

The schematic diagram illustrates the power supply section of the PCB. It features a Mini_USB connector (U1) with pins VCC (1), D- (2), D+ (3), ID (4), GND (5), and SHELL (6). The VCC pin is connected to a 5V regulator (U2, LM1117-3.3) through a 0.1uF/50V capacitor (C2). The regulator's output (Vout) is connected to a red LED (LED1) through a 470Ω resistor (R1). The LED is labeled "LED1 RED". A JUMP jumper (J1) is also shown. The ground connection is labeled "GND".

The diagram illustrates the pin connections for the STM32F103C8T6 microcontroller. It shows two headers, P1 and P2, with their respective pin numbers and the signals connected to them.

Header	Pin	Signal
P1	1	MCU VDD
	2	MCU VDD
	3	MCU VDD
	4	PA1
	5	PA0
	6	PA2
	7	PA3
	8	PA4
	9	PA5
	10	PA6
	11	PA7
	12	PA12
	13	MCU VDD
	14	MCU VDD
	15	MCU VDD
P2	1	GND
	2	GND
	3	GND
	4	MCU RST
	5	OSCOUT
	6	OSCIN
	7	GND
	8	Boot0
	9	PB7
	10	PB6
	11	PB5
	12	SWCLK
	13	SWDIO
	14	GND
	15	GND

The diagram shows a push button switch labeled SW1. One terminal of the switch is connected to a common ground symbol. The other terminal is connected to a microcontroller pin labeled PA12. A resistor labeled R3 with a value of 1.5K is connected between the switch terminal and the PA12 pin. The PA12 pin is also connected to a supply voltage labeled MCU VDD.

PA2	1	PA2	PA1	20	PA1
PA3	2	PA3	PA0	19	PA0
PA4	3	PA4	PF2-NRST	18	MCU_RST
PA5	4	PA5	PF1-OSCCOUT	17	OSCCOUT
PA6	5	PA6	PF0-OSCCIN	16	OSCCIN
PA7	6	PA7	PF4-BOOT0	15	Boot0
GND	7	VSS	PB7	14	PB7
PA12	8	PA12	PB6	13	PB6
MCU_VDD	9	VCC	PB5	12	PB5
SWDIO	10	PA13	PA14	11	SWCLK

U3
PY32F030

Pin	Signal
1	MCU RST
2	SWCLK
3	GND
4	SWDIO
5	MCU VDD

Capacitor C9: 0.1uF/50V

Ground connection: GND

Header label: P4

Header pins: 1, 2, 3, 4, 5

Label: Debug

File:	C:\Users\...\PY32F003 StartKit V1.0.SchDoc	Drawn By:
-------	--	-----------