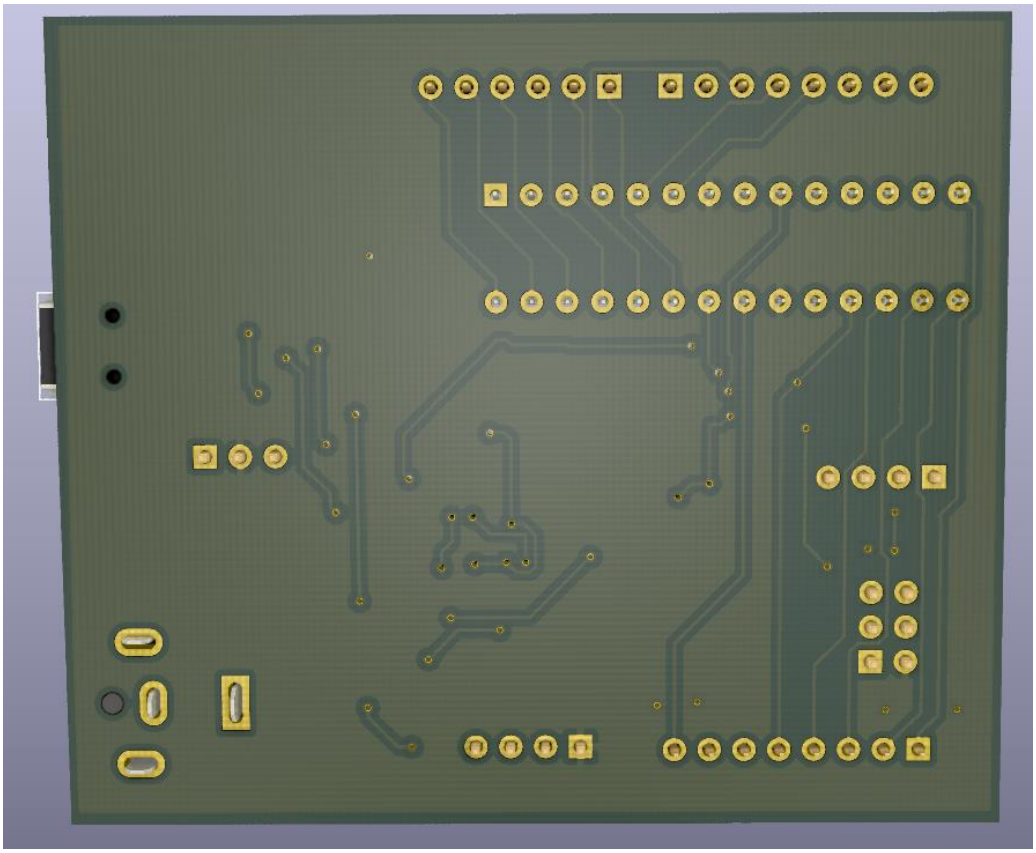
Front of the circuit board, the red surface symbolizes the earth plane and the green surface symbolizes the voltage plane, Pcbnew, KiCad, *Daniel Gripenstedt*



Back of the circuit board, the green surface symbolizes the voltage plane and the red surface symbolizes the Earth plane, Pcbnew, KiCad, *Daniel Gripenstedt*



