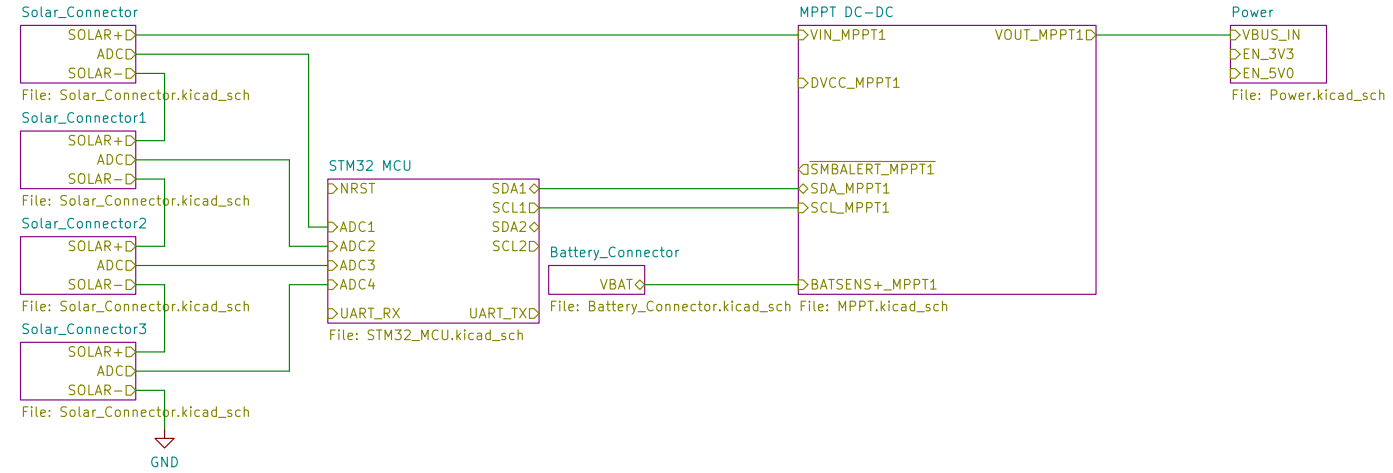


USB Power

USB_VBUS

File: USB_Power.kicad_sch

Mounting Holes

File: Mounting_Holes.kicad_sch

Board Connector

PIN_0
PIN_1
PIN_2
PIN_3
PIN_4
PIN_5
PIN_6
PIN_7
PIN_8
PIN_9
PIN_10
PIN_11
PIN_12
PIN_13
PIN_14
PIN_15
PIN_16

File: Board_Connector.kicad_sch

Sheet: /
File: EPS.kicad_sch

Title:

