

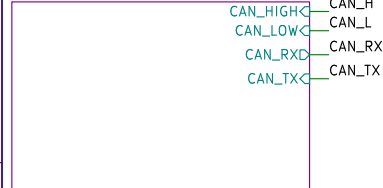
## Peripherals

### PowerDist



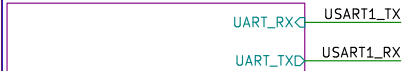
File: PowerDist.kicad\_sch

### CAN



File: CAN.kicad\_sch

### USB



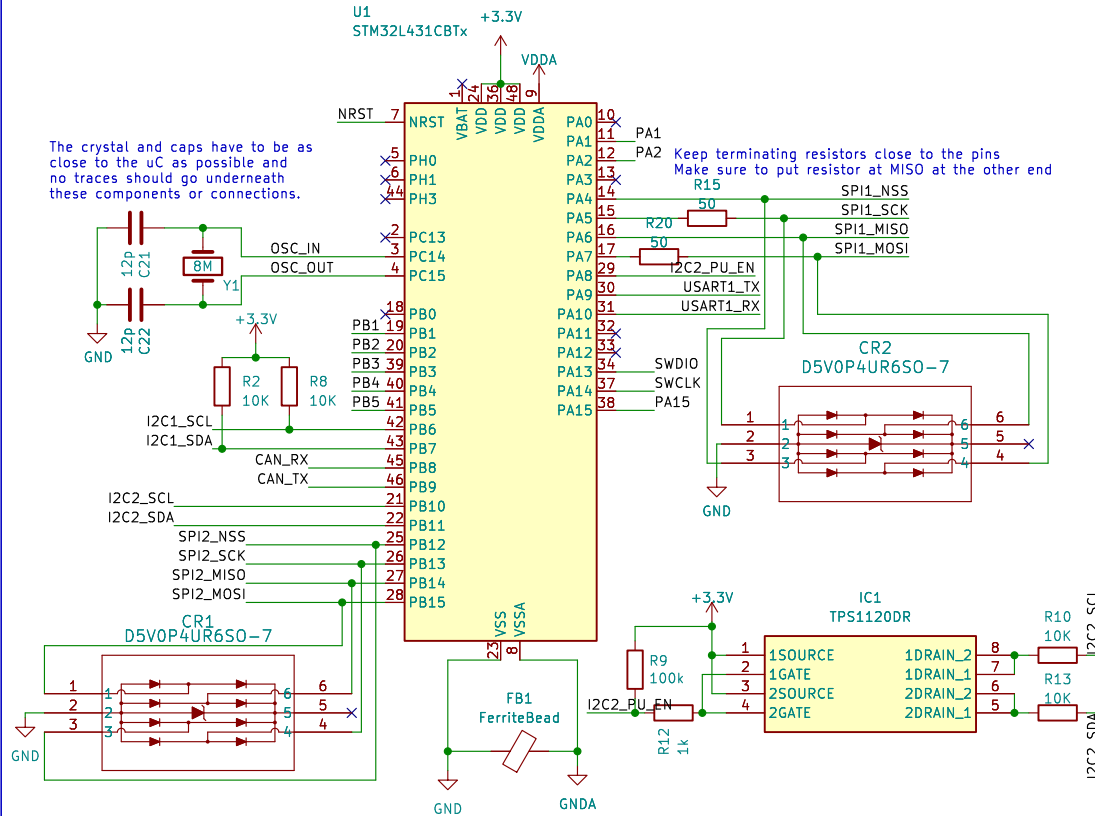
File: USB.kicad\_sch

### Switches and LEDs



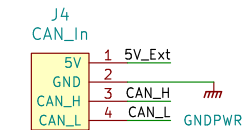
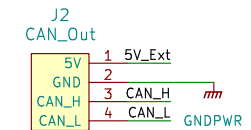
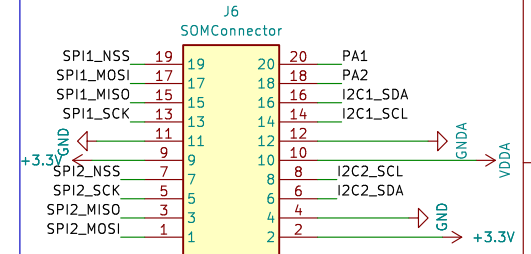
File: Leds.kicad\_sch

