MCU sheet

File: mcu_sheet.kicad_sch

Batt

File: batt.kicad_sch

power

File: power.kicad_sch

Sensor Sheet

File: sensors.kicad_sch

drivers

File: drivers.kicad_sch

usb_pd

File: usb_pd.kicad_sch

Sheet: /
File: Catlin RP2040 Sensing Controller.kicad_sch

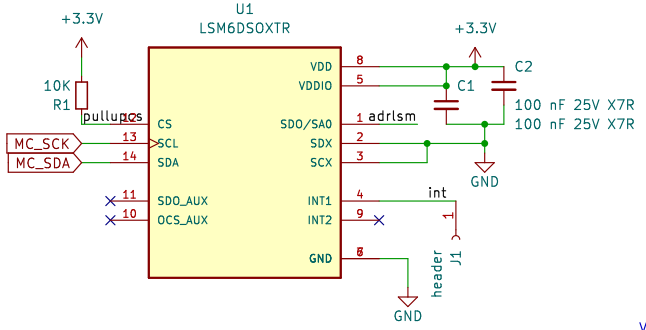
Title:

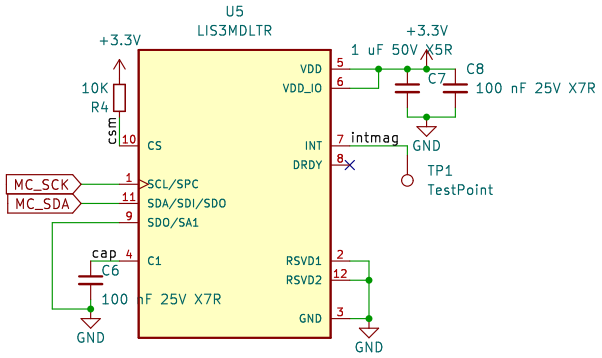
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 7.0.11		Id: 1/7

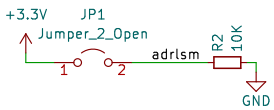
NB: make sure an X, Y, Z axis is printed on silkscreen

Dots on BHI and BMM should be on same relative corners in layout to orient

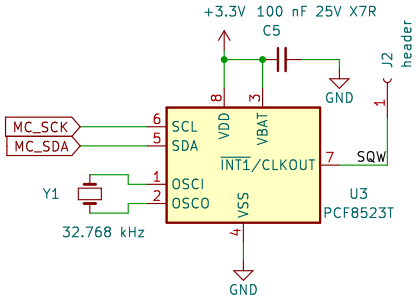
LSM should be dot top right and LIS should be dot bottom right