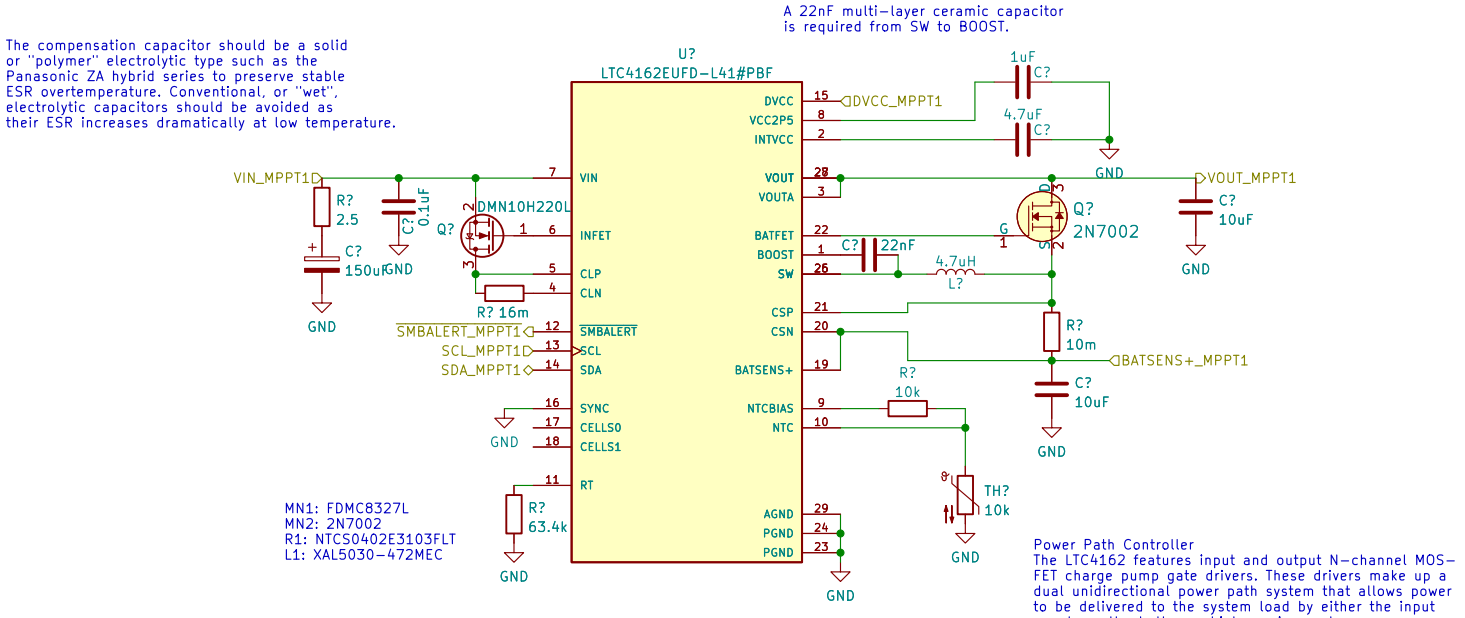
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.11−2627ca5db0~126~ubuntu20.04.1		Id: 1/12

Table 5. CELLS0 and CELLS1 Pin Mapping to Series Cell Count

CELLS1	CELLS0	cell_count
$INTV_{CC}$	$INTV_{CC}$	1
$INTV_{CC}$	VCC2P5	2
$INTV_{CC}$	GND	3
VCC2P5	$INTV_{CC}$	4
VCC2P5	VCC2P5	5
VCC2P5	GND	6
GND	$INTV_{CC}$	7
GND	VCC2P5	8

Figure 5 shows an example of a proper Kelvin connection to the current sense resistors.

Figure 5. Kelvin Current Sensing with an 0805 Resistor.