## STM32

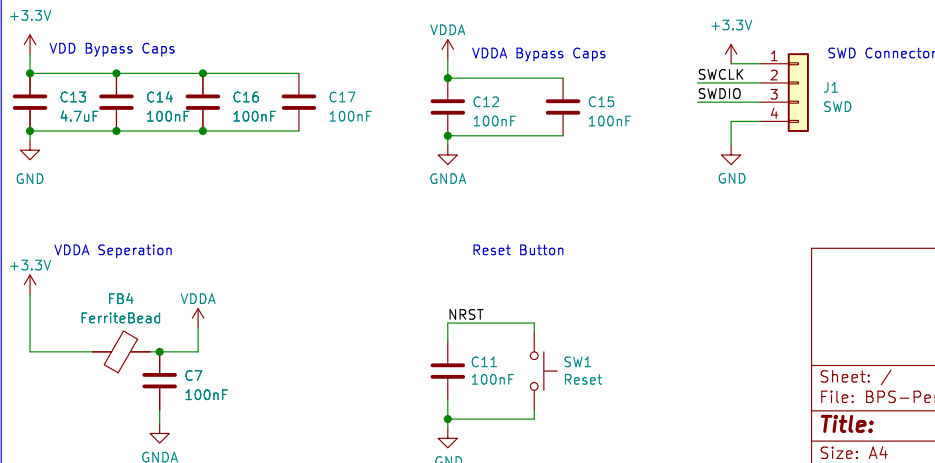


## Board Connectors

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 \*\*



## STM32 Peripherals



## Graphics/Logos



MH1  
M.3



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:

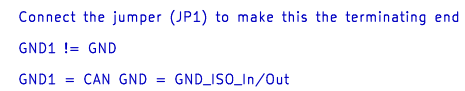
Size: A4

Date:

KiCad E.D.A. kicad 7.0.2-0

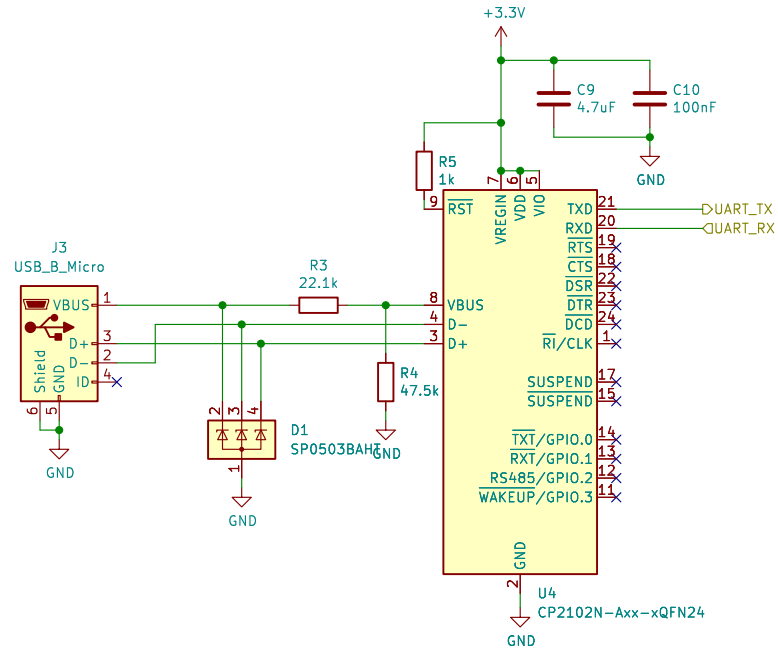
### Rev:

Id: 1/5



|                            |       |             |
|----------------------------|-------|-------------|
| Sheet: /CAN/               |       | D           |
| File: CAN.kicad_sch        |       |             |
| <b>Title:</b>              |       |             |
| Size: A4                   | Date: | <b>Rev:</b> |
| KiCad E.D.A. kicad 7.0.2—0 |       | Id: 2/5     |

The microcontroller should have Rx connected to UART\_TX  
and Tx connected to UART\_RX



