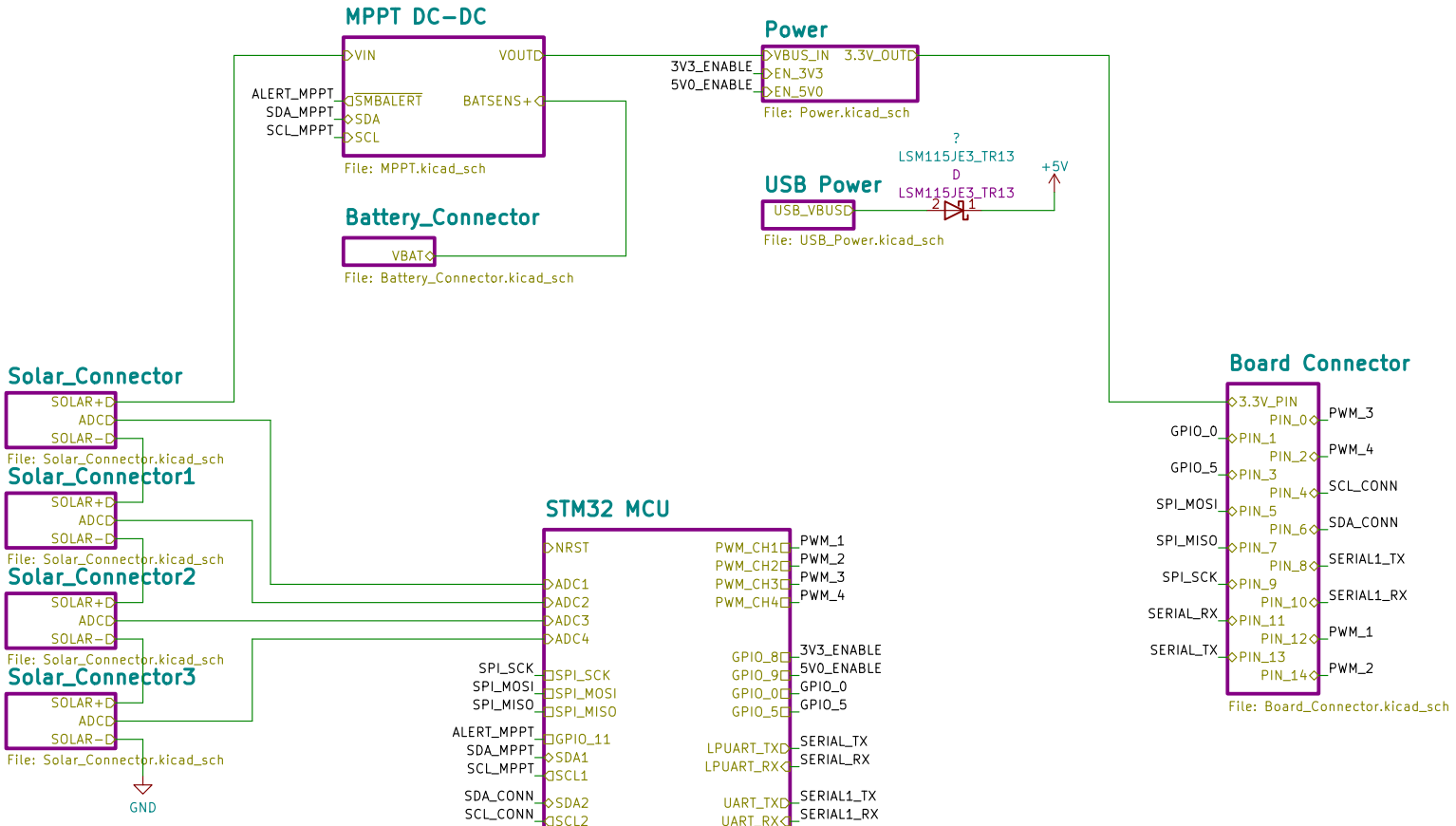
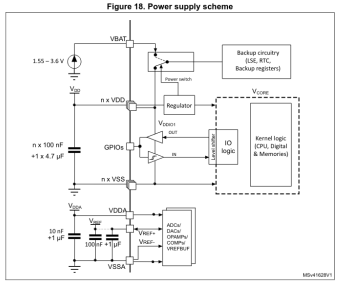