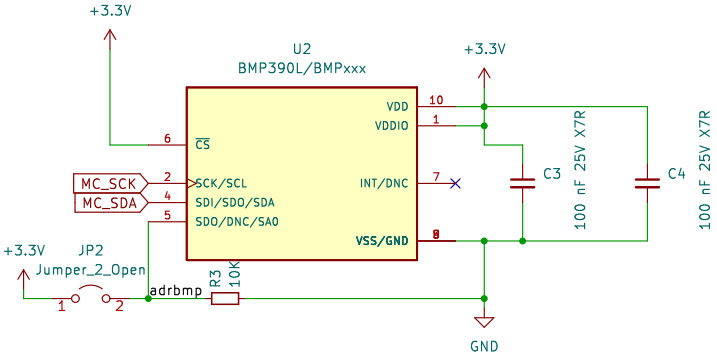


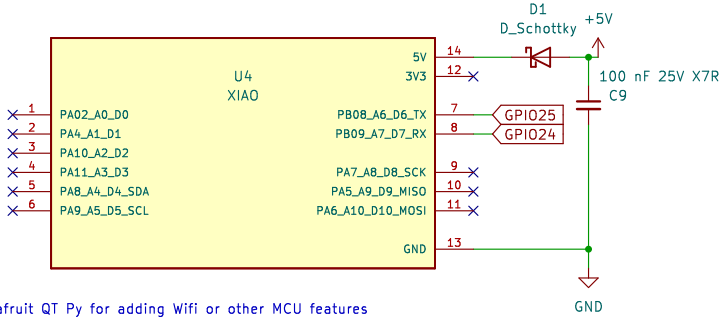




VBAT not used since entire system is connected to battery







Xiao or Adafruit QT Py for adding Wifi or other MCU features

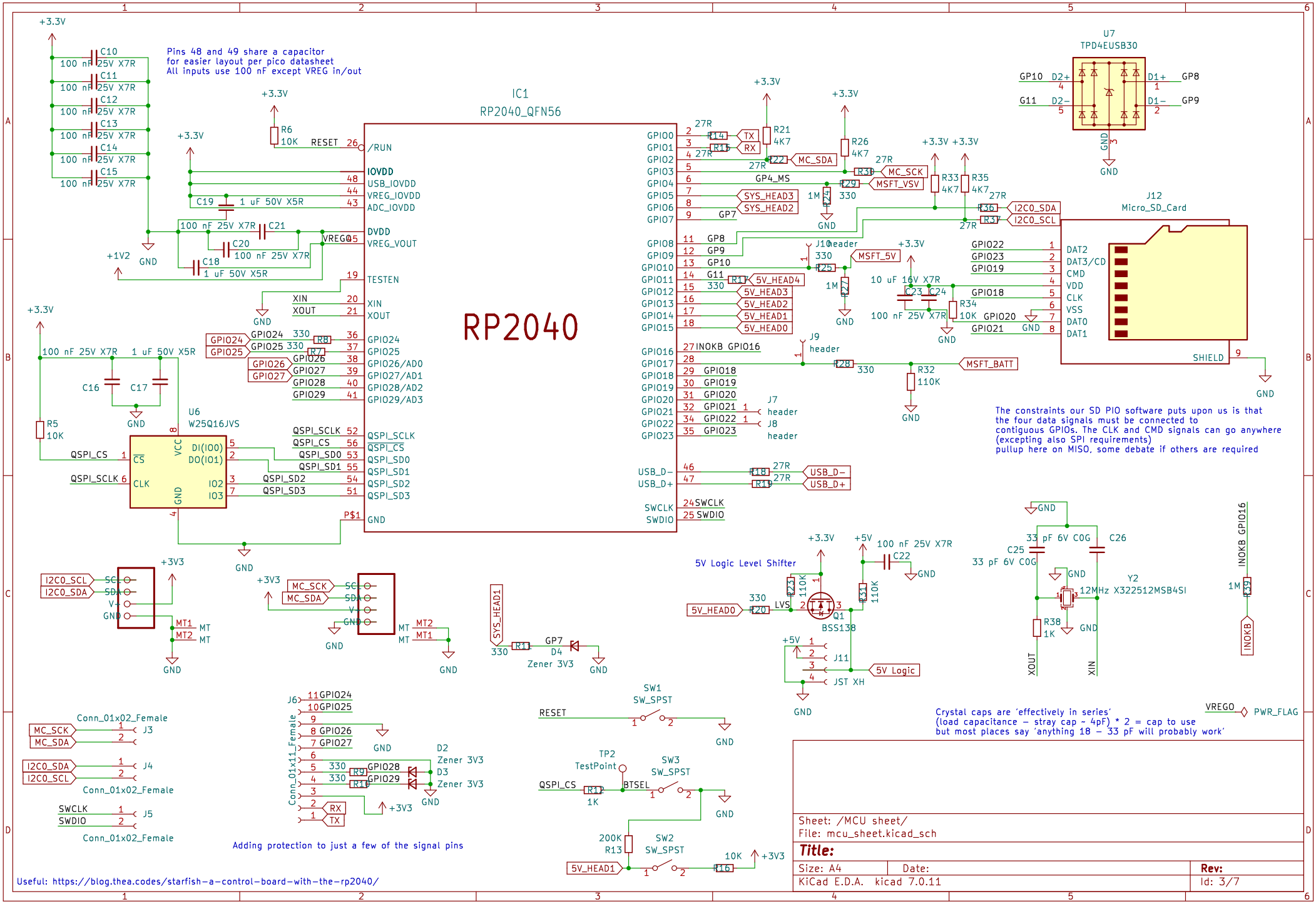
BMP390 the 7-bit device address is 111011x. The 6 MSB bits are fixed. The last bit is changeable by SDO value and can be changed during operation. Connecting SDO to GND results in slave address 1110110 (0x76); connection it to V DDIO results in slave address 1110111 (0x77), which is the same as BMP180's I²C address. The SDO pin cannot be left floating; if left floating, the I²C address will be undefined

