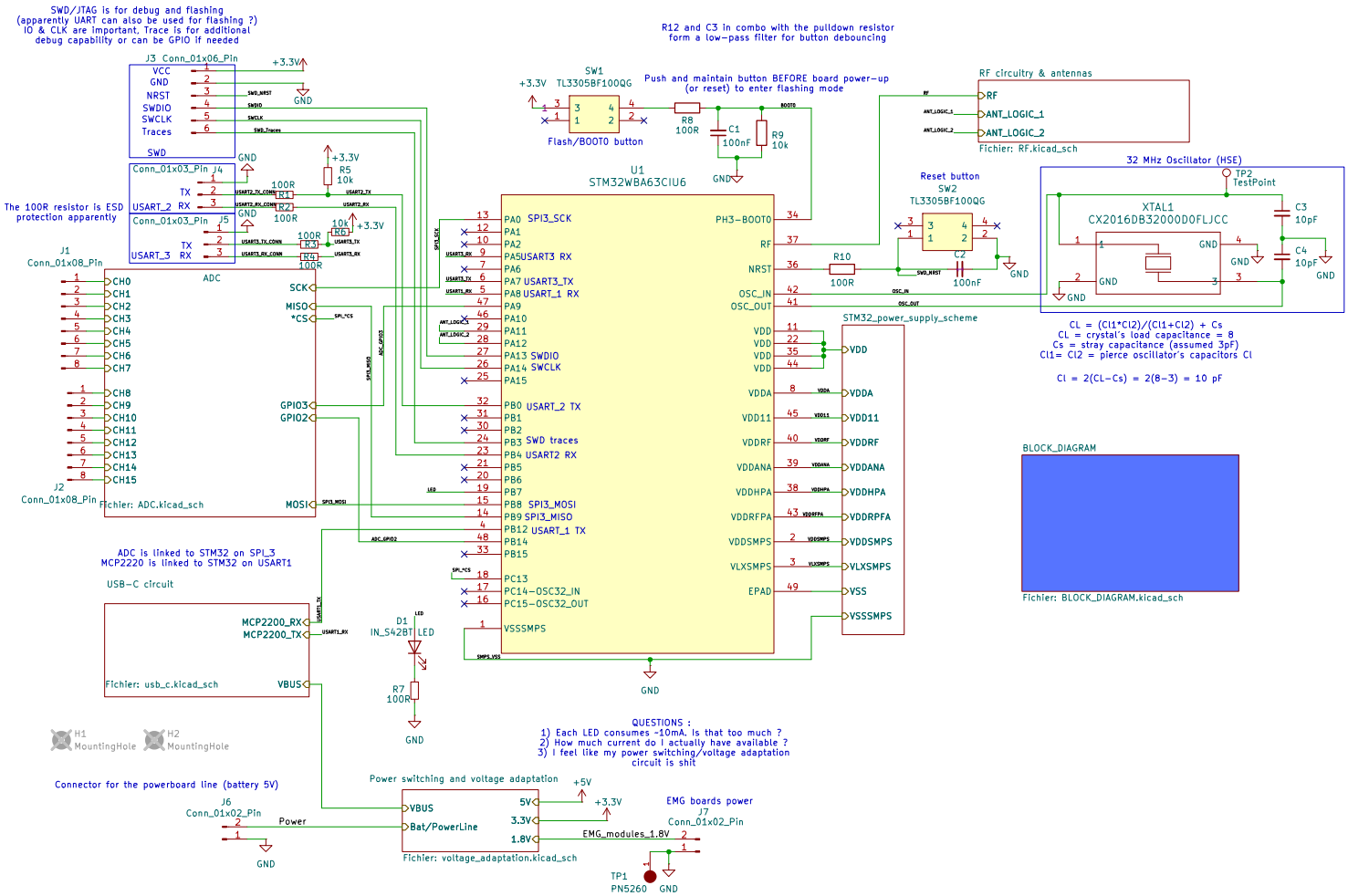
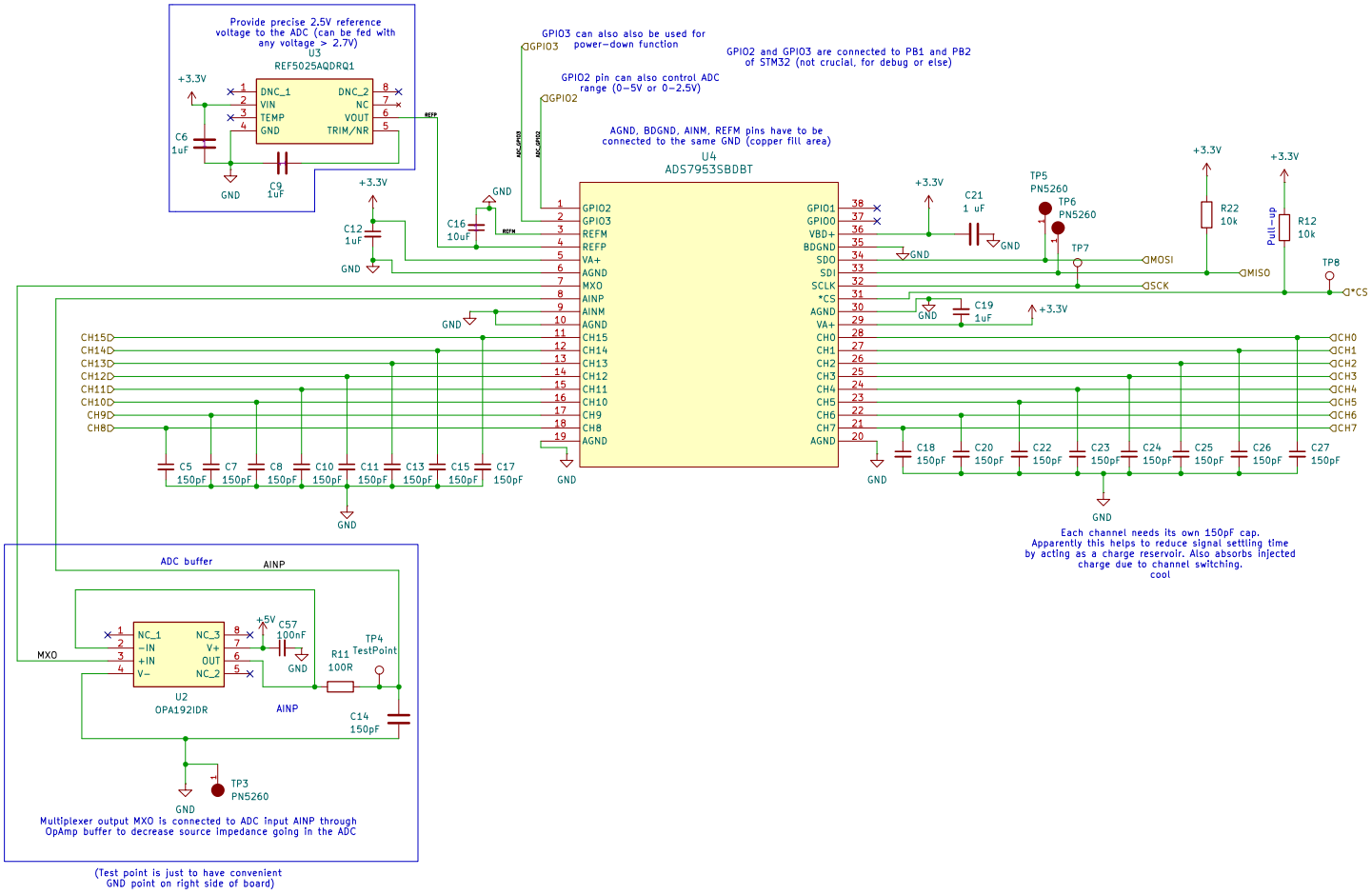
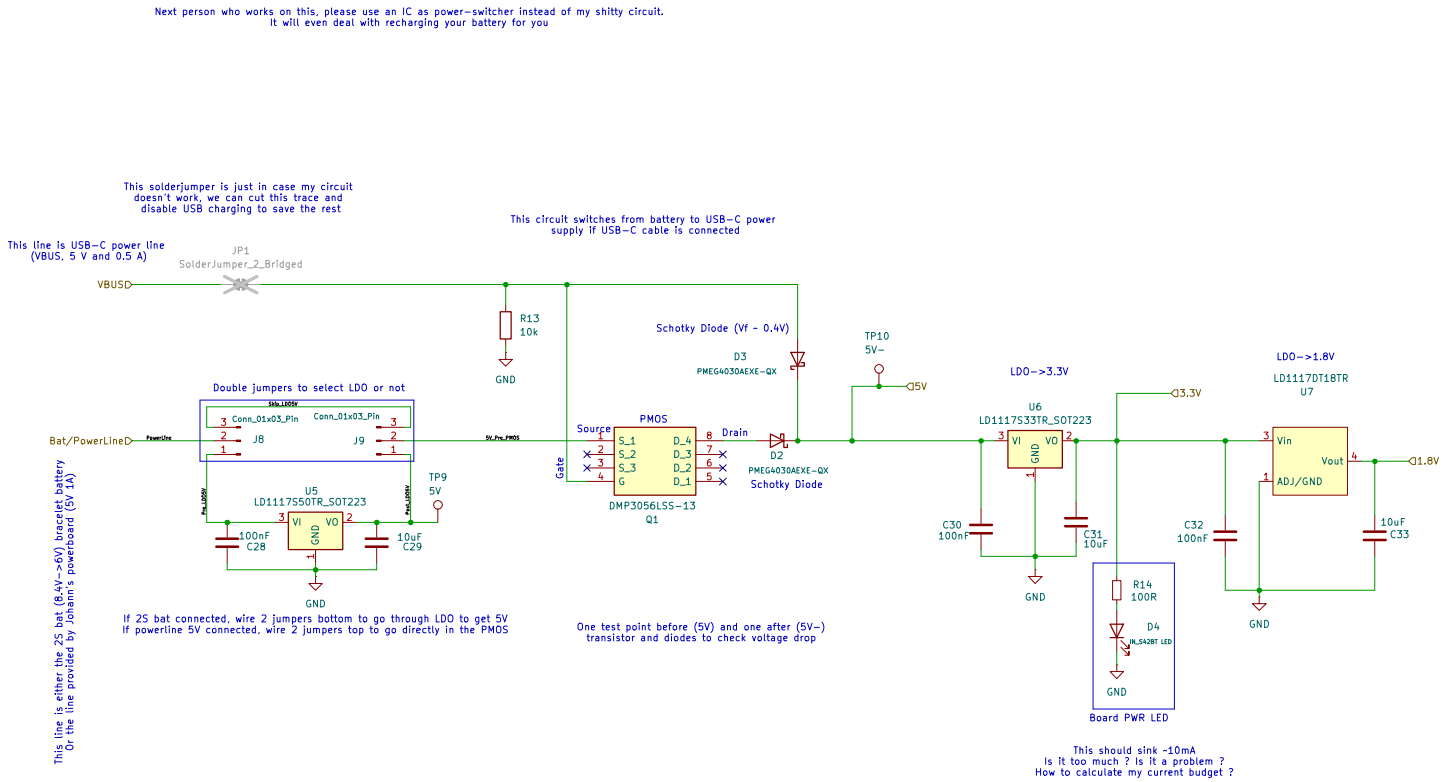
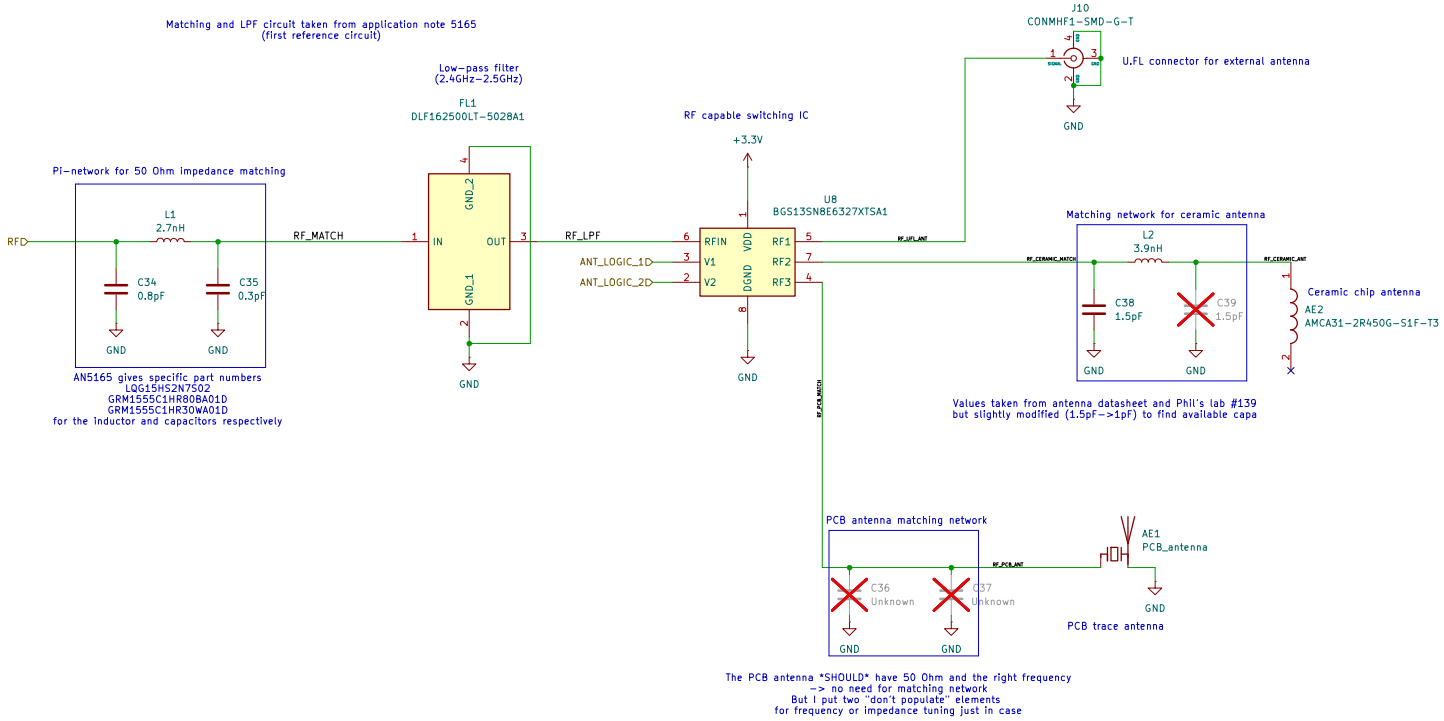
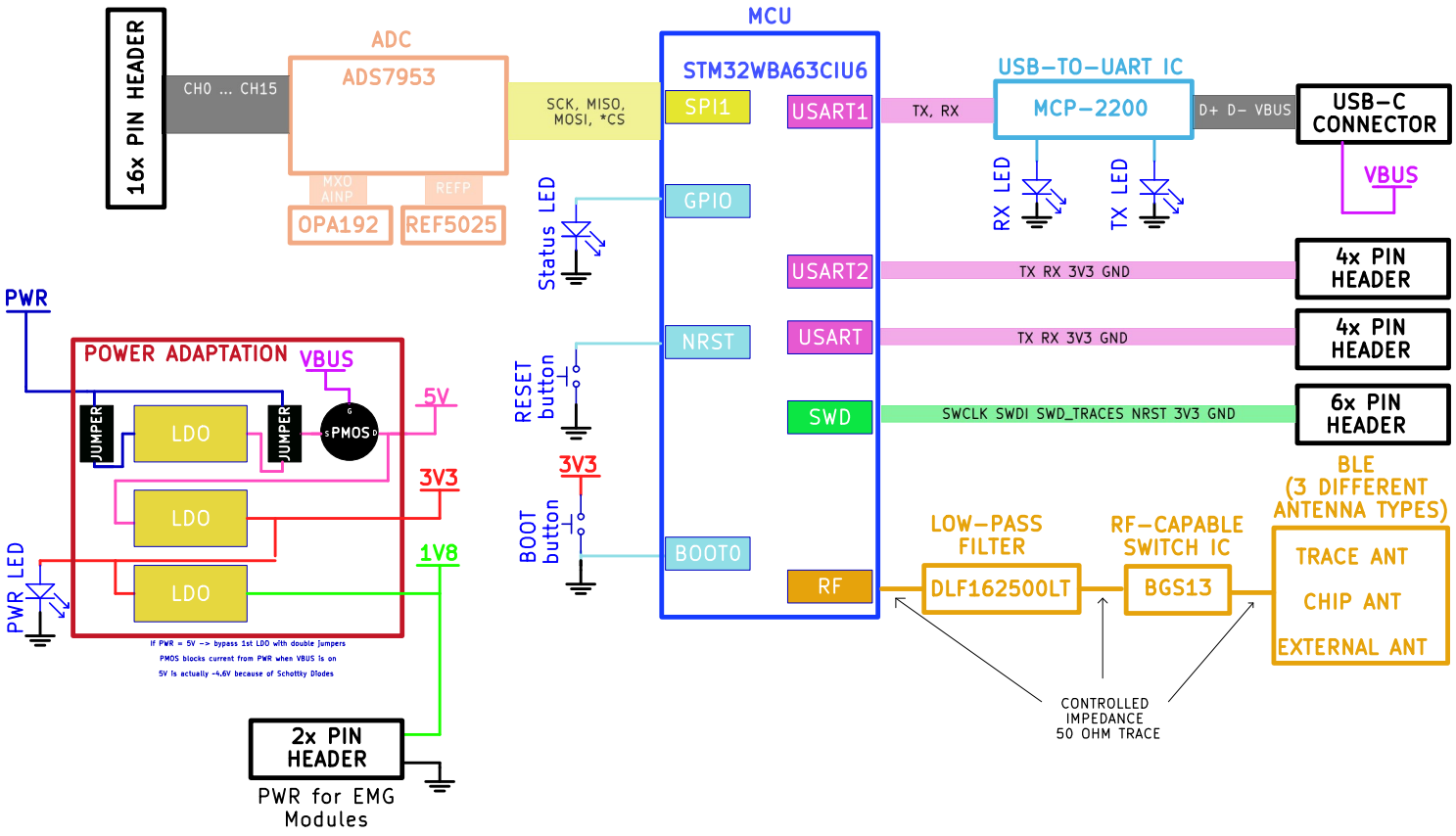
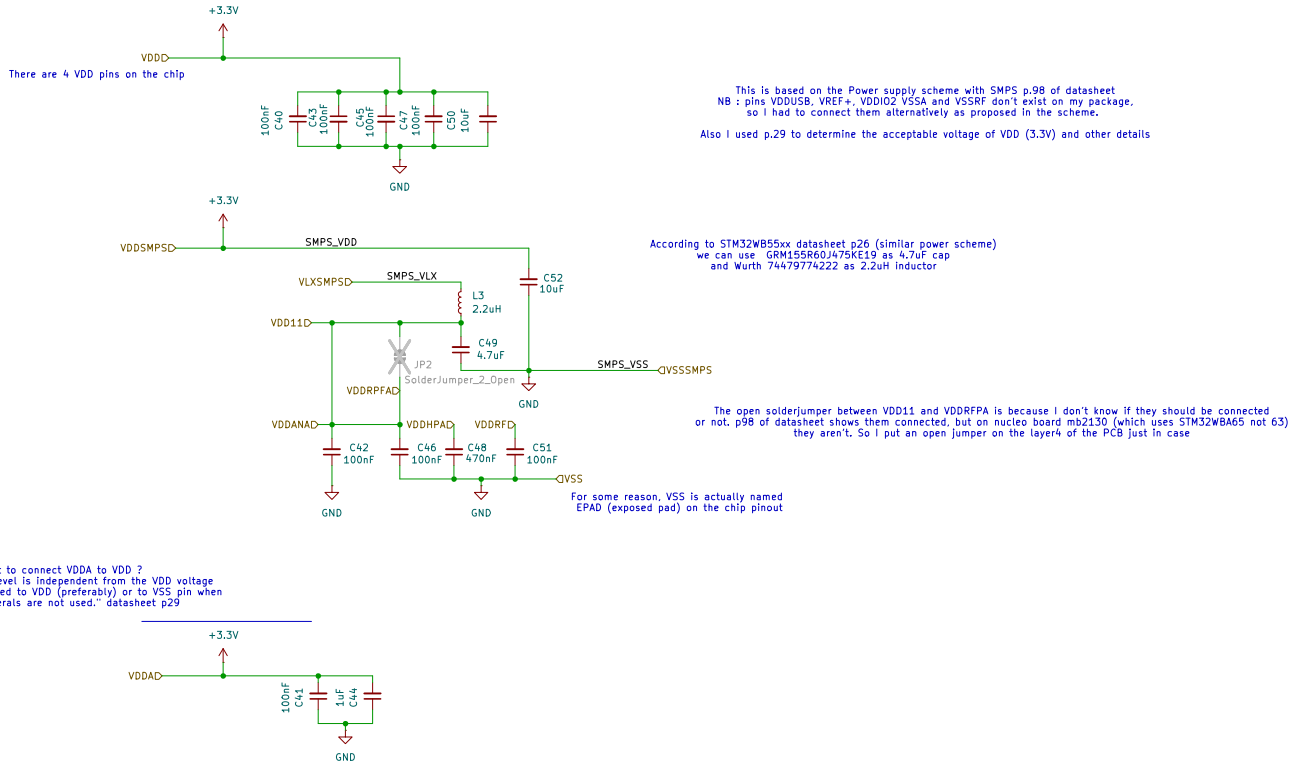

SWD/JTAG is for debug and flashing
(apparently UART can also be used for flashing ?)
IO & CLK are important, Trace is for additional
debug capability or can be GPIO if needed
J3 Conn_01x06_Pin
VCC
GND
NRST
SWDIO