MPPT DC-DC
VIN
VOUT
ALERT_MPPT
SDA_MPPT
SCL_MPPT
SMBALERT
SDA
SCL
BATSENS+
File: MPPT.kicad_sch
Power
3V3_ENABLE
5V0_ENABLE
VBUS_IN
3.3V_OUT
EN_3V3
EN_5V0
File: Power.kicad_sch
USB Power
USB_VBUS
?
LSM115JE3_TR13
D
LSM115JE3_TR13
+5V
File: USB_Power.kicad_sch
Battery_Connector
VBAT
File: Battery_Connector.kicad_sch
Solar_Connector
SOLAR+
ADC
SOLAR-
File: Solar_Connector.kicad_sch
Solar_Connector1
File: Solar_Connector.kicad_sch
Solar_Connector2
File: Solar_Connector.kicad_sch
Solar_Connector3
File: Solar_Connector.kicad_sch
GND
STM32 MCU
NRST
ADC1
ADC2
ADC3
ADC4
SPI_SCK
SPI_MOSI
SPI_MISO
ALERT_MPPT
SDA_MPPT
SCL_MPPT
SDA_CONN
SCL_CONN
GPIO_11
SDA1
SCL1
SDA2
SCL2
PWM_CH1
PWM_CH2
PWM_CH3
PWM_CH4
PWM_1
PWM_2
PWM_3
PWM_4
GPIO_8
GPIO_9
GPIO_0
GPIO_5
3V3_ENABLE
5V0_ENABLE
GPIO_0
GPIO_5
LPUART_TXD
LPUART_RXD
SERIAL_TX
SERIAL_RX
UART_TXD
UART_RXD
SERIAL1_TX
SERIAL1_RX
File: STM32_MCU.kicad_sch
Board Connector
3.3V_PIN
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
GPIO_0
GPIO_5
SPI_MOSI
SPI_MISO
SPI_SCK
SERIAL_RX
SERIAL_TX
PWM_3
PWM_4
SCL_CONN
SDA_CONN
SERIAL1_TX
SERIAL1_RX
PWM_1
PWM_2
File: Board_Connector.kicad_sch
Mounting Holes
File: Mounting_Holes.kicad_sch
Author: Carlos Martinez Mora
e-mail:
Sheet: /
File: EPS.kicad_sch
Title: EPS Schematics
Size: A4
Date: 2023-04-08
Rev: v0.0.3
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1
Id: 1/12

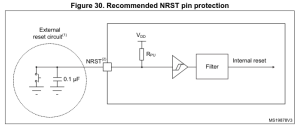
6.1.6 **Power supply scheme**

Figure 18. Power supply scheme



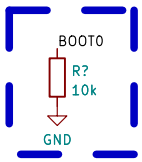
Caution: Each power supply pair (V_{DD}/V_{SS}, V_{DDA}/V_{SSA} etc.) must be decoupled with filtering ceramic capacitors as shown above. These capacitors must be placed as close as possible to, or below, the appropriate pins on the underside of the PCB to ensure the good functionality of the device.

Figure 30. Recommended NRST pin protection

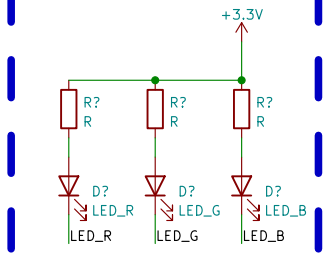


1. The reset network protects the device against parasitic resets.
2. The user must ensure that the level on the NRST pin can go below the $V_{IL(NRST)}$ max level specified in Table 63: *NRST pin characteristics*. Otherwise the reset will not be taken into account by the device.
3. The external capacitor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