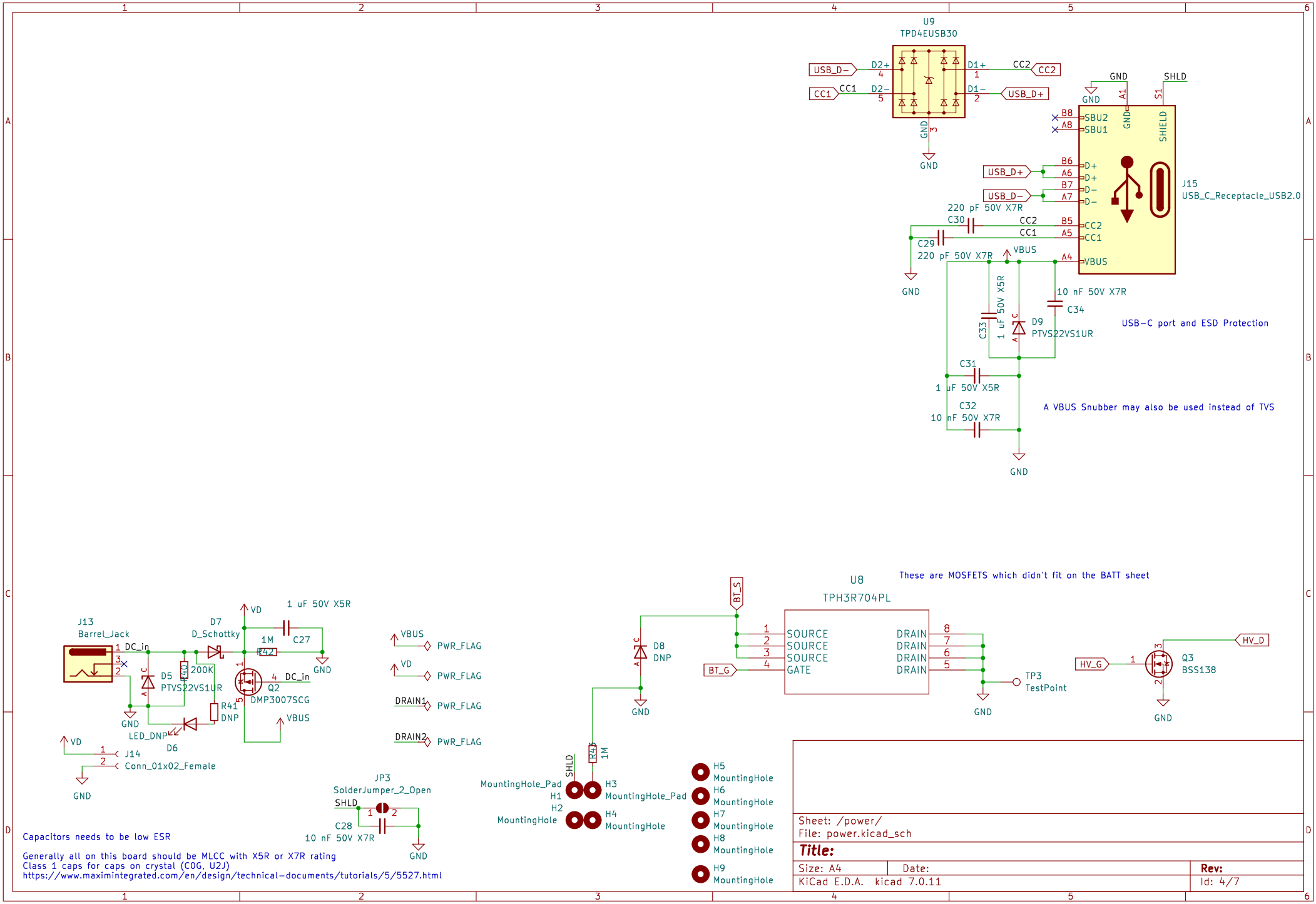
Sheet: /Sensor Sheet/
File: sensors.kicad_sch