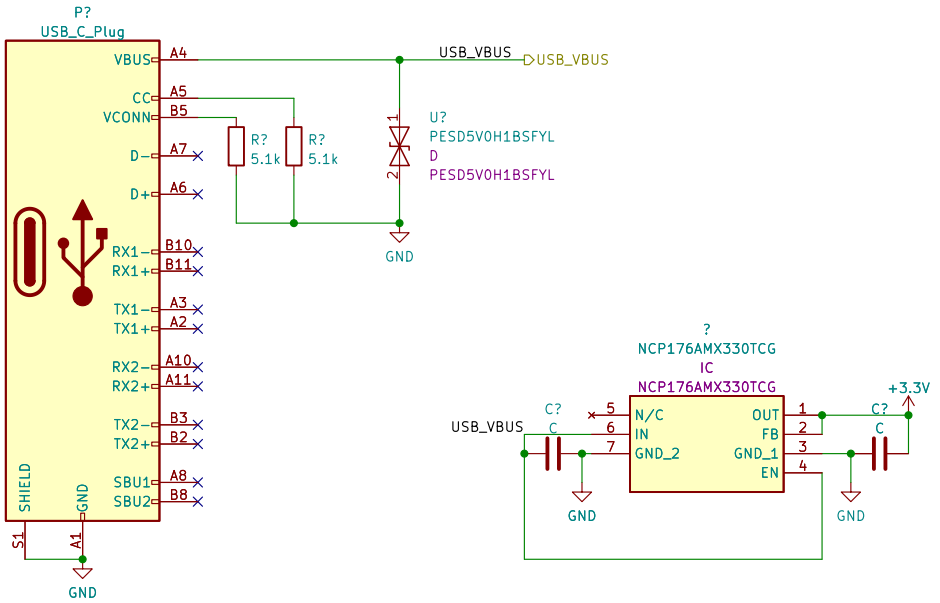


Sheet: /MPPT DC-DC/
File: MPPT.kicad_sch

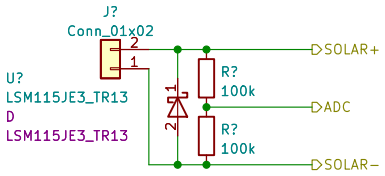
Title:

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

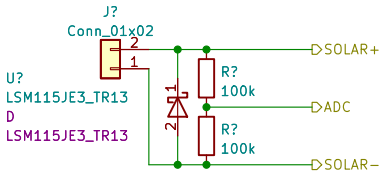
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1 | Id: 2/12



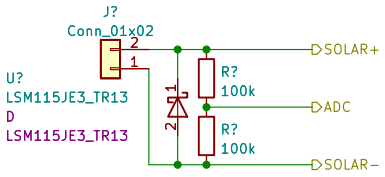
Sheet: /USB Power/		
File: USB_Power.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 3/12



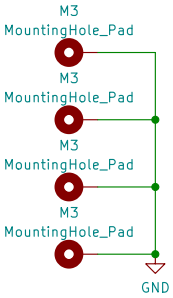
Sheet: /Solar_Connector1/		
File: Solar_Connector.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 4/12



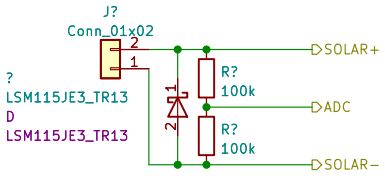
Sheet: /Solar_Connector2/		
File: Solar_Connector.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 5/12



Sheet: /Solar_Connector3/		
File: Solar_Connector.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 6/12



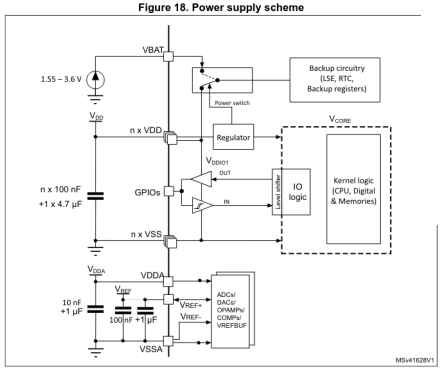
Sheet: /Mounting Holes/ File: Mounting_Holes.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 8/12



Sheet: /Solar_Connector/		
File: Solar_Connector.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 9/12

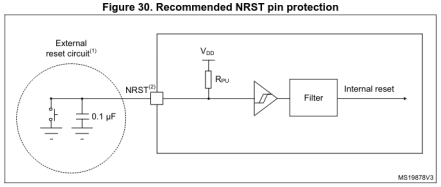
6.1.6 Power supply scheme

Figure 18. Power supply scheme

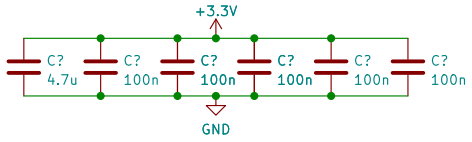


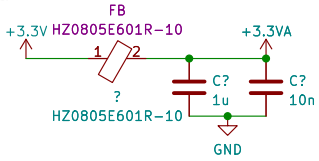
Caution: Each power supply pair (V_{DD}/V_{SS}, V_{DDA}/V_{SSA} etc.) must be decoupled with filtering ceramic capacitors as shown above. These capacitors must be placed as close as possible to, or below, the appropriate pins on the underside of the PCB to ensure the good functionality of the device.

