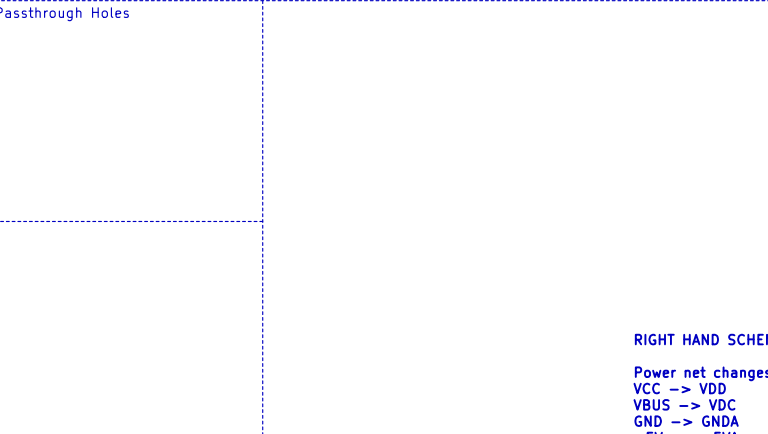
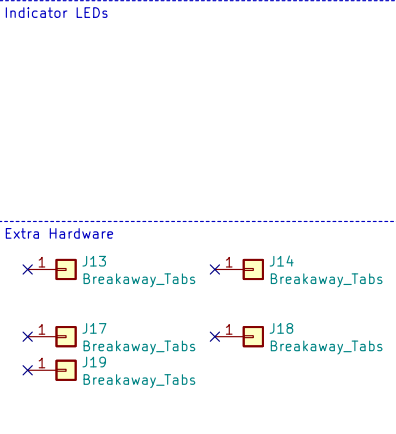
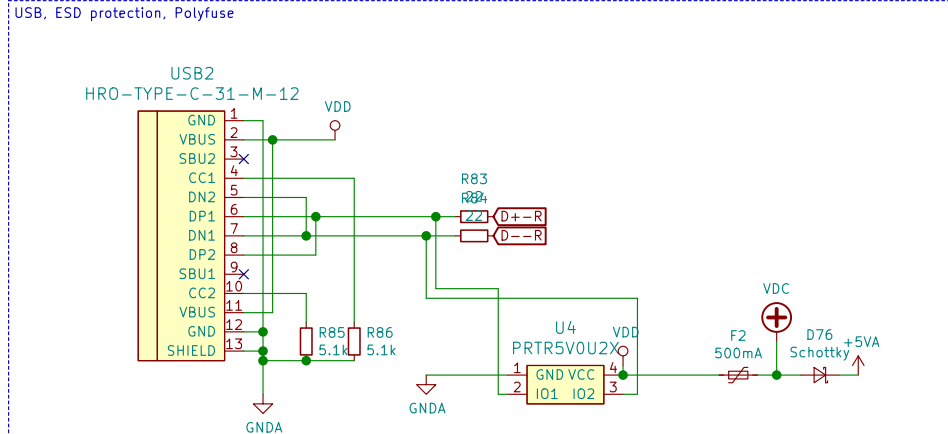
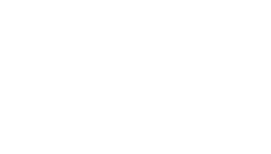
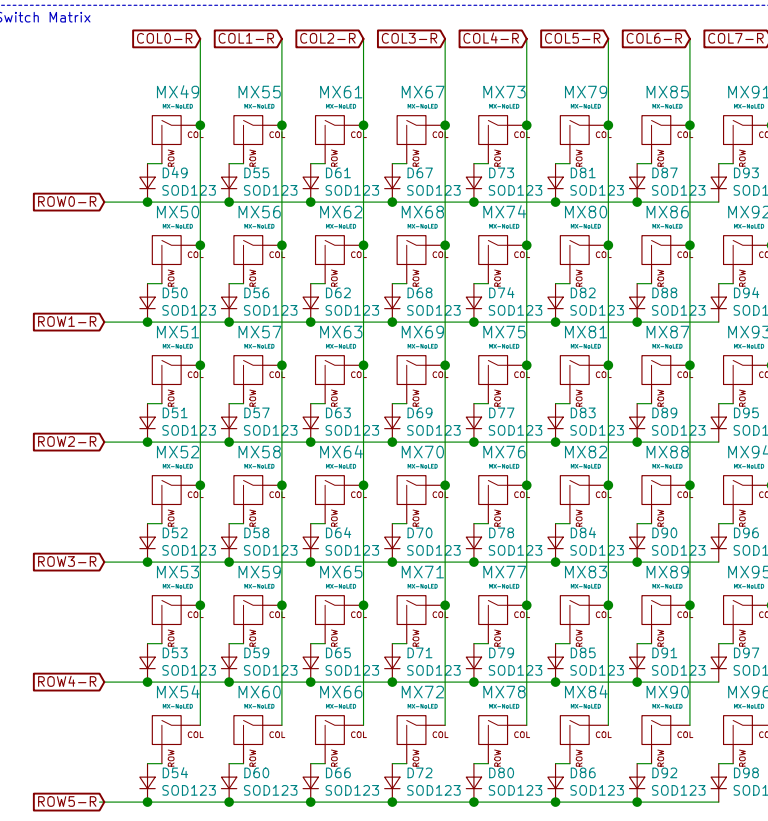
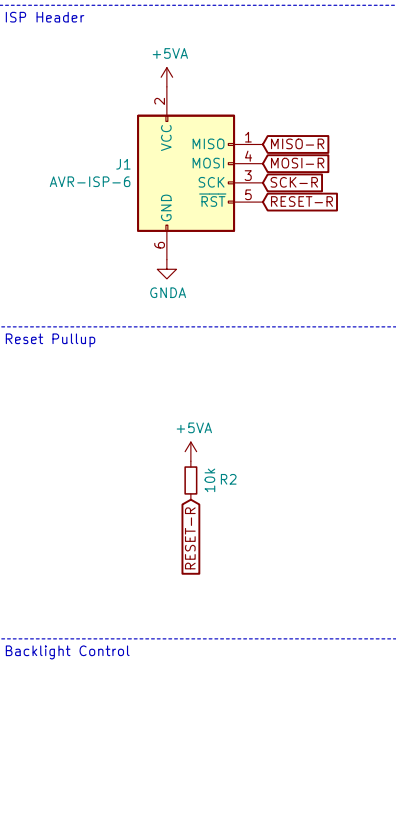
