

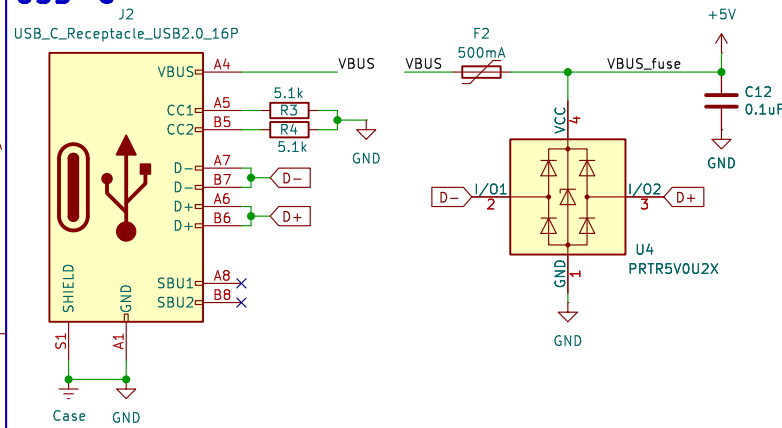
	1	2	3	4	5	6
A						A
B						B
C						C
D						D
	1	2	3	4	5	6

left

File: left.kicad_sch

Sheet: /		
File: freaky75_v1.kicad_sch		
Title:		
Size: A4	Date:	Rev:
KiCad E.D.A. 9.0.0		Id: 1/2

J2
USB_C_Receptacle_USB2.0_16P



Default pins for nice view

- CS = D1 / P0.06
- SCL = D2 / 0.20
- MOSI = D3 / P0.17

+3V3

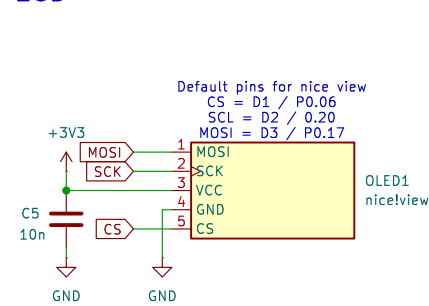
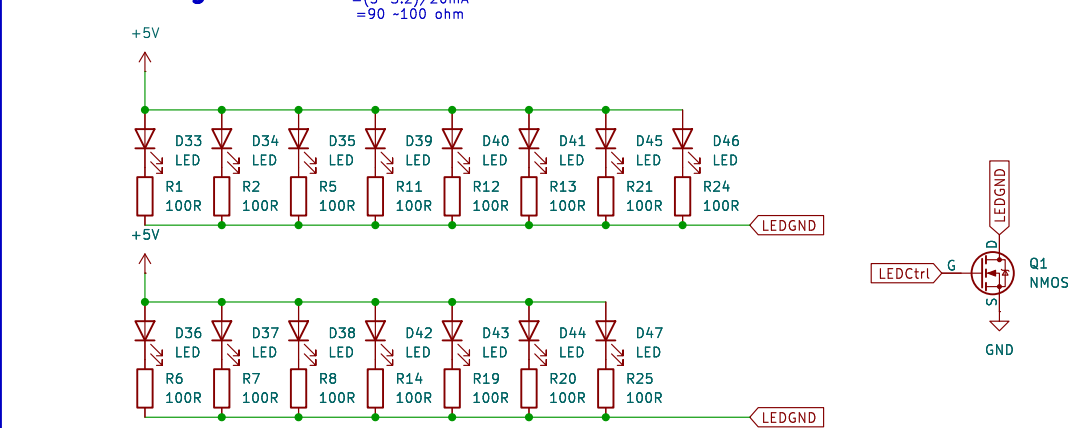
MOSI