SWCLK
Traces
SWD
+3.3V
GND
Conn_01x03_Pin J4
USART_2 TX
USART_2 RX
Conn_01x03_Pin J5
USART_3 TX
USART_3 RX
The 100R resistor is ESD
protection apparently
100R
10k
J1
Conn_01x08_Pin
CH0
CH1
CH2
CH3
CH4
CH5
CH6
CH7
CH8
CH9
CH10
CH11
CH12
CH13
CH14
CH15
J2
Conn_01x08_Pin
ADC
SCK
MISO
*CS
GPIO3
GPIO2
MOSI
Fichier: ADC.kicad_sch
ADC is linked to STM32 on SPI_3
MCP2220 is linked to STM32 on USART1
USB-C circuit
MCP2200_RX
MCP2200_TX
VBUS
Fichier: usb_c.kicad_sch
D1
IN_S42BT LED
R7
100R
GND
H1
MountingHole
H2
MountingHole
Connector for the powerboard line (battery 5V)
J6
Conn_01x02_Pin
Power
GND
R12 and C3 in combo with the pulldown resistor
form a low-pass filter for button debouncing
SW1
TL3305BF100QG
+3.3V
Flash/BOOT0 button
Push and maintain button BEFORE board power-up
(or reset) to enter flashing mode
R8
100R
C1
100nF
R9
10k
GND
U1
STM32WBA63CIU6
PA0 SPI3_SCK
PA1
PA2
PA5USART3 RX
PA6
PA7 USART3_TX