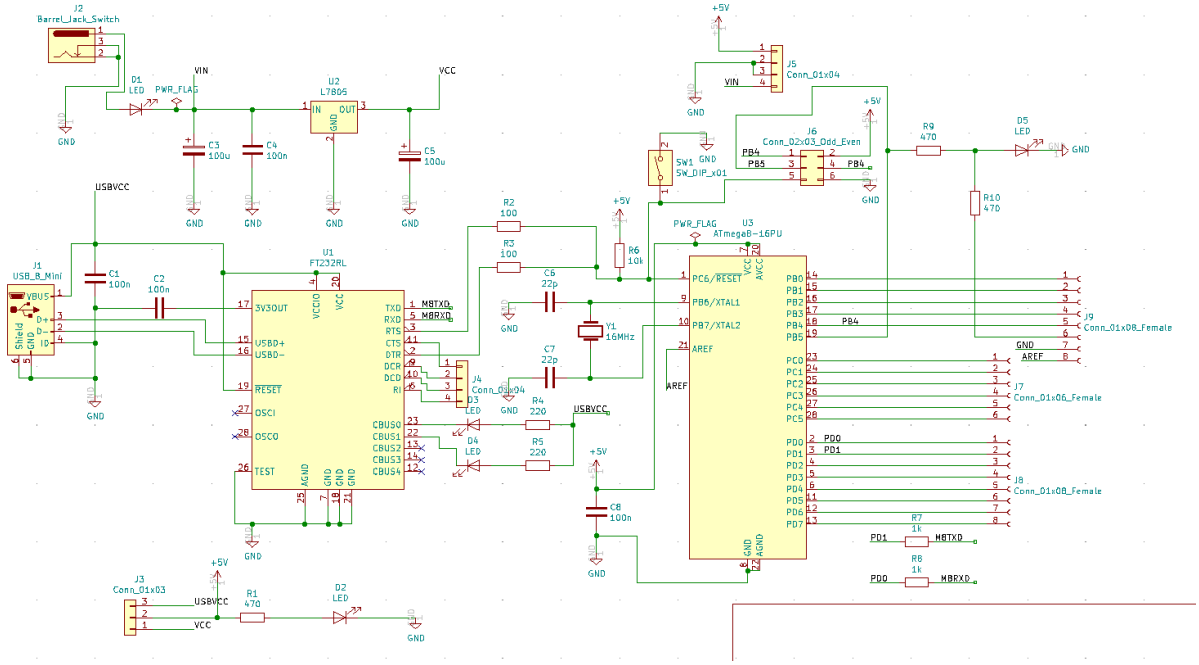
Front of the 3D visualized circuit board, Pcbnew, KiCad, *Daniel Gripenstedt*



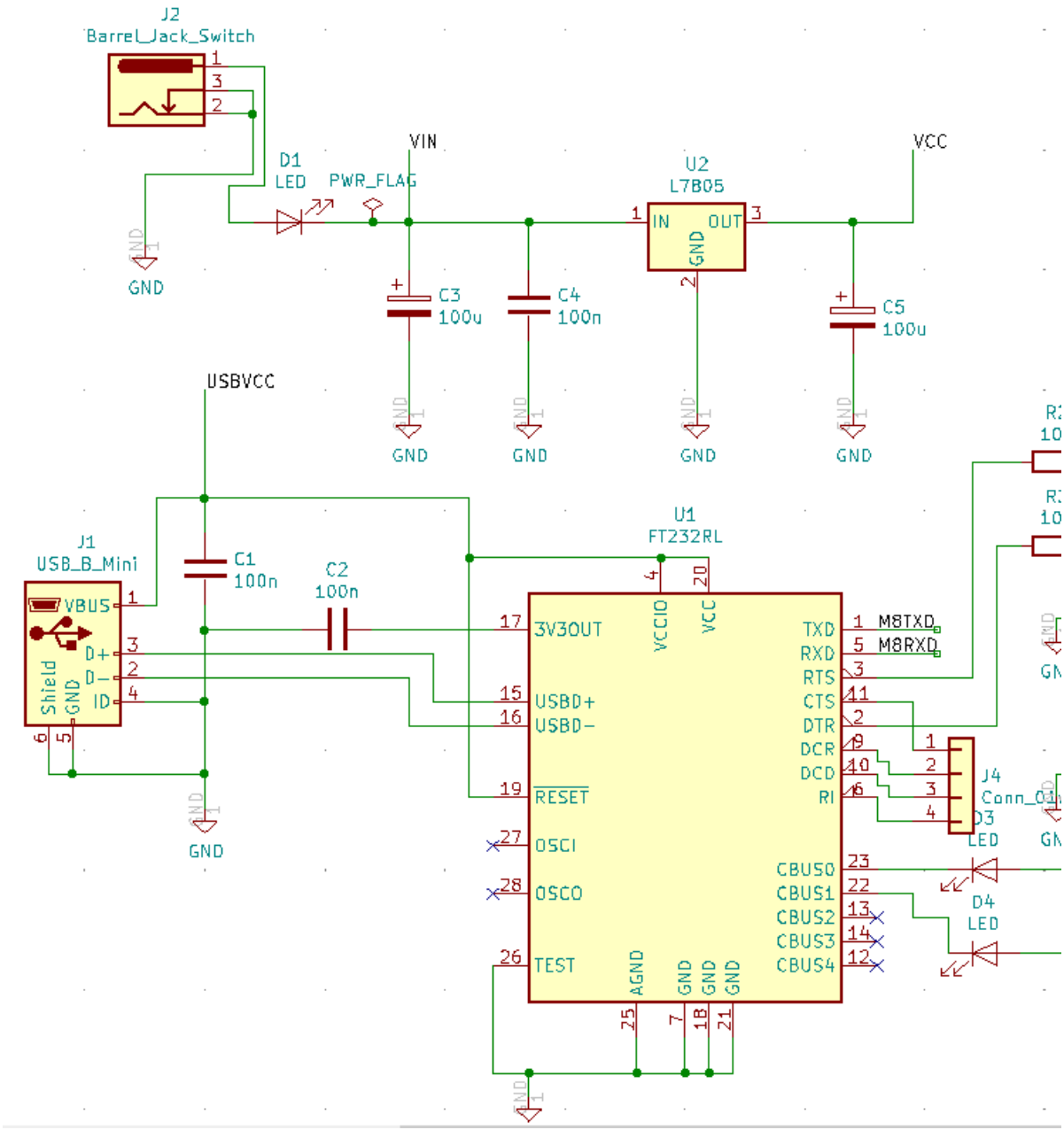
Back of the 3D visualized circuit board, Pcbnew, KiCad, *Daniel Gripenstedt*

