

# FTDI Programmer/UART/FIFO

**3V3 VReg Device**

Enable Delay  
~0.001s  
Should enable  
after 1v2 rail  
(test!!!!)

**1V2 VReg Device**

The diagram shows a 1V2 VReg Device (U3) with the following connections:

- Pin 1 (IN):** Connected to a +5V supply.
- Pin 2 (OUT):** Connected to a +1V2 output.
- Pin 3 (EN):** Connected to GND.
- Pin 4 (BYP):** Connected to GND.
- Pin 5 (GND):** Connected to GND.

Bypass capacitors are connected as follows:

- C8 (4u7):** Connected between the +5V supply and GND.
- C11 (10n):** Connected between the IN pin and GND.
- C15 (4u7):** Connected between the +1V2 output and GND.

[illegible][illegible][illegible]

SAO/I2C "Ear"

EB\_OD2 1 J33 2 EB\_OD1  
EB\_1V2 3 4 EB\_OD3  
EB\_GND 5 6 EB\_GND  
EB\_5V 7 8 EB\_3V3

RGB Led

EB\_3V3 1 J34 2 EB\_OD1  
EB\_GND 3 4 EB\_GND  
SADD 5 6 EB\_3V3

R28 10k R29 10k

Unpopulated

### Dual PMOD

The diagram illustrates a Dual PMOD circuit. It features two PMOD modules, J21 and J22, connected to a 5V AUX power source. J21 is connected to +3V3 and GND, while J22 is connected to +5V and GND. Both modules have 12 pins, with pins 1-4 and 5-8 connected to resistors (R20A-R20D and R23A-R23D for J21; R21A-R21D and R24A-R24D for J22). Pins 9-12 are connected to a 33E resistor and a DNP capacitor (C32 for J21, C33 for J22).

The diagram illustrates the wiring for the Single PMOD module. It features two modules, J23 and J27, connected to a power supply. J23 is a 12-pin module with pins 1-6 connected to +3V3 and pins 7-12 connected to GND. J27 is a 5V AUX module with pin 1 connected to +5V and pin 2 connected to GND. The diagram also shows various resistors (R4A, R4C, R22A, R22C, R4B, R4D, R22B, R22D) and a capacitor (C34) connected to the modules.

The diagram shows a 3x3 LED matrix and a 4x4 button grid. The LED matrix has columns labeled LEDG, LEDR, and LEDB, and rows labeled GRN, RED, and BLUE. Each LED is connected to a common +3V3 supply through a 330Ω resistor (R14, R15, R16). The button grid has columns labeled SW1, SW2, SW3, and SW4, and rows labeled BUT1, BUT2, BUT3, and BUT4. Each button is connected to a common GND through a 10kΩ resistor (R13, R17, R18, R19).

The schematic diagram illustrates the wiring for the RGB LED. The J25 connector is a central component with 8 pins. The connections are as follows:

- Pin 1: +1V2
- Pin 2: +3V3
- Pin 3: LED\_RGB1
- Pin 4: LED\_RGB2
- Pin 5: LED\_RGB0
- Pin 6: GND
- Pin 7: +5V
- Pin 8: +3V3

The J30 connector is also shown, with the following connections:

- Pin 1: +5V
- Pin 2: +3V3
- Pin 3: GND

The D11 component is shown with the following connections:

- Pin 1: LED\_RGB2
- Pin 2: LED\_RGB1
- Pin 3: LED\_RGB0

The D11 component is labeled "Unpopulated".

Sheet: /		
File: icebreaker.sch		
<b>Title: iCEBreaker</b>		
Size: A3	Date:	Rev: V1.0d
KiCad E.D.A. kicad 5.1.2-f72e74a84ubuntu18.04.1		Id: 1/1