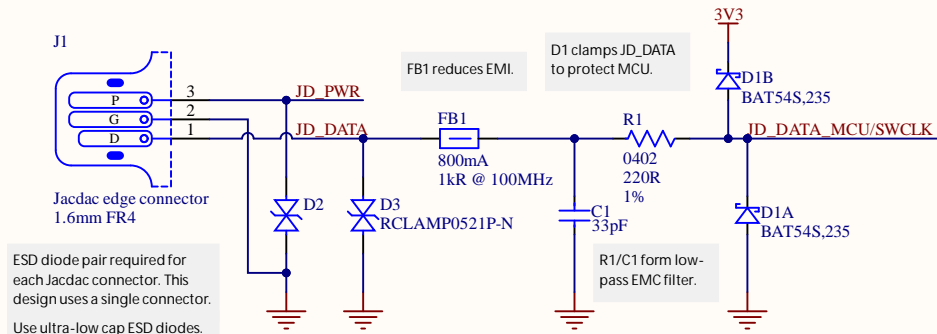
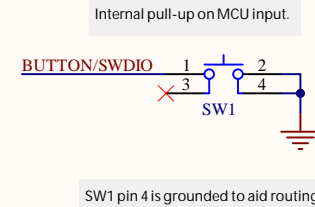


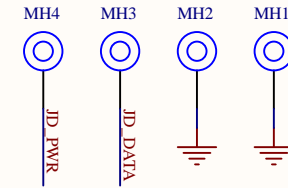
Jacdac connector



Button



Mounting holes



Mounting holes are electrically connected to the Jacdac bus nets so they can be used as an alternative to the PCB edge connector. Please use the following reference designators and net mapping:

MH1 & MH2: GND
MH3: JD_DATA
MH4: JD_PWR

This design uses Jacdac 'small' mounting holes: PTH with finished diameter of 2.1mm, annular copper ring of 3.0mm diameter & copper/component keepout of 5.0mm. The mounting holes must be on 2.5mm pitch. Mounting holes should have appropriate silkscreen marker, and MH1 should have a pin 1 marker on the top side.

This reference design is a guideline. Please refer to the Jacdac docs online at <https://aka.ms/jacdac> for the definitive and most up-to-date information.

This design uses a 'cute' board shape.

Silkscreen should include text to identify the module type and revision, and optionally a QR code.

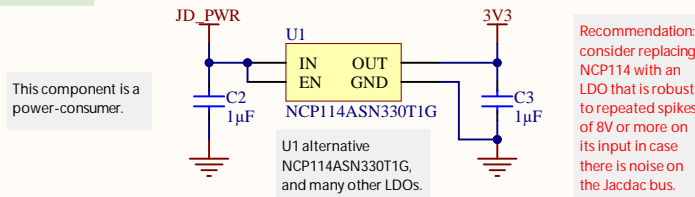
Silkscreen & layout notes

Block name

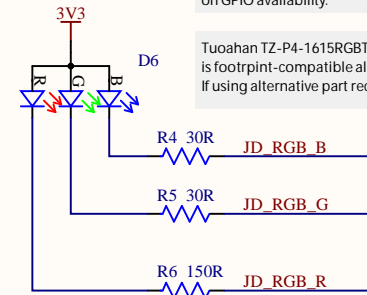
Design notes

When this PDF is viewed with Adobe Reader, clicking on components shows part numbers and other details.

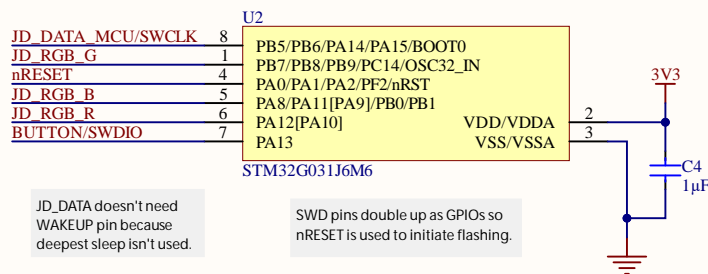
3V3 regulator



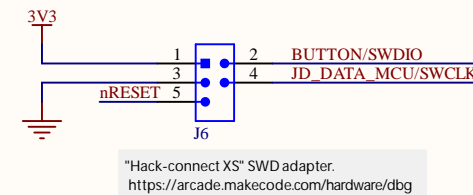
LED



MCU



Programming/debug connector



Microsoft

PROJECT DESCRIPTION
Jacdac button based on STM32 G0 Series MCU

SHEET DESCRIPTION
Complete design

PROJECT FILENAME JacdacG0Button 40.PrjPCB

PROJECT CODENAME JacdacG0Button

LAST MODIFIED 03/12/2021

PAGE 1 OF 1

DRAWN BY D. Gakure & S. Hodges

REVISION 0.3

PCB ID 40-0.3

SHEET FILENAME JacdacG0Button 40.SchDoc

LICENCE Attribution 4.0 International (CC BY 4.0)