Peripherals

PowerDist

5V_Ext → 5V_Ext

File: PowerDist.kicad_sch

CAN

CAN_HIGH → CAN_H
CAN_LOW → CAN_L
CAN_RX → CAN_RX
CAN_TX → CAN_TX
V_CAN_IN → V_CAN_IN
SILENT → SILENT
STDBY → STDBY

File: CAN.kicad_sch

USB

UART_RX → USART1_TX
UART_TX → USART1_RX

File: USB.kicad_sch

Switches and LEDs

LED[1..3] → PB[3..5]
LED_HB → PA15
SW_PUSH → PB1
SW_SLIDE → PB2

File: Leds.kicad_sch

STM32

U1
STM32L431CBTx

+3.3V
VDDA
NRST
VBAT
VDD
VDD
VDD
VDDA
VSS
VSSA

The crystal and caps have to be as close to the uC as possible and no traces should go underneath these components or connections.

12p C21
8M Y1
12p C22
GND
OSC_IN
OSC_OUT

+3.3V
R2 10K
R8 10K
I2C1_SCL
I2C1_SDA

CAN_RX
CAN_TX

I2C2_SCL
I2C2_SDA
SPI2_NSS
SPI2_SCK
SPI2_MISO
SPI2_MOSI

CR1
D5V0P4UR6SO-7

FB1
FerriteBead
GNDA

Keep terminating resistors close to the pins
Make sure to put resistor at MISO at the other end

R15 49.9
R20 49.9
I2C2_PU_EN
USART1_TX
USART1_RX
SILENT
STDBY
SWDIO
SWCLK
PA15

CR2
D5V0P4UR6SO-7

IC1
TPS1120DR
1SOURCE
1GATE
2SOURCE
2GATE
1DRAIN_2
1DRAIN_1
2DRAIN_2
2DRAIN_1

R9 100k
R12 1k
R10 10K
R13 10K

Board Connectors

*Connector mates with 53307-**71*
Please consult README for pin usage guide
*** Note that the pin naming scheme is not indicative of proper usage guidelines ***