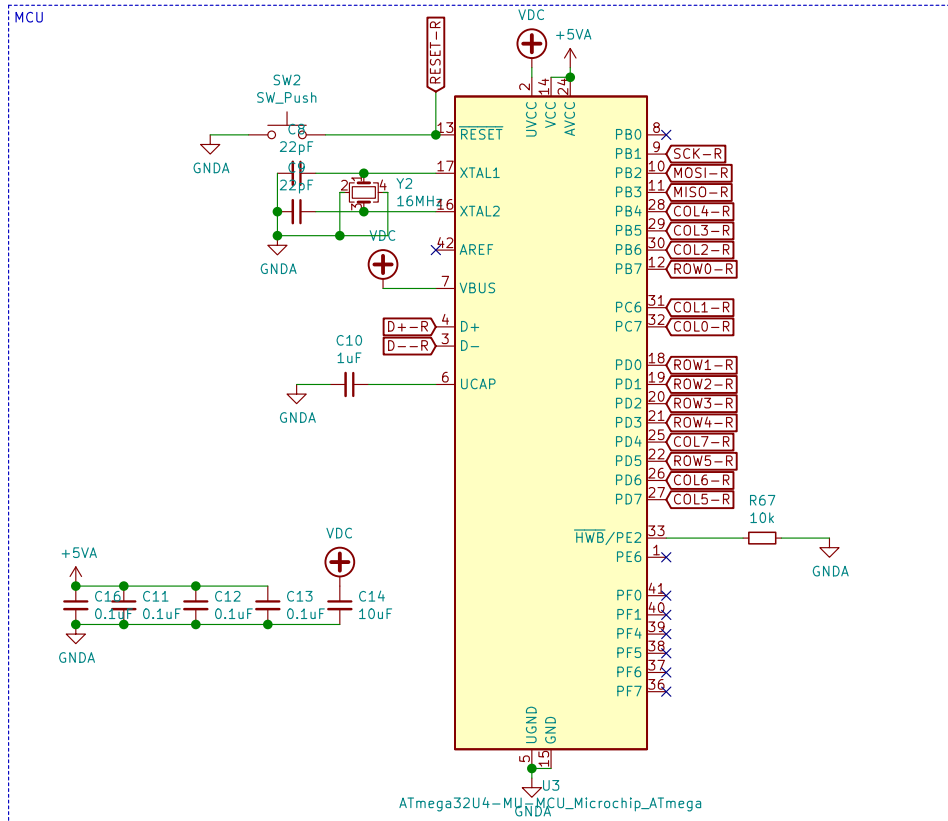


LEFT HAND SCHEMATIC



RIGHT HAND SCHEMATIC

Power net changes:
VCC -> VDD
VBUS -> VDC
GND -> GND
+5V -> +5VA
to prevent kicad from complaining about no connections between halves