Title:

Size: A4 | Date: | **Rev:**

KiCad E.D.A. kicad 7.0.11 | Id: 2/7


RP2040
IC1
RP2040_QFN56
Pins 48 and 49 share a capacitor
for easier layout per pico datasheet
All inputs use 100 nF except VREG in/out
U7
TPD4EUSB30
J12
Micro_SD_Card
U6
W25Q16JVS
The constraints our SD PIO software puts upon us is that
the four data signals must be connected to
contiguous GPIOs. The CLK and CMD signals can go anywhere
(excepting also SPI requirements)
pullup here on MISO, some debate if others are required
5V Logic Level Shifter
BSS138
Crystal caps are 'effectively in series'
(load capacitance - stray cap ~ 4pF) * 2 = cap to use
but most places say 'anything 18 - 33 pF will probably work'
Adding protection to just a few of the signal pins
Useful: https://blog.thea.codes/starfish-a-control-board-with-the-rp2040/
Sheet: /MCU sheet/
File: mcu_sheet.kicad_sch
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. kicad 7.0.11
Id: 3/7


U9
TPD4EUSB30
USB_D−
CC1
D2+
D2−
D1+
D1−
CC2
USB_D+
GND
J15
USB_C_Receptacle_USB2.0
SHLD
SBU2
SBU1
SHIELD
B8
A8
B6
A6
B7
A7
B5
A5
A4
220 pF 50V X7R
C30
C29
CC2
CC1
VBUS
10 nF 50V X7R
C34
1 uF 50V X5R
C33
D9
PTVS22VS1UR
C31
C32
USB−C port and ESD Protection
A VBUS Snubber may also be used instead of TVS
These are MOSFETS which didn't fit on the BATT sheet
U8
TPH3R704PL
BT_S
BT_G
SOURCE
GATE
DRAIN
D8
DNP
TP3
TestPoint
HV_D
HV_G
Q3
BSS138
J13
Barrel_Jack
DC_in
D7
D_Schottky
1 uF 50V X5R
C27
1M
R42
200K
R40
D5
PTVS22VS1UR
Q2
DMP3007SCG
R41
DNP
LED_DNP
D6
J14
Conn_01x02_Female
VD
PWR_FLAG
DRAIN1
DRAIN2
JP3
SolderJumper_2_Open
C28
10 nF 50V X7R
R43
MountingHole_Pad
MountingHole
H1
H2
H3
H4
H5
H6
H7
H8
H9
Capacitors needs to be low ESR
Generally all on this board should be MLCC with X5R or X7R rating
Class 1 caps for caps on crystal (C0G, U2J)
https://www.maximintegrated.com/en/design/technical-documents/tutorials/5/5527.html
Sheet: /power/
File: power.kicad_sch
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. kicad 7.0.11
Id: 4/7

The slave address of the IC is 0xD2h/0xD3h
The least significant bit is
the read/write indicator (1 for read, 0 for write)

Vsys is guaranteed 4V – 14V
Vsys should be between Vsysmin (8.4V programmable)
to Vbattreg (12V, programmable)
Max Current 10A

TI has some nice alternatives
but currently impossible to find
bq25792

Extra header
for XT60 connector (7.2 mm pitch)
From other pin of terminal

lower the inductor DCR (DC resistance) is, the higher the buck–boost efficiency is
DR127–3R3–R, SER2915, 7443320330

No voltage is measured across the config resistors after powerup.
The voltages are sampled at powerup and then each current source is disabled.