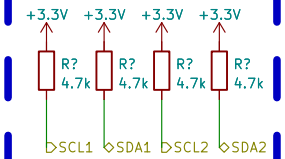
BOOT PULL-DOWN



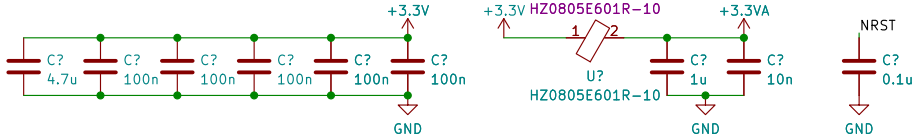
LEDS



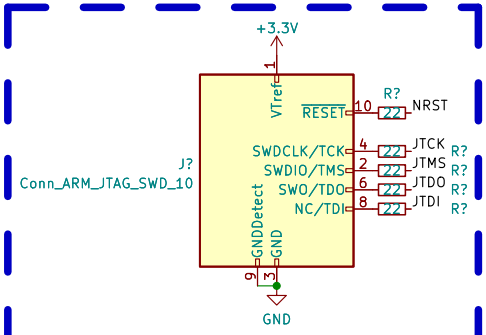
I2C PULL-UPS



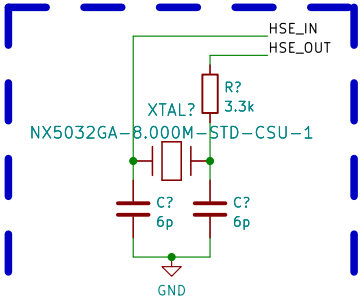
DECOUPLING



JTAG

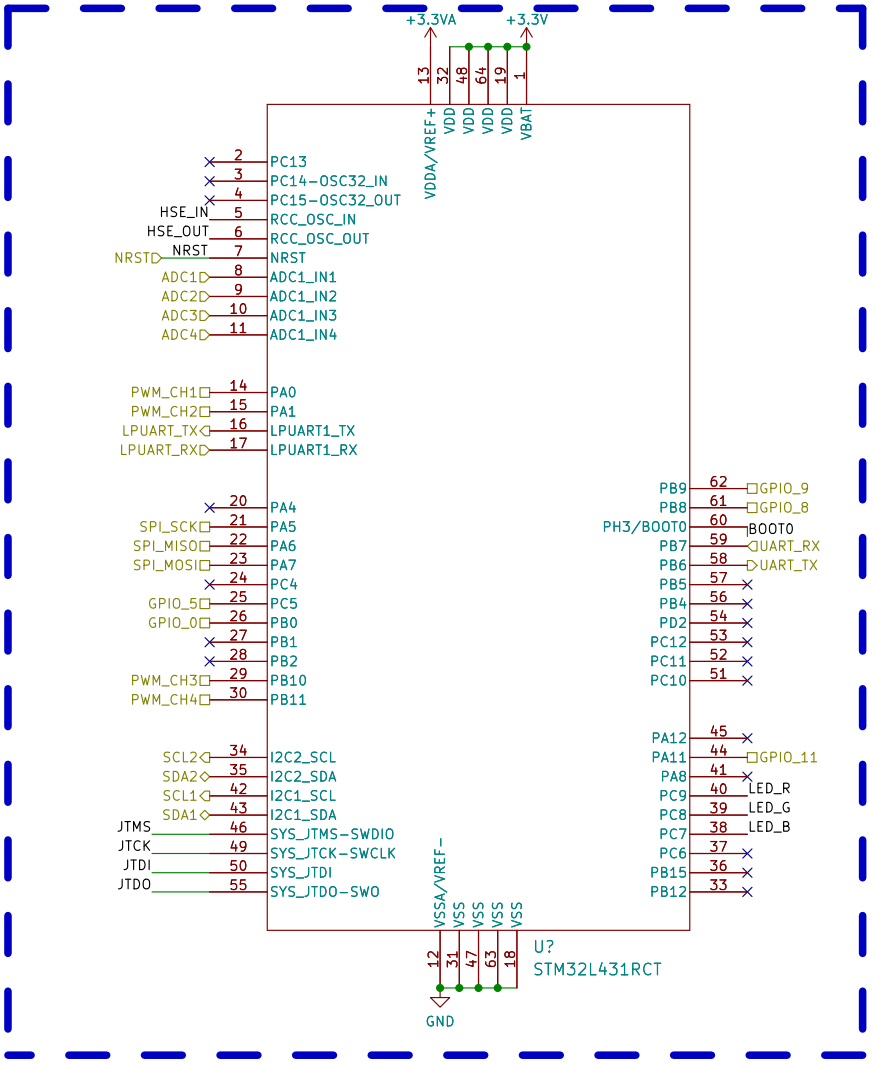


HSE CRYSTAL RESONATOR



Rext must be 3316 Ohms aprox. so resistor measurement is advised to get as close as possible.

