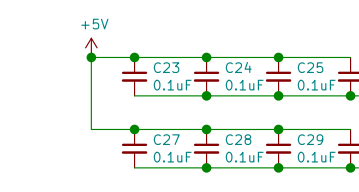
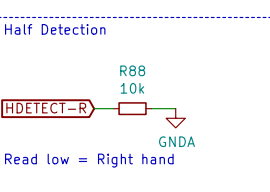
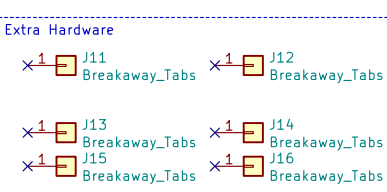
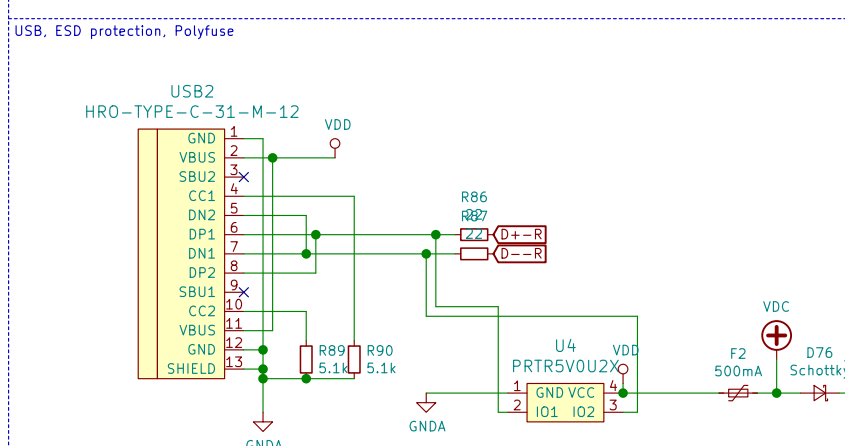
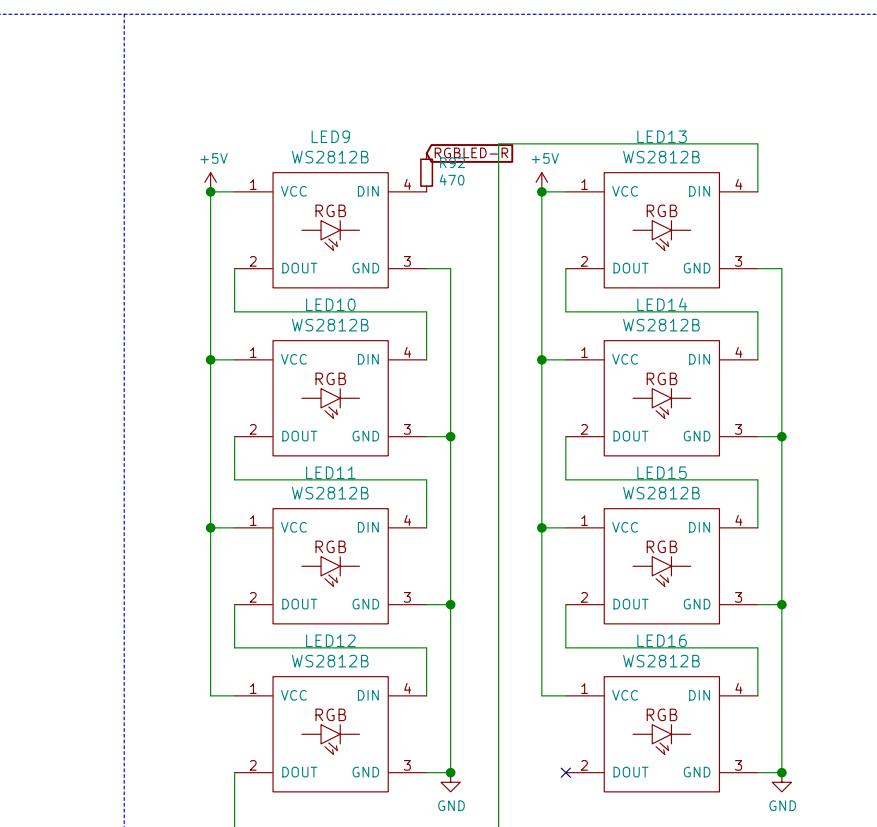
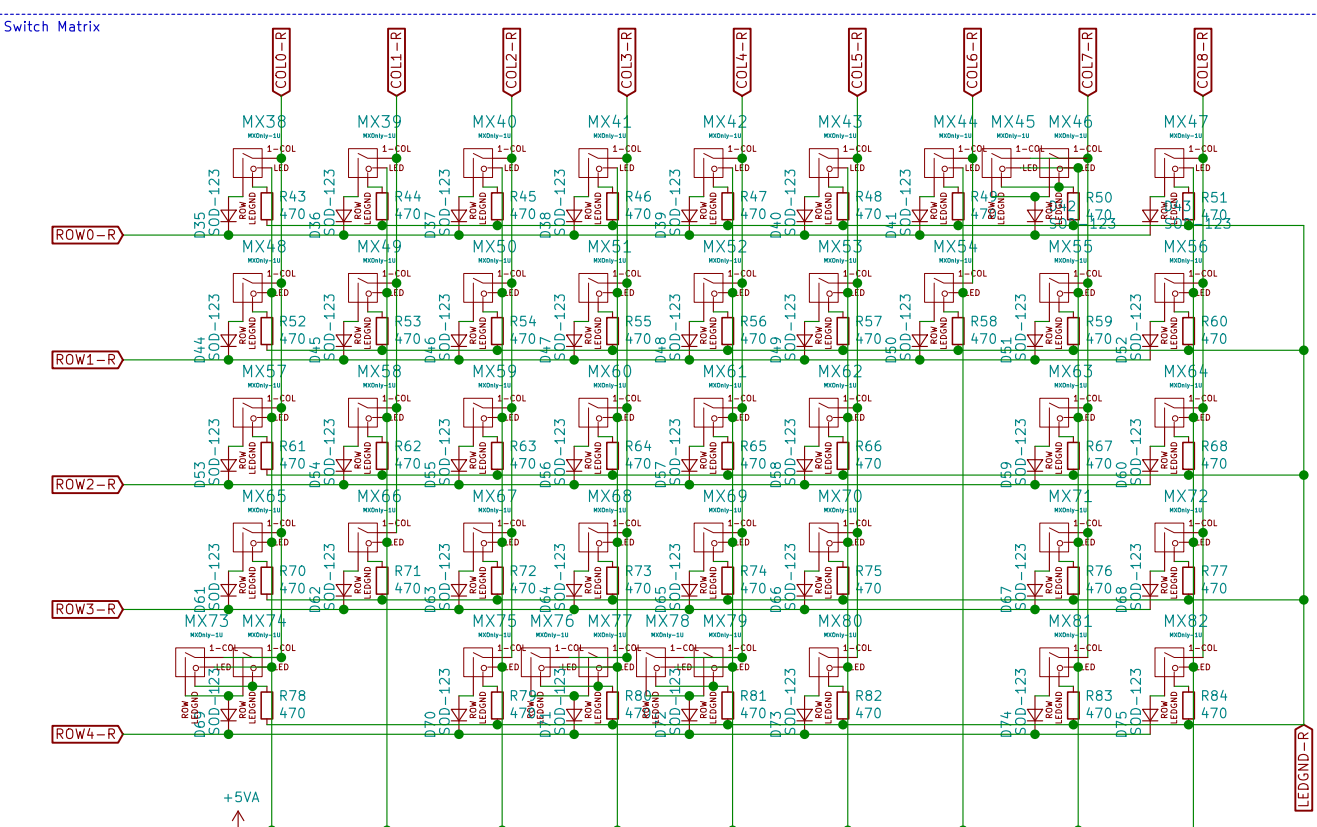