J6
SOMConnector

SPI1_NSS
SPI1_MOSI
SPI1_MISO
SPI1_SCK
SPI2_NSS
SPI2_SCK
SPI2_MISO
SPI2_MOSI
PA1
PA2
I2C1_SDA
I2C1_SCL
I2C2_SCL
I2C2_SDA

J2
CAN_In
CAN_L
CAN_H
V_CAN_IN
5V_Ext
GNDPWR
GND1

J5
CAN_Out
CAN_L
CAN_H
V_CAN_IN
5V_Ext
GNDPWR
GND1

STM32 Peripherals

VDD Bypass Caps
+3.3V
C13 4.7uF
C14 100nF
C16 100nF
C17 100nF
GND

VDDA Bypass Caps
VDDA
C12 100nF
C15 100nF
GNDA

SWD Connector
+3.3V
SWCLK
SWDIO
J1
SWD
GND

VDDA Seperation
+3.3V
FB4
FerriteBead
VDDA
C7 100nF
GNDA

Reset Button
NRST
C11 100nF
SW1
Reset
GND

Graphics/Logos

Dr. Hallock
MH1 M.3
UTSVT Logo
MH2 M.3

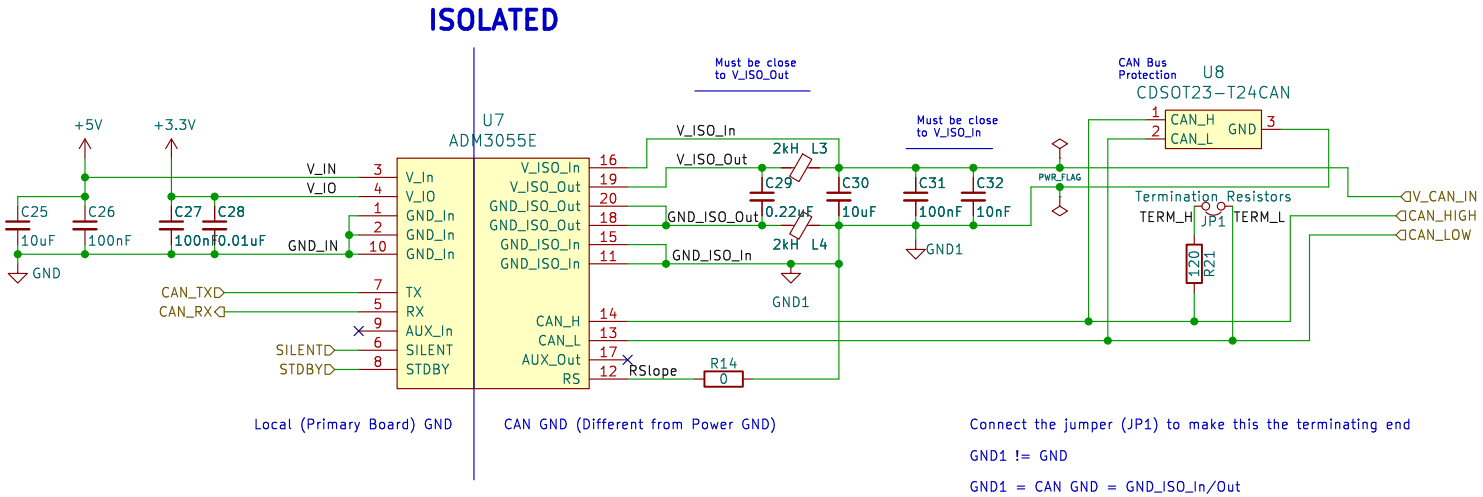
Version History

V1.0:
Using STM32L4CBT1 with 2 I2C, 2 SPI, 1 CAN, 1 UART (USB)
v1.1:
Add ADC pins as a quick hotfix

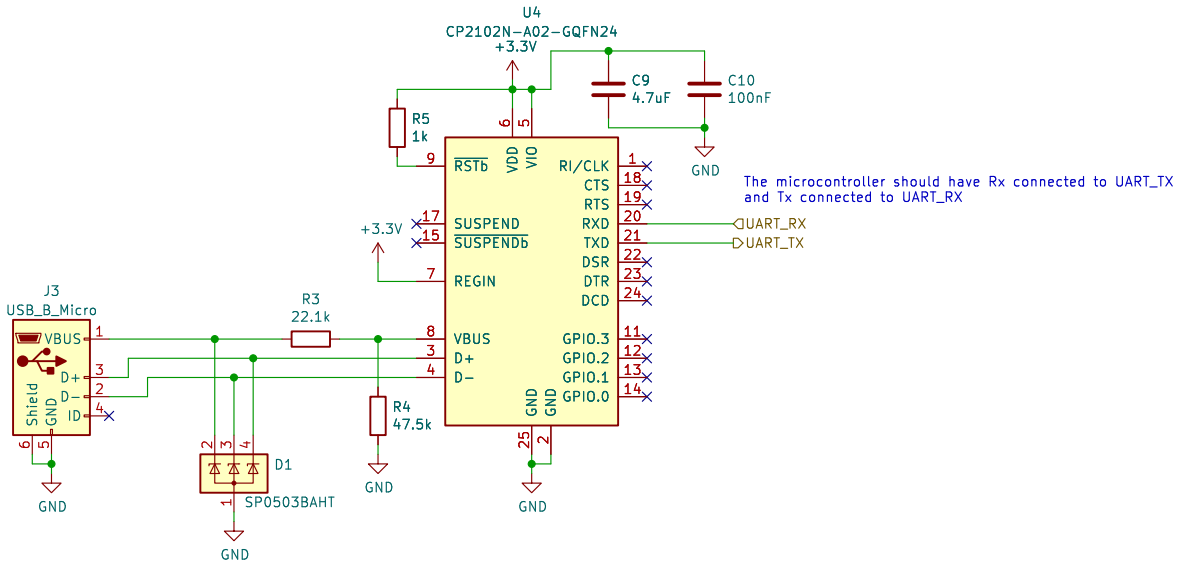
Sheet: /
File: BPS-PeripheralSOM.kicad_sch

Title:

Size: A4 | Date: | **Rev:**
KiCad E.D.A. kicad 7.0.6 | Id: 1/5

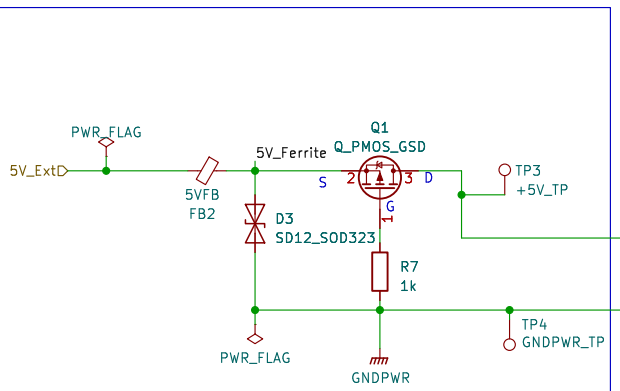


Sheet: /CAN/		
File: CAN.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 7.0.6		Id: 2/5

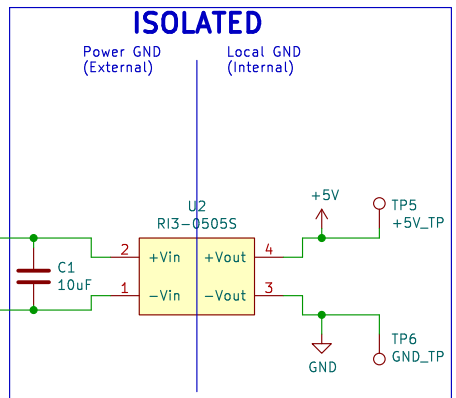


Sheet: /USB/		
File: USB.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 7.0.6		Id: 3/5

Vin protection

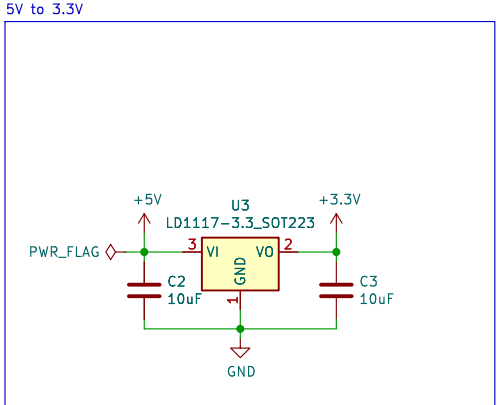


+5V (unisolated) to +5V (isolated)

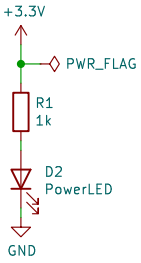


Isolated DC-DC Converter for +5V power
to an isolated +5V. The input gnd (Power GND) is
different from the output gnd (Local GND)

5V to 3.3V



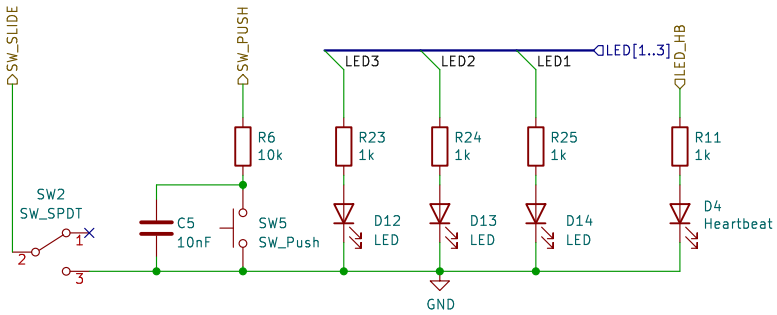
+5V is already isolated so no need to isolate +3.3V
Linear Regulator to step down +5V to +3.3V.
An isolated converter was not used for the +3.3V
line because of price and space. May need to
change this to DC-DC converter if we really
want to make the BPS more energy efficient.



Sheet: /PowerDist/
File: PowerDist.kicad_sch

Title:

Size: A4 | Date: | **Rev:**
KiCad E.D.A. kicad 7.0.6 | Id: 4/5



Sheet: /Switches and LEDs/
File: Leds.kicad_sch

Title:

Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 7.0.6		Id: 5/5