STM32 MCU



Author: Carlos Martinez Mora
e-mail:

Sheet: /STM32 MCU/
File: STM32_MCU.kicad_sch

Title: EPS Schematics

Size: A4	Date: 2023-04-08	**Rev: v0.0.3**
KiCad E.D.A.	kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1	Id: 2/12

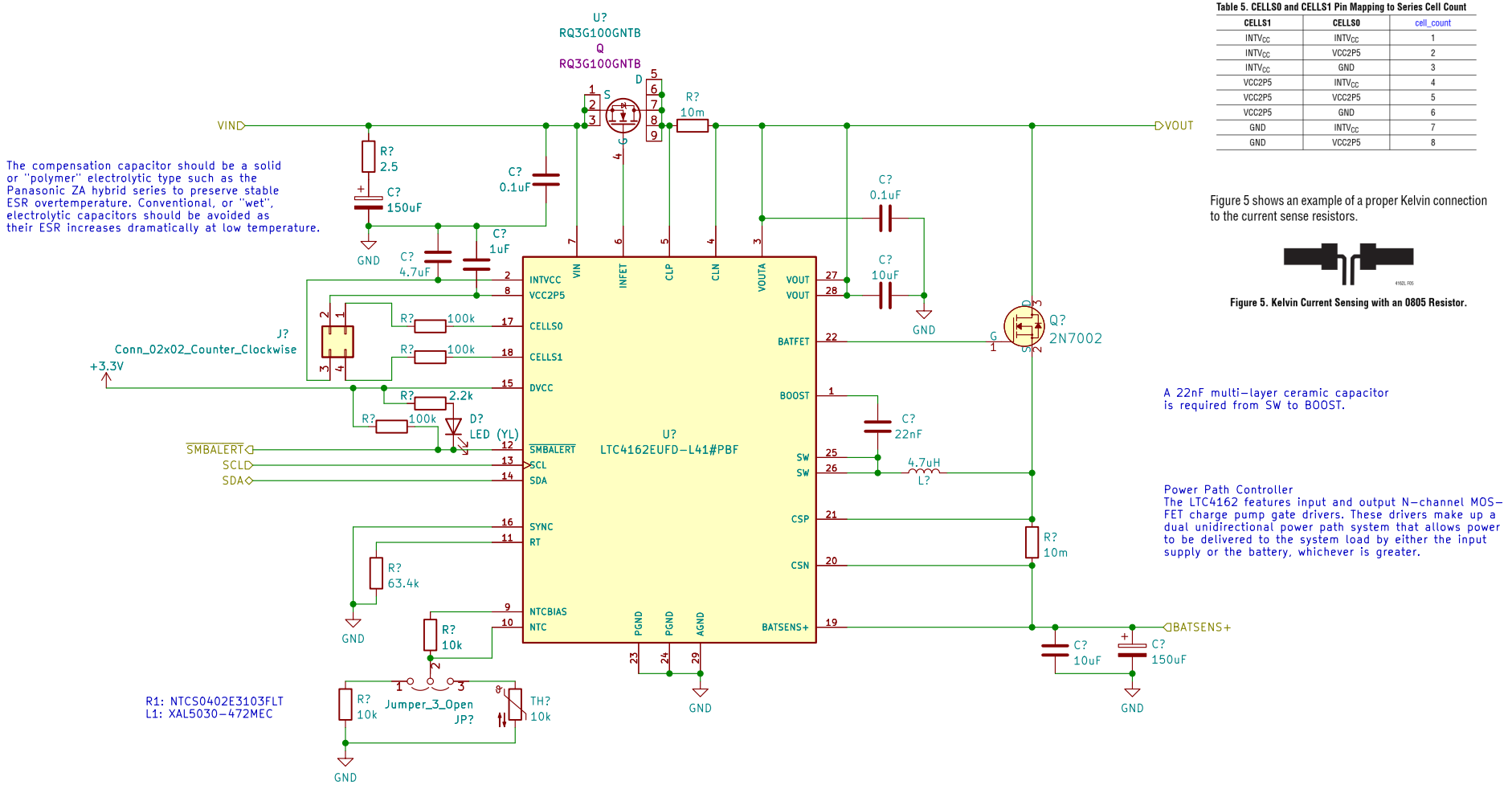


Table 5. CELLS0 and CELLS1 Pin Mapping to Series Cell Count

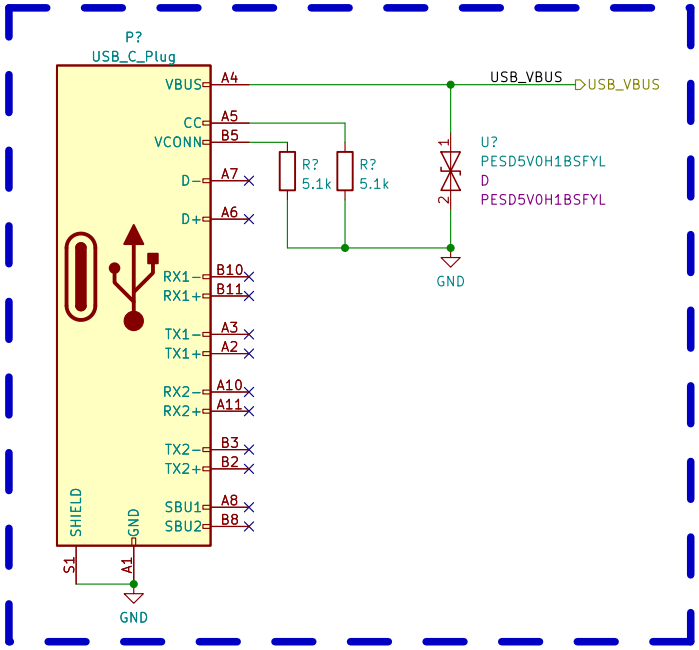
CELLS1	CELLS0	cell_count
$INTV_{CC}$	$INTV_{CC}$	1
$INTV_{CC}$	VCC2P5	2
$INTV_{CC}$	GND	3
VCC2P5	$INTV_{CC}$	4
VCC2P5	VCC2P5	5
VCC2P5	GND	6
GND	$INTV_{CC}$	7
GND	VCC2P5	8

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

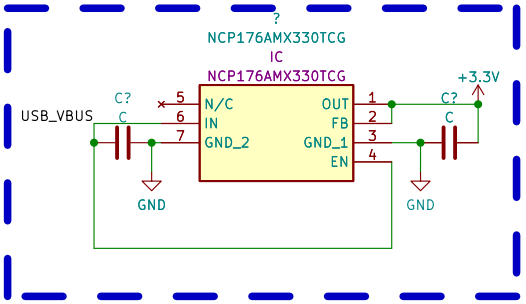
Figure 5. Kelvin Current Sensing with an 0805 Resistor.

Author: Carlos Martinez Mora e-mail:		
Sheet: /MPPT DC-DC/ File: MPPT.kicad_sch		
Title: EPS Schematics		
Size: A4	Date: 2023-04-08	**Rev: v0.0.3**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 3/12

USB CONNECTOR



3.3V LDO



Author: Carlos Martinez Mora
e-mail:

Sheet: /USB Power/
File: USB_Power.kicad_sch

Title: EPS Schematics

Size: A4	Date: 2023-04-08	**Rev: v0.0.3**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 4/12

Layout Consideration

Follow the PCB layout guidelines for optimal performance of RT8258.

- Keep the traces of the main current paths as short and wide as possible.
- Put the input capacitor as close as possible to the device pins (VIN and GND).
- PHASE node is with high frequency voltage swing and should be kept at small area. Keep sensitive components away from the PHASE node to prevent stray capacitive noise pick-up.
- Place the feedback components to the FB pin as close as possible.
- Connect the GND to a ground plane for noise reduction and thermal dissipation.