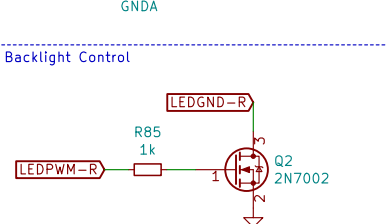
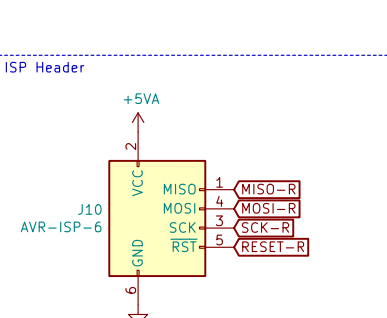
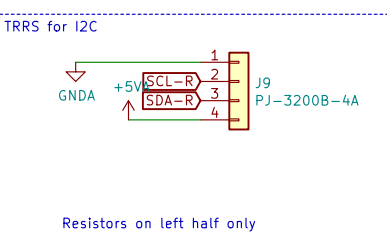
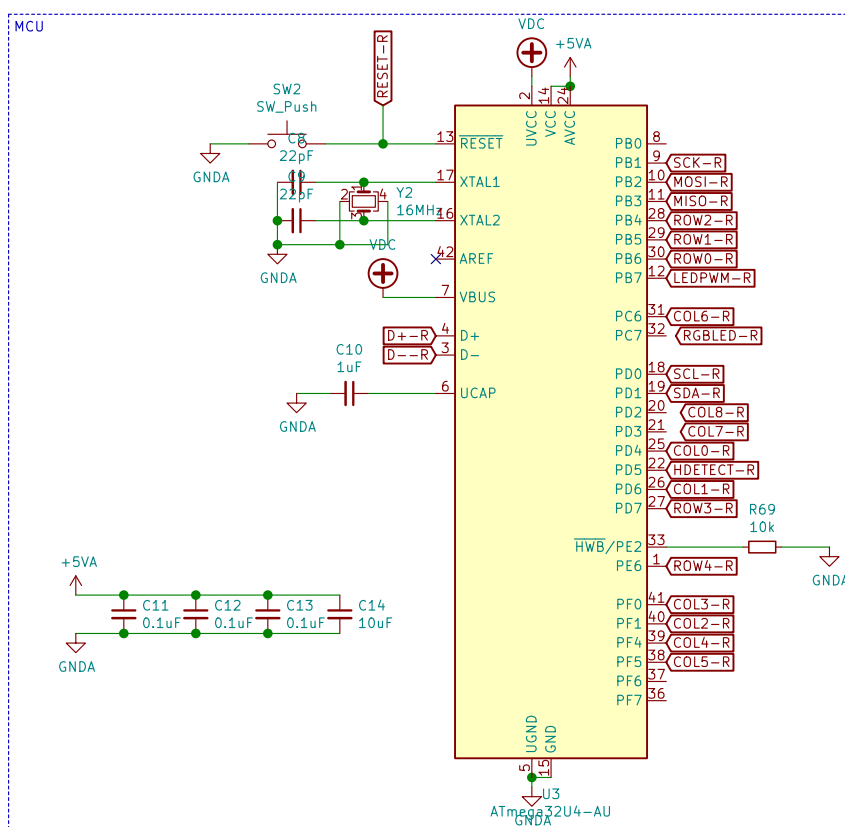


LEFT HAND SCHEMATIC



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