EVAL has 15nF caps on CSINP and CSINN, 10 nF used here

U10
MAX77960x06

JP6
Jumper_2_Open
CNFG PVL
2S Lipo Selector

3S Lipo Plug, JST

3S ONLY

EVAL has 220K and 470K here
datasheet has 470K and 910K
I used 200K + 200K and 1M as used elsewhere already

Hot Insertion Protection RC, copied from datasheet

DMN3016LFDE in EVAL, N–channel

FETS are Terminals, controlled by switch from MCU

The primary distinction between an N–Channel and a P–Channel MOSFET
is that the N–Channel is usually connected to the Ground (–) side of the load,
while the P–Channel is connected to the VCC (+) side

TVS good for surge suppression
these TVS have some inductive load rev protection (500 mah)
but the Schottky is needed for higher current reverse protection
but Schottky has high leakage current

Voltage/Power sensing resistor

LIPO Charging LED
STAT goes high when no charge or no power
Cycles low/high when charging and low when charged

STAT tied to PVL with 200K in datasheet, no LED there

THM to GND means battery removal cannot be detected,
but allows charging
THM to AVL is equivalent to battery removal detection,
and prevents charging

This switch is here to assure safe removal of battery

Optional thermistor, as shown can only detect too hot to charge
unless THM to GND pull down is removed

Extra N–channel is needed to make it
high 3.3V = on, GND = off

Sheet: /Batt/
File: batt.kicad_sch

Title:

Size: A4 | Date: | **Rev:**
KiCad E.D.A. kicad 7.0.11 | Id: 5/7

Useful: https://community.element14.com/products/roadtest/b/blog/posts/mps-mpm54304-evm-roadtest---snippets

"For parallel mode, you are welcome to connect the
SW1 and SW2 pins together or leave them untied.
We suggest floating them in parallel mode to avoid the wrong SMT."

Different Suffix versions of MPM54304 have different default settings
for which VOUT can be parallel (diff startup times otherwise).
Suffix 0002 is expected, 0004 should also work without modification

R0 between the two separate V5 nets should allow for the not-parallel setup
of the 0001 and 0003 suffixes, DNP this part for those

U12
MPM54304-0002

By default, the I2C slave address is 0x68

GPIO default is PG (power good)

EVAL board shows PG to VCC with 100K
I'm using 200K just for simpler BOM

Space for optional large ripple capacitors

1150 for 5.118V
1170 for 5.041V
1180 for 5.003
against 8660
can do 8660 + 27R for slightly higher

4K7 v 1K for 3.42V
8660 v 1910 for 3.32V
(1180 + 470 + 470) v 470 for 3.306V
8660 + 1k + 1k vs 1180 + 1180 for 3.363
10k vs 1k + 1180 for 3.352
4k7 vs 1k + 27 for 3.346

should work with any 3 same R for 1.8V

Voltage Dividers to set Buck V outs

FB Voltage on must suffixes is 0.6 V
which I assume is the same as Vref

A feed-forward cap on these Vout to FB can be used
Helpful when large current loads are present
33 nF is recommended

Using same USB TVS
to minimize unique parts

EN pin needs to be pulled high to activate (so to VS)
but can't tolerate more than 6V, hence the divider
150k and 49.9k is used in example