PA8 USART_1 RX
PA9
PA10
PA11
PA12
PA13 SWDIO
PA14 SWCLK
PA15
PB0 USART_2 TX
PB1
PB2
PB3 SWD traces
PB4 USART2 RX
PB5
PB6
PB7
PB8 SPI3_MOSI
PB9 SPI3_MISO
PB12 USART_1 TX
PB14
PB15
PC13
PC14-OSC32_IN
PC15-OSC32_OUT
VSSSMPS
PH3-BOOT0
RF
NRST
OSC_IN
OSC_OUT
VDD
VDD
VDD
VDD
VDDA
VDD11
VDDRF
VDDANA
VDDHPA
VDDRFPA
VDDSMPS
VLXSMPS
EPAD
STM32_power_supply_scheme
VDD
VDDA
VDD11
VDDRF
VDDANA
VDDHPA
VDDRFPA
VDDSMPS
VLXSMPS
VSS
VSSSMPS
Reset button
SW2
TL3305BF100QG
R10
100R
C2
100nF
RF circuitry & antennas
RF
ANT_LOGIC_1
ANT_LOGIC_2
Fichier: RF.kicad_sch
32 MHz Oscillator (HSE)
TP2
TestPoint
XTAL1
CX2016DB32000D0FLJCC
C3
10pF
C4
10pF
CL = (Cl1*Cl2)/(Cl1+Cl2) + Cs
CL = crystal's load capacitance = 8
Cs = stray capacitance (assumed 3pF)
Cl1= Cl2 = pierce oscillator's capacitors Cl
Cl = 2(CL-Cs) = 2(8-3) = 10 pF
BLOCK_DIAGRAM
Fichier: BLOCK_DIAGRAM.kicad_sch
QUESTIONS :
1) Each LED consumes ~10mA. Is that too much ?