Drawing

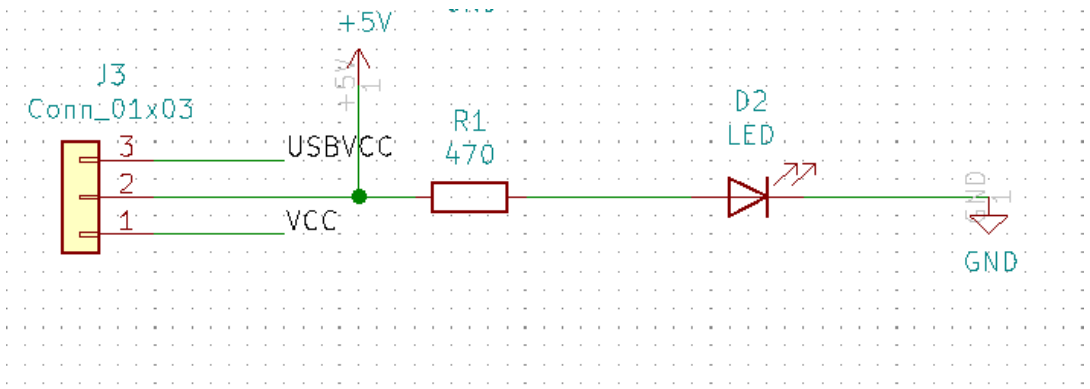
The first image is on the entire drawing, the three lower slides are parts of the drawing but on a larger scale.



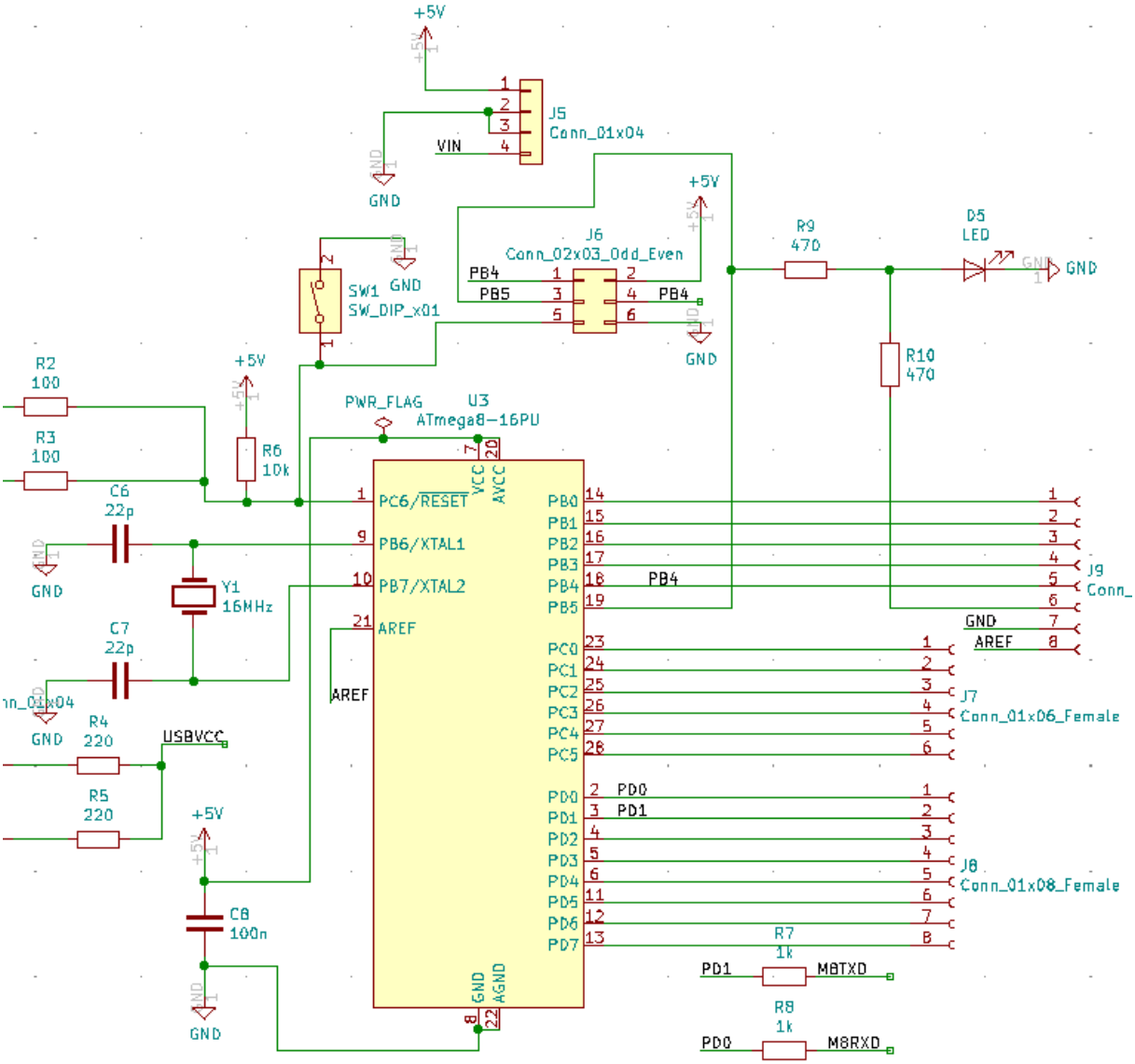
Drawing in Easchema, KiCad, *Daniel Gripenstedt*