SCK

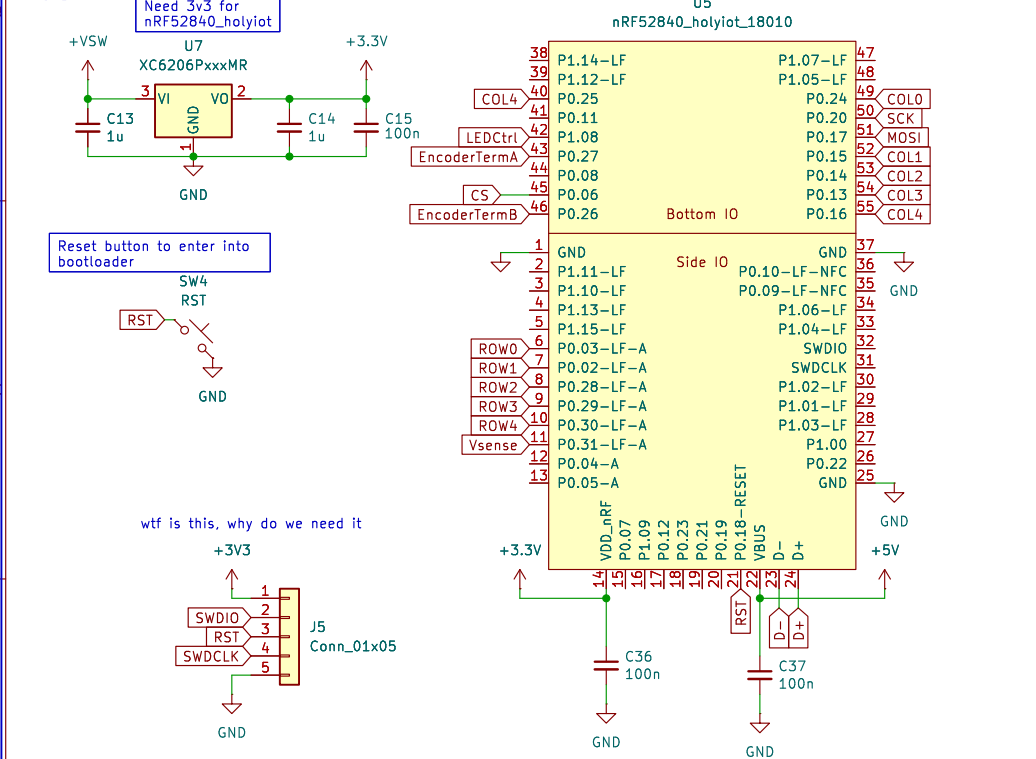
VCC

GND

OLED1
niceview


$$= (5 \cdot 3.2) / 20 \text{ mA}$$
$$= 90 \sim 100 \text{ ohm}$$


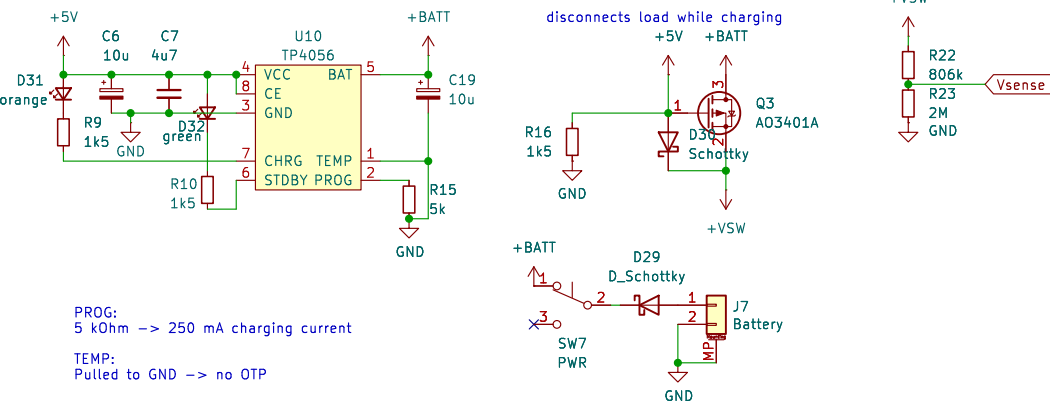
Need 3v3 for
pRF52840, holviot



The diagram illustrates a 6x6 grid of switches, labeled ROW0 to ROW4 and COL0 to COL5. Each switch is represented by a green line with a red switch symbol and a label like 'S4', 'D4', 'D_Small', and 'choc_v1_SW_solder'. A legend in the bottom right corner shows a yellow box with a red switch symbol and labels 'S2' and 'S1', with the text 'Rotary' and 'Extra switch footprint'.

	Col0	Col1	Col2	Col3	Col4	Col5
Row0	Esc	1	2	3	4	5
Row1	Tab	Q	W	E	R	T
Row2	Shift	A	S	D	F	G
Row3	Ctrl	Z	X	C	V	B
Row4			Win	FN	Spc	Enc

	Col5	Col4	Col3	Col2	Col1	Col0
Row0	6	7	8	9	0	Bck
Row1	Y	U	I	O	P	\
Row2	H	J	K	L	:	"
Row3	N	M	<	>	?	Ctrl
Row4	Enc	Ent	FN	Alt		



H1 MountingHole H2 MountingHole

H3 MountingHole H4 MountingHole H5 MountingHole

Sheet: /left/
File: left.kicad_sch

Title:

Size: A3

Date:

Rev:

KiCad E.D.A. 9.0.0