2) How much current do I actually have available ?
3) I feel like my power switching/voltage adaptation
circuit is shit
Power switching and voltage adaptation
VBUS
Bat/PowerLine
5V
3.3V
1.8V
+5V
+3.3V
EMG_modules_1.8V
Fichier: voltage_adaptation.kicad_sch
EMG boards power
J7
Conn_01x02_Pin
TP1
PN5260
GND

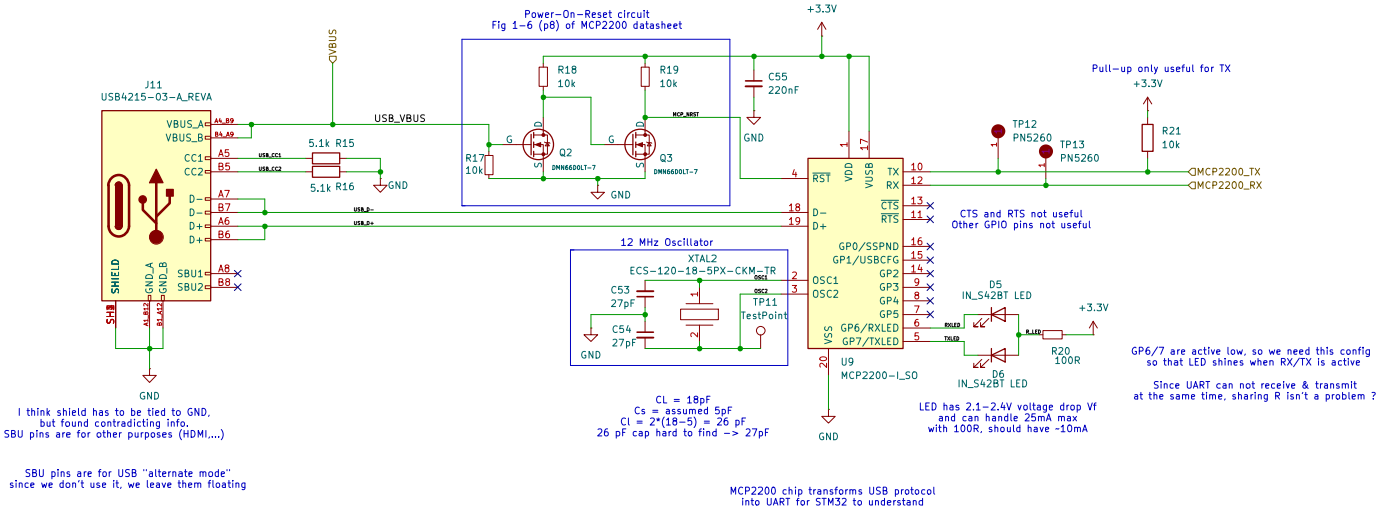

Provide precise 2.5V reference voltage to the ADC (can be fed with any voltage > 2.7V)
U3
REF5025AQDRQ1
DNC_1
VIN
TEMP
GND
DNC_2
NC
VOUT
TRIM/NR
+3.3V
C6 1uF
C9 1uF
REFP
GPIO3 can also also be used for power-down function
GPIO3
GPIO2 and GPIO3 are connected to PB1 and PB2 of STM32 (not crucial, for debug or else)
GPIO2 pin can also control ADC range (0-5V or 0-2.5V)
GPIO2
AGND, BDGND, AINM, REFM pins have to be connected to the same GND (copper fill area)
U4
ADS7953SBDBT
GPIO2
GPIO3
REFM
REFP
VA+
AGND
MXO
AINP
AINM
AGND
CH15
CH14
CH13
CH12
CH11
CH10
CH9
CH8
AGND
GPIO1
GPIO0
VBD+
BDGND
SDO
SDI
SCLK
*CS
AGND
VA+
CH0
CH1
CH2
CH3
CH4
CH5
CH6
CH7
AGND
C16 10uF
C12 1uF
C21 1 uF
C19 1uF
TP5 PN5260
TP6 PN5260
TP7
TP8
R22 10k
R12 10k
Pull-up
MOSI
MISO
SCK
*CS
C5 150pF
C7 150pF
C8 150pF
C10 150pF
C11 150pF
C13 150pF
C15 150pF
C17 150pF
C18 150pF
C20 150pF
C22 150pF
C23 150pF
C24 150pF
C25 150pF
C26 150pF
C27 150pF
Each channel needs its own 150pF cap. Apparently this helps to reduce signal settling time by acting as a charge reservoir. Also absorbs injected charge due to channel switching. cool
ADC buffer
AINP
MXO
U2
OPA192IDR
NC_1
-IN
+IN
V-
NC_3
V+
OUT
NC_2
+5V
C57 100nF
R11 100R
TP4 TestPoint
C14 150pF
TP3 PN5260
Multiplexer output MXO is connected to ADC input AINP through OpAmp buffer to decrease source impedance going in the ADC
(Test point is just to have convenient GND point on right side of board)


Next person who works on this, please use an IC as power-switcher instead of my shitty circuit.