Drawing in Easchema, KiCad, *Daniel Gripenstedt*



Drawing in Easchema, KiCad, Daniel *Gripenstedt*



Drawing in Easchema, KiCad, *Daniel Gripenstedt*

Bom List

Designator	Package	Quantity	Designation
J1	USB_Mini-B_Lumberg_2486_01_Horizontal	1	USB_B_Mini
J8,J9	PinSocket_1x08_P2.54mm_Vertical	2	Conn_01x08_Female
C3,C5	CP_Elec_3x5.3	2	100u
D1,D2,D3,D4,D5	LED_1206_3216Metric	5	LED
J2	BarrelJack_CUI_PJ-063AH_Horizontal	1	Barrel_Jack_Switch
J6	PinHeader_2x03_P2.54mm_Vertical	1	Conn_02x03_Odd_Even
J7	PinSocket_1x06_P2.54mm_Vertical	1	Conn_01x06_Female
R1,R9,R10	R_1206_3216Metric	3	470
R2,R3	R_1206_3216Metric	2	100
R4,R5	R_1206_3216Metric	2	220
R6	R_1206_3216Metric	1	10k
R7,R8	R_1206_3216Metric	2	1k
SW1	SW_SPST_PTS645	1	SW_DIP_x01
U1	SSOP-28_5.3x10.2mm_P0.65mm	1	FT232RL_my
U2	SOT-223	1	L7805
U3	DIP-28_W7.62mm	1	ATmega8-16PU
Y1	Crystal_SMD_MicroCrystal_MS1V-T1K	1	16MHz
C1,C2,C4,C8	C_0805_2012Metric	4	100n
C6,C7	C_0805_2012Metric	2	22p
J3	PinHeader_1x03_P2.54mm_Vertical	1	Conn_01x03
J4,J5	PinHeader_1x04_P2.54mm_Vertical	2	Conn_01x04

BOM list generated by Pcbnew, KiCad, *Daniel Gripenstedt*

References

[1] (2020), ATmega8/L datasheet - ATMEGA8-16PU-Microchip datasheet-14702621.pdf, Available: https://datasheet.octopart.com/ATMEGA8-16PU-Microchip-datasheet-14702621.pdf

[2] (2020), FT232R - DS_FT232R.pdf, Available: https://www.ftdichip.com/Support/Documents/DataSheets/ICs/DS_FT232R.pdf

[3] (2020), KiCad EDA, A Cross Platform and Open Source Electronics Design Automation Suite, Available: https://kicad-pcb.org/