

The diagram shows a USB to UART bridge circuit. A USB connector (J2) is labeled "USB_B_Micro". The pins are connected as follows:

- Pin 1 (VBUS) is connected to the VBUS pin of the CP2104 (pin 1).
- Pin 3 (D+) is connected to the D+ pin of the CP2104 (pin 3).
- Pin 2 (D-) is connected to the D- pin of the CP2104 (pin 2).
- Pin 4 (ID) is connected to the ID pin of the CP2104 (pin 4).
- Pin 5 (GND) is connected to the GND pin of the CP2104 (pin 5).
- Pin 6 (Shield) is connected to the Shield pin of the CP2104 (pin 6).

The CP2104 is a USB to UART bridge IC. The UART pins are connected to a UART module (D1, SP0503BAHT) as follows:

- UART VBUS is connected to the VBUS pin of the CP2104 (pin 1).
- UART_D+ is connected to the D+ pin of the CP2104 (pin 3).
- UART_D- is connected to the D- pin of the CP2104 (pin 2).
- GND is connected to the GND pin of the CP2104 (pin 5).

The CP2104 is a USB to UART bridge IC. The UART pins are connected to a UART module (D1, SP0503BAHT) as follows:

- UART VBUS is connected to the VBUS pin of the CP2104 (pin 1).
- UART_D+ is connected to the D+ pin of the CP2104 (pin 3).
- UART_D- is connected to the D- pin of the CP2104 (pin 2).
- GND is connected to the GND pin of the CP2104 (pin 5).



Direct USB

B

J1
USB_B_Micro

Shield

VBUS

D+ D-

D+ D-

ID

GND

5

6

1

2

3

4

5

6

USB_D-

USB_D+

USB_VBUS

C6 100pF

D2

GND

GND

U1 USBLC6-2P6

ESD07P06-090VMS

No external resistors needed for USB peripheral for G4

ESD is according to datasheet for USB FS

J1
USB_B_Micro

1 VBUS
2 D-
3 D+
4 ID
5 Shield/GND
6 GND

5 6
4 3
2 1

U1
USBLC6-2P6

6 USB_D-
4 USB_D+
3 D-
2 D+
1

C6
100pF

D2
ESDA7P60-1UM

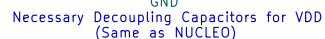
GND

ESD is according to
datasheet for USB FS

No external resistors needed
for USB peripheral for G4



Necessary Decoupling Capacitors for VDD
(Same as NUCLEO)



MCU 3V3

L1
600R@100MHz

MCU_VDD3

C3
100nF

C5
1uF

MCU_VDDA

GND

NOTE: VBAT MAY NEED A DECOUPLING CAP



5V to 3V3

Same setup used on NUCLEO G4 Board

The diagram shows a 5V to 3V3 voltage converter circuit. It features an LD3985M33R_SOT23 voltage regulator (U2). The input (VIN, pin 1) is connected to a 5V source (USB_VBUS) through a switch (SW1) and a 100nF capacitor (C14). The output (VOUT, pin 5) is connected to a 3V3 source (MCU 3V3) through a 10nF capacitor (C15). The ground (GND, pin 2) is connected to a common ground. A 1uF capacitor (C10) is connected between the input and ground, and another 1uF capacitor (C16) is connected between the output and ground. The switch (SW1) is labeled SW_SPDT and has two positions: one connecting the 5V source to the regulator input, and another connecting the 3V3 source to the regulator input.

Need some kind of diode?
NUCLEO has



U3
CP2104

UART VBUS 8 VBUS
UART_D- 4 D-
UART_D+ 3 D+

TXD 21
RXD 20
RTS 19
CTS 18
DSR 22
DTR 23
DCD 24
RT 1

SUSPEND 17
SUSPEND 15

TXT/GPIO.0 14
RXT/GPIO.1 13
RS485/GPIO.2 12
GPIO.3 11

VPP 16

C9 0.1uF
C11 1uF
C13 4.7uF

81 4.7k

RST 9
REGIN 7
VDD 6
VIO 5

GND

GND

GND

Finds DSR DTR Circuits & Implement RST once VIO & VDD are set



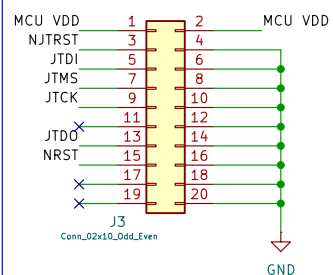
NRST

CR1
ESDALC6V1-1U2

C12
100nF

SW2
SW_Push

GND



The schematic diagram illustrates the STM32 G4 LQFP64 board, highlighting the ST-LINK/V2 connector (CN3) and the BOOT0 pin configuration. The connector is a 20-pin header with the following connections:

- MCU_VDD (Pin 1) to MCU_VDD (Pin 2)
- NJTRST (Pin 3) to GND (Pin 4)
- JTDI (Pin 5) to GND (Pin 6)
- JTMS (Pin 7) to GND (Pin 8)
- JTCK (Pin 9) to GND (Pin 10)
- JTDO (Pin 11) to GND (Pin 12)
- NRST (Pin 13) to GND (Pin 14)
- NRST (Pin 15) to GND (Pin 16)
- NRST (Pin 17) to GND (Pin 18)
- NRST (Pin 19) to GND (Pin 20)

The BOOT0 pin (Pin 61) is connected to GND via a 10kΩ pull-down resistor (R10). The BOOT0 pin is also connected to the BOOT0 pin of the STM32G474RETx microcontroller (U4) via a 10kΩ pull-down resistor (R10). The BOOT0 pin is also connected to the BOOT0 pin of the STM32G474RETx microcontroller (U4) via a 10kΩ pull-down resistor (R10).