It will even deal with recharging your battery for you
This solderjumper is just in case my circuit doesn't work, we can cut this trace and disable USB charging to save the rest
This line is USB-C power line (VBUS, 5 V and 0.5 A)
JP1
SolderJumper_2_Bridged
VBUS
R13
10k
GND
This circuit switches from battery to USB-C power supply if USB-C cable is connected
Schotky Diode (Vf ~ 0.4V)
D3
PMEG4030AEXE-QX
TP10
5V-
5V
LDO->3.3V
3.3V
LDO->1.8V
LD1117DT18TR
U7
Double jumpers to select LDO or not
Conn_01x03_Pin
J8
J9
Bat/PowerLine
5V_Pre_PMOS
PMOS
Source
Drain
Gate
S_1
S_2
S_3
G
D_4
D_3
D_2
D_1
DMP3056LSS-13
Q1
D2
PMEG4030AEXE-QX
Schotky Diode
U6
LD1117S33TR_SOT223
VI
VO
GND
Vin
Vout
ADJ/GND
1.8V
U5
LD1117S50TR_SOT223
TP9
5V
100nF
C28
10uF
C29
C30
100nF
C31
10uF
C32
100nF
10uF
C33
R14
100R
D4
IN_S42BT LED
Board PWR LED
This line is either the 2S bat (8.4V->6V) bracelet battery
Or the line provided by Johann's powerboard (5V 1A)
If 2S bat connected, wire 2 jumpers bottom to go through LDO to get 5V
If powerline 5V connected, wire 2 jumpers top to go directly in the PMOS
One test point before (5V) and one after (5V-)
transistor and diodes to check voltage drop
This should sink ~10mA
Is it too much ? Is it a problem ?
How to calculate my current budget ?


Matching and LPF circuit taken from application note 5165
(first reference circuit)
Low-pass filter
(2.4GHz-2.5GHz)
FL1
DLF162500LT-5028A1
RF capable switching IC
+3.3V
U8
BGS13SN8E6327XTSA1
J10
CONMHF1-SMD-G-T
U.FL connector for external antenna
GND
Pi-network for 50 Ohm impedance matching
L1
2.7nH
RFO
RF_MATCH
C34
0.8pF
C35
0.3pF
GND
GND
AN5165 gives specific part numbers
LQG15HS2N7S02
GRM1555C1HR80BA01D
GRM1555C1HR30WA01D
for the inductor and capacitors respectively
GND_2
GND_1
IN
OUT
RF_LPF
ANT_LOGIC_1
ANT_LOGIC_2
RFIN
V1
V2
VDD
DGND
RF1
RF2
RF3
RF_UFL_ANT
RF_CERAMIC_MATCH
RF_CERAMIC_ANT
RF_PCB_MATCH
RF_PCB_ANT
Matching network for ceramic antenna
L2
3.9nH
C38
1.5pF
C39
1.5pF
Ceramic chip antenna
AE2
AMCA31-2R450G-S1F-T3
Values taken from antenna datasheet and Phil's lab #139
but slightly modified (1.5pF->1pF) to find available capa
PCB antenna matching network
C36
Unknown
C37
Unknown
AE1
PCB_antenna
PCB trace antenna
The PCB antenna *SHOULD* have 50 Ohm and the right frequency
-> no need for matching network
But I put two "don't populate" elements
for frequency or impedance tuning just in case


16x PIN HEADER
CH0 ... CH15
ADC
ADS7953
MXO
AINP
REFP
OPA192
REF5025
SCK, MISO, MOSI, *CS
MCU
STM32WBA63CIU6
SPI1
GPIO
NRST
BOOT0
USART1
USART2
USART
SWD
RF
Status LED
RESET button
BOOT button
3V3
TX, RX
USB–TO–UART IC
MCP–2200
D+ D– VBUS
USB–C CONNECTOR
VBUS
RX LED
TX LED
TX RX 3V3 GND
4x PIN HEADER
TX RX 3V3 GND
4x PIN HEADER
SWCLK SWDI SWD_TRACES NRST 3V3 GND
6x PIN HEADER
PWR
POWER ADAPTATION
VBUS
JUMPER
LDO
JUMPER
PMOS
5V
3V3
LDO
1V8
LDO
PWR LED
If PWR = 5V –> bypass 1st LDO with double jumpers
PMOS blocks current from PWR when VBUS is on
5V is actually ~4,6V because of Schottky Diodes
2x PIN HEADER
PWR for EMG Modules
LOW–PASS FILTER
DLF162500LT
RF–CAPABLE SWITCH IC
BGS13
BLE (3 DIFFERENT ANTENNA TYPES)
TRACE ANT
CHIP ANT
EXTERNAL ANT
CONTROLLED IMPEDANCE 50 OHM TRACE


+3.3V
VDDD
There are 4 VDD pins on the chip
100nF
C40
C43
100nF
C45
100nF
C47
100nF
C50
10uF
GND
This is based on the Power supply scheme with SMPS p.98 of datasheet
NB : pins VDDUSB, VREF+, VDDIO2 VSSA and VSSRF don't exist on my package,
so I had to connect them alternatively as proposed in the scheme.