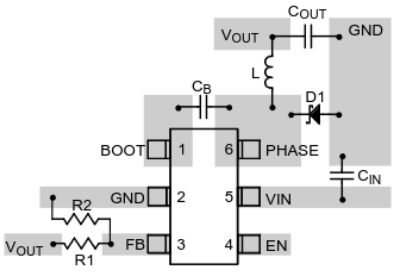
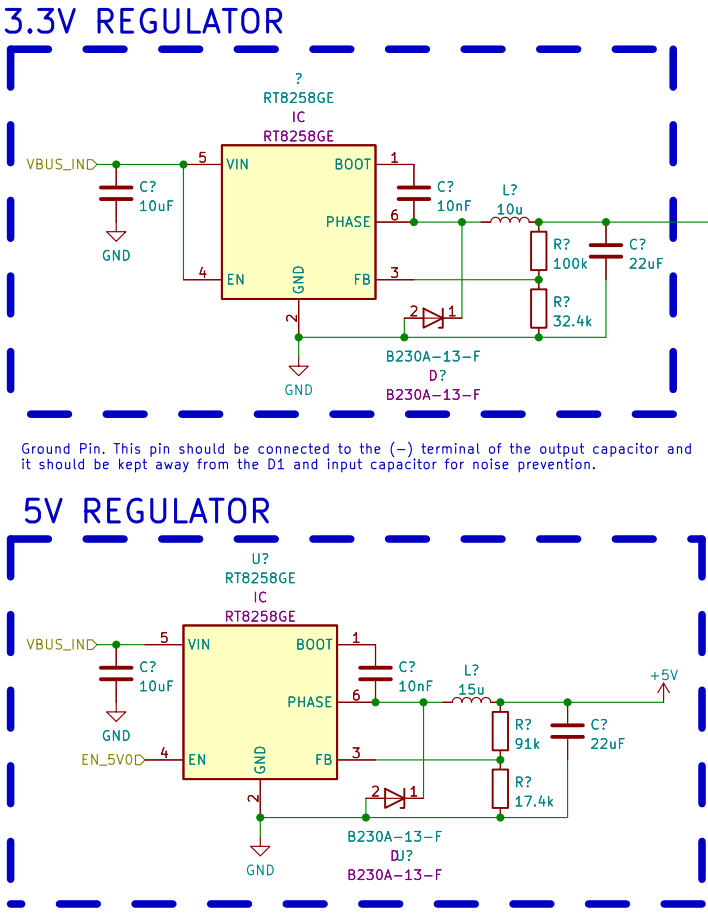


Figure 4. PCB Layout Guide



10.2 Layout Example

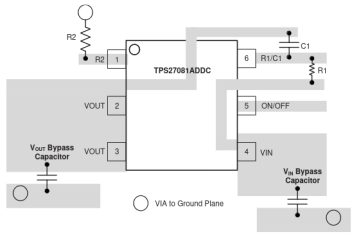
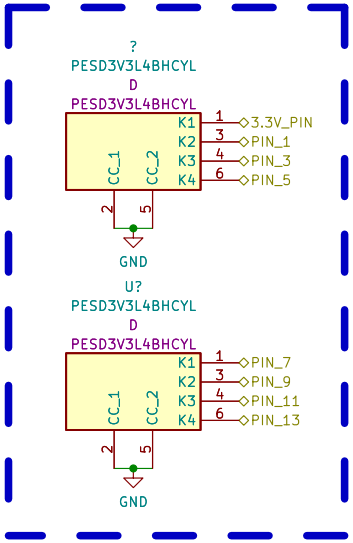


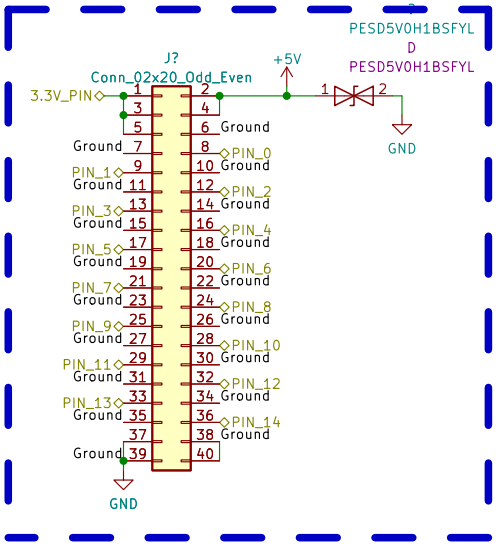
Figure 23. Layout Example

Author: Carlos Martinez Mora e-mail:		
Sheet: /Power/ File: Power.kicad_sch		
Title: EPS Schematics		
Size: A4	Date: 2023-04-08	**Rev: v0.0.3**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 5/12