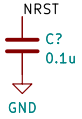
Figure 30. Recommended NRST pin protection

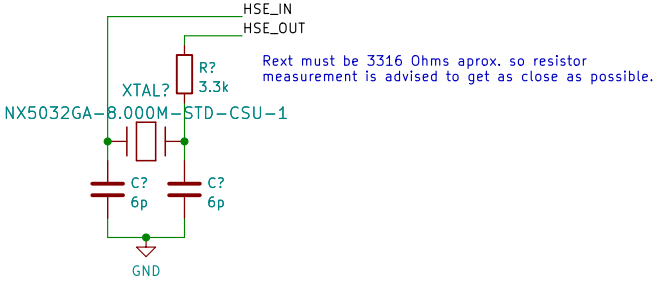


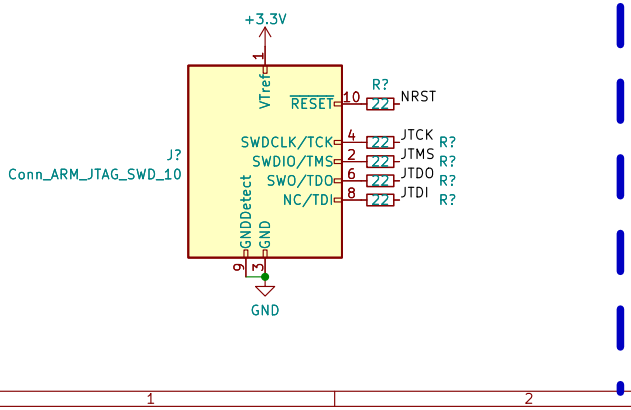
1. The reset network protects the device against parasitic resets.
2. The user must ensure that the level on the NRST pin can go below the $V_{IL(NRST)}$ max level specified in *Table 63: NRST pin characteristics*. Otherwise the reset will not be taken into account by the device.
3. The external capacitor on NRST must be placed as close as possible to the device.