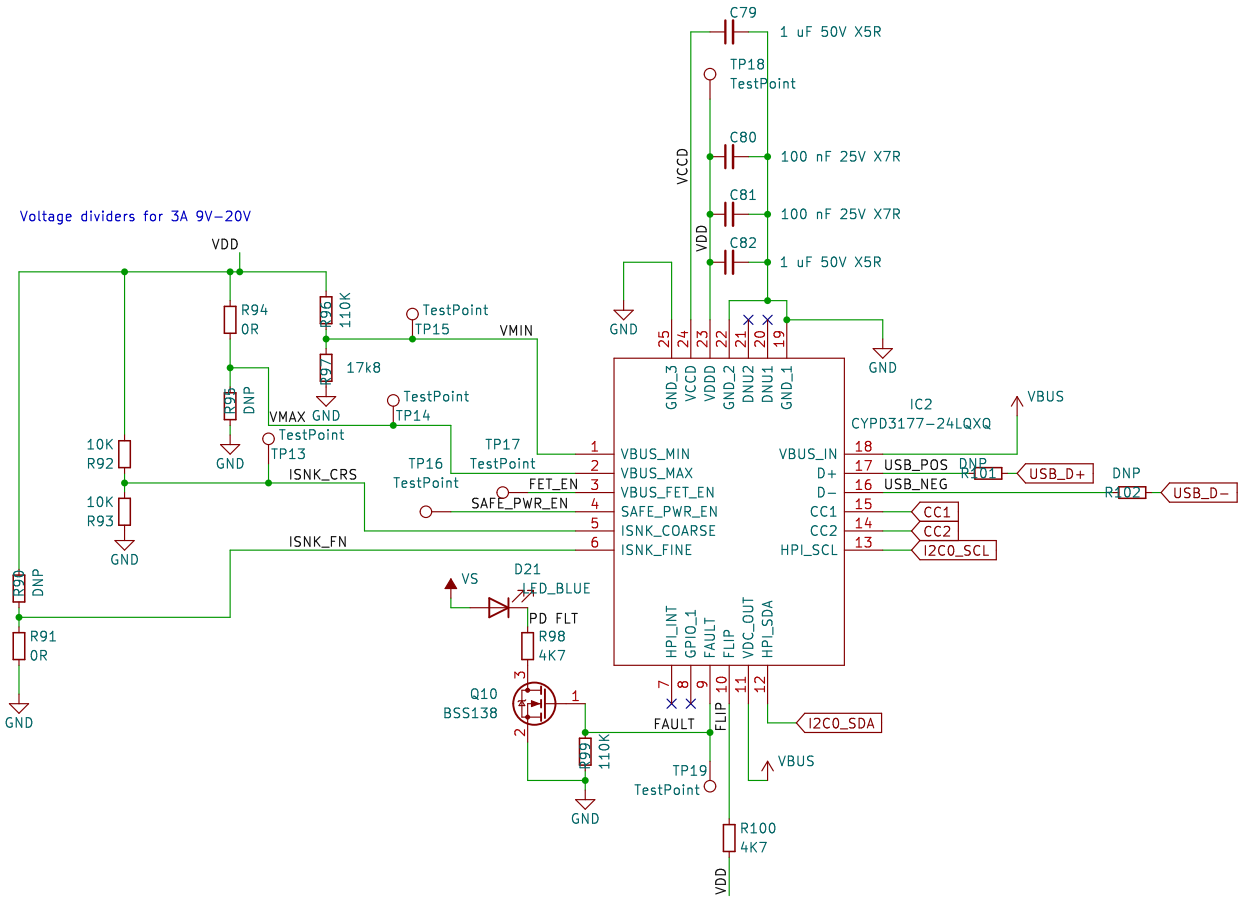
Internally EN is pulled low by 2M resistor

This capacitor enables soft start, not required here

Filtering option:
27R to each Vm
and 100 nF to 1 uF cap
between Vm + and -

i2c address: 1000000 with gnd and gnd, 0x40

Sheet: /drivers/	
File: drivers.kicad_sch	
Title:	
Size: A4	Date:
KiCad E.D.A. kicad 7.0.11	**Rev:** Id: 6/7



Sheet: /usb_pd/		
File: usb_pd.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 7.0.11		Id: 7/7