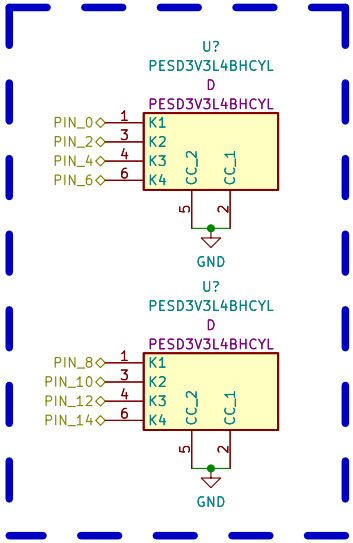
ESD 3.3V



BOARD CONNECTOR



ESD 3.3V

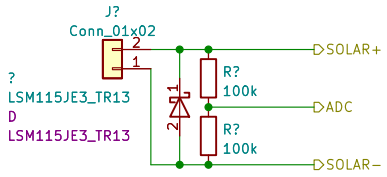


Author: Carlos Martinez Mora
e-mail:

Sheet: /Board Connector/
File: Board_Connector.kicad_sch

Title: EPS Schematics

Size: A4	Date: 2023-04-08	**Rev: v0.0.3**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 6/12

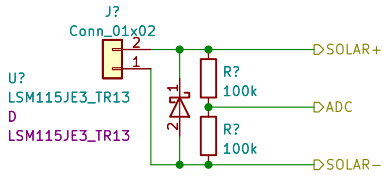


Author: Carlos Martinez Mora
e-mail:

Sheet: /Solar_Connector/
File: Solar_Connector.kicad_sch

Title: EPS Schematics

Size: A4	Date: 2023-04-08	**Rev: v0.0.3**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 7/12

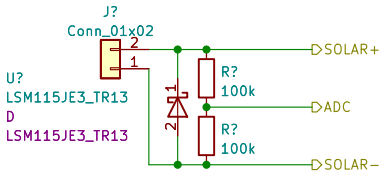


Author: Carlos Martinez Mora
e-mail:

Sheet: /Solar_Connector1/
File: Solar_Connector.kicad_sch

Title: EPS Schematics

Size: A4	Date: 2023-04-08	**Rev: v0.0.3**
KiCad E.D.A.	kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1	Id: 8/12

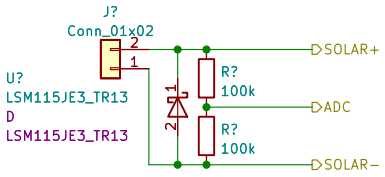


Author: Carlos Martinez Mora
e-mail:

Sheet: /Solar_Connector2/
File: Solar_Connector.kicad_sch

Title: EPS Schematics

Size: A4	Date: 2023-04-08	**Rev: v0.0.3**
KiCad E.D.A.	kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1	Id: 9/12

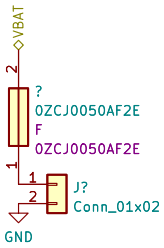


Author: Carlos Martinez Mora
e-mail:

Sheet: /Solar_Connector3/
File: Solar_Connector.kicad_sch

Title: EPS Schematics

Size: A4	Date: 2023-04-08	**Rev: v0.0.3**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 10/12

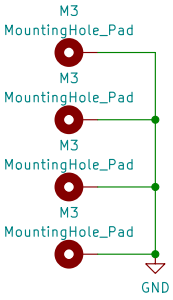


Author: Carlos Martinez Mora
e-mail:

Sheet: /Battery_Connector/
File: Battery_Connector.kicad_sch

Title: EPS Schematics

Size: A4	Date: 2023-04-08	**Rev: v0.0.3**
KiCad E.D.A.	kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1	Id: 11/12



Author: Carlos Martinez Mora
e-mail:

Sheet: /Mounting Holes/
File: Mounting_Holes.kicad_sch

Title: EPS Schematics

Size: A4	Date: 2023-04-08	**Rev: v0.0.3**
KiCad E.D.A. kicad 6.0.11-2627ca5db0~126~ubuntu20.04.1		Id: 12/12