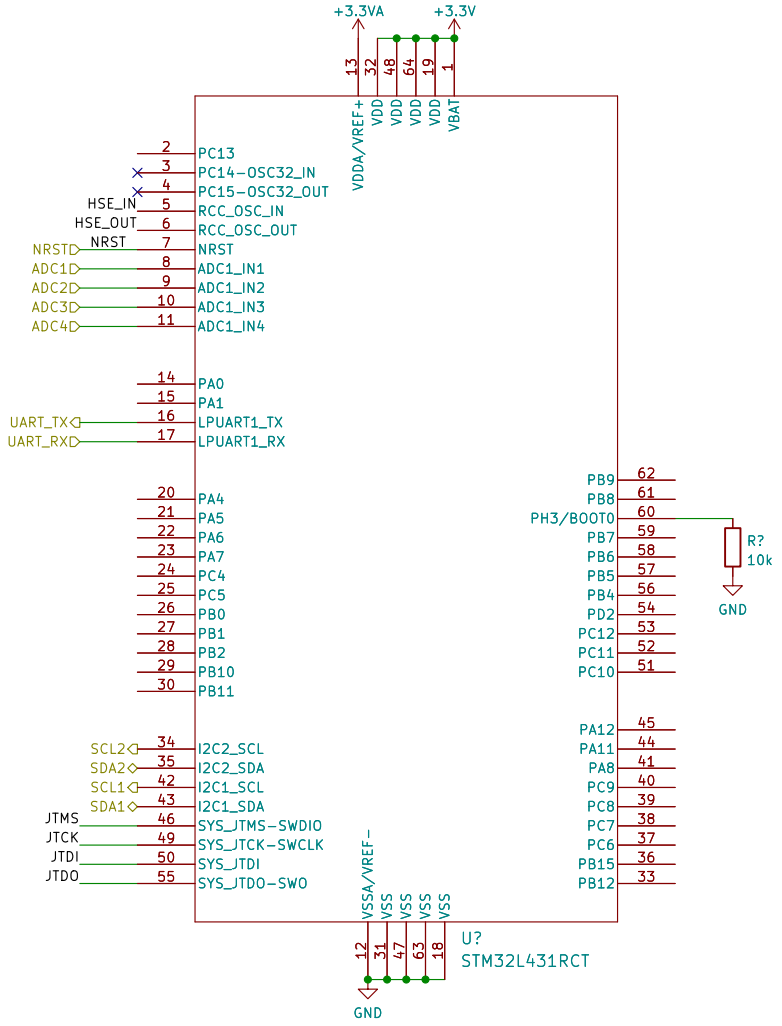




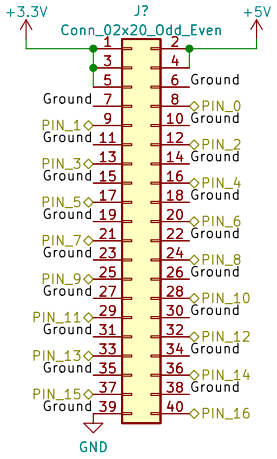




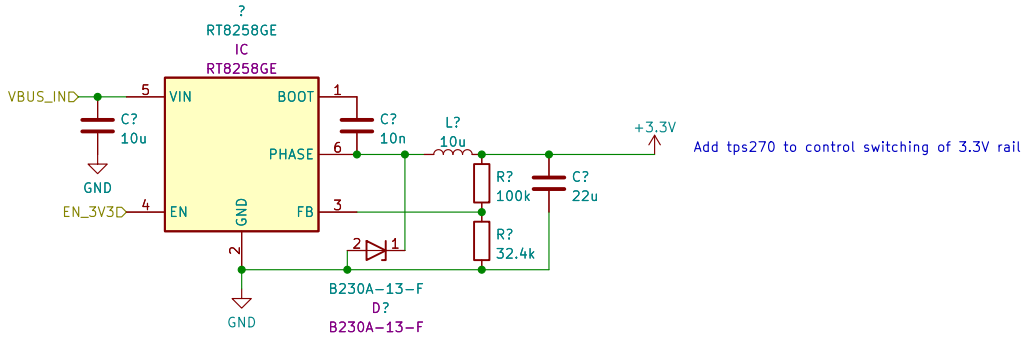




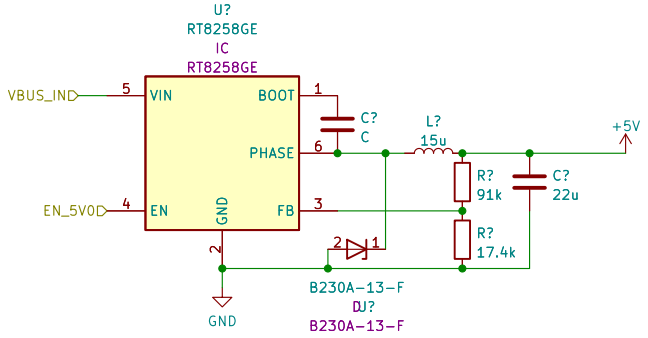
Sheet: /STM32 MCU/		
File: STM32_MCU.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 9/12



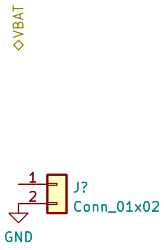
Sheet: /Board Connector/		
File: Board_Connector.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 10/12



Ground Pin. This pin should be connected to the (−) terminal of the output capacitor and it should be kept away from the D1 and input capacitor for noise prevention.



Sheet: /Power/ File: Power.kicad_sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.11−2627ca5db0~126~ubuntu20.04.1		Id: 11/12



Sheet: /Battery_Connector/
File: Battery_Connector.kicad_sch

Title:

Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 12/12