Also I used p.29 to determine the acceptable voltage of VDD (3.3V) and other details
+3.3V
VDDSMPS
SMPS_VDD
According to STM32WB55xx datasheet p26 (similar power scheme)
we can use GRM155R60J475KE19 as 4.7uF cap
and Wurth 74479774222 as 2.2uH inductor
VLXSMPS
SMPS_VLX
C52
10uF
L3
2.2uH
VDD11
C49
4.7uF
JP2
SolderJumper_2_Open
SMPS_VSS
VSSSMPS
GND
VDDRFPAD
The open solderjumper between VDD11 and VDDRFPA is because I don't know if they should be connected
or not. p98 of datasheet shows them connected, but on nucleo board mb2130 (which uses STM32WBA65 not 63)
they aren't. So I put an open jumper on the layer4 of the PCB just in case
VDDANAD
VDDHPAD
VDDRFD
C42
100nF
C46
100nF
C48
470nF
C51
100nF
VSS
GND
GND
For some reason, VSS is actually named
EPAD (exposed pad) on the chip pinout
Correct to connect VDDA to VDD ?
"The VDDA voltage level is independent from the VDD voltage
and must be connected to VDD (preferably) or to VSS pin when
these peripherals are not used." datasheet p29
+3.3V
VDDA
100nF
C41
1uF
C44
GND


J11
USB4215-03-A_REVA
VBUS_A
VBUS_B
CC1
CC2
D-
D-
D+
D+
SBU1
SBU2
SHIELD
GND_A
GND_B
SHIELD
A4_B9
B4_A9
A5
B5
A7
B7
A6
B6
A8
B8
A1_B12
B1_A12
USB_CC1
USB_CC2
USB_D-
USB_D+
VBUS
USB_VBUS
5.1k R15
5.1k R16
GND
Power-On-Reset circuit
Fig 1-6 (p8) of MCP2200 datasheet
R18
10k
R19
10k
R17
10k
Q2
Q3
DMN66D0LT-7
MCP_RST
C55
220nF
+3.3V
GND
Pull-up only useful for TX
+3.3V
TP12
PN5260
TP13
PN5260
R21
10k
MCP2200_TX
MCP2200_RX
RST
VDD
VUSB
TX
RX
D-
D+
CTS
RTS
GP0/SSPND
GP1/USBCFG
GP2
GP3
GP4
GP5
GP6/RXLED
GP7/TXLED
OSC1
OSC2
VSS
U9
MCP2200-I_SO
CTS and RTS not useful
Other GPIO pins not useful
12 MHz Oscillator
XTAL2
ECS-120-18-5PX-CKM-TR
C53
27pF
C54
27pF
TP11
TestPoint
D5
IN_S42BT LED
D6
IN_S42BT LED
R20
100R
R_LED
RXLED
TXLED
GP6/7 are active low, so we need this config
so that LED shines when RX/TX is active
Since UART can not receive & transmit
at the same time, sharing R isn't a problem ?
I think shield has to be tied to GND,
but found contradicting info.
SBU pins are for other purposes (HDMI,...)
SBU pins are for USB "alternate mode"
since we don't use it, we leave them floating
CL = 18pF
Cs = assumed 5pF
Cl = 2*(18-5) = 26 pF
26 pF cap hard to find -> 27pF
LED has 2.1-2.4V voltage drop Vf
and can handle 25mA max
with 100R, should have ~10mA
MCP2200 chip transforms USB protocol
into UART for STM32 to understand