Sheet: /USB/  
File: USB.kicad\_sch

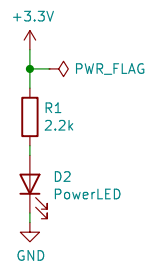
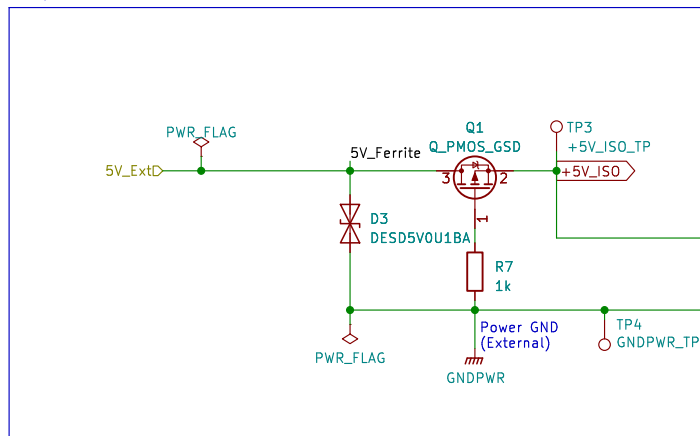
**Title:**

Size: A4  
KiCad E.D.A. kicad 7.0.2-0

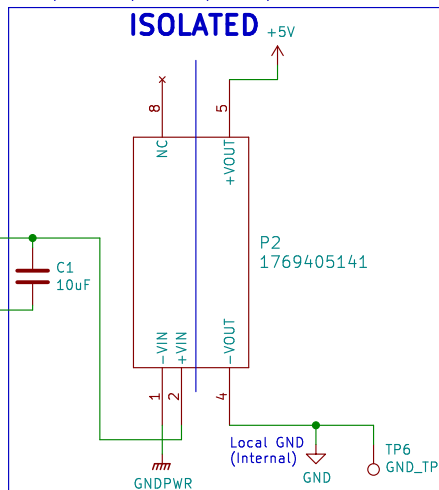
Date:

Rev:  
Id: 3/5

Vin protection

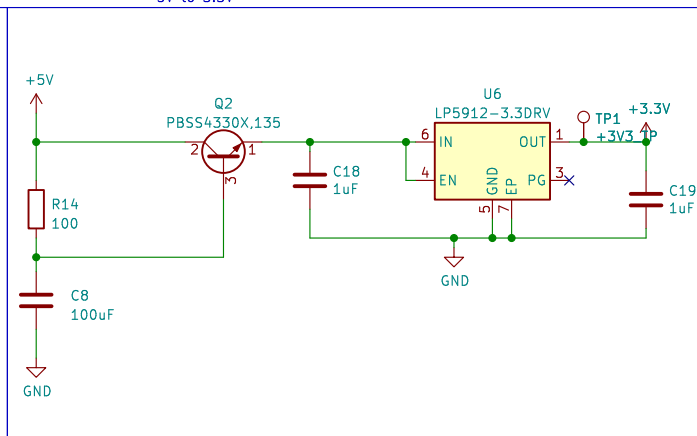


+5V (unisolated) to +5V (isolated)



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



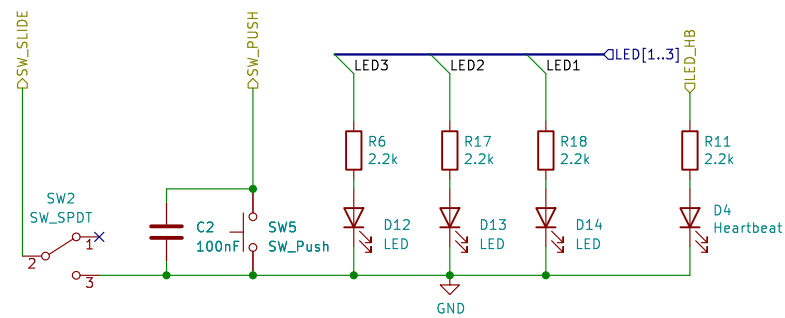
+5V is already isolated so no need to isolate +3.3V. Linear Regulator to step down +5V to +3.3V. An isolated converter was not used for the +3.3V line because of price and space. May need to change this to DC-DC converter if we really want to make the BPS more energy efficient.

Sheet: /PowerDist/  
File: PowerDist.kicad\_sch

**Title:**

Size: A4 Date:  
KiCad E.D.A. kicad 7.0.2-0

**Rev:**  
Id: 4/5



Sheet: /Switches and LEDs/  
File: Leds.kicad\_sch

**Title:**

|                            |       |
|----------------------------|-------|
| Size: A4                   | Date: |
| KiCad E.D.A. kicad 7.0.2-0 |       |

|             |
|-------------|
| <b>Rev:</b> |
| Id: 5/5     |