ELECTRICAL ENGINEERING

STM32 custom PCB: microcontroller unresponsive to the debugger, draws too much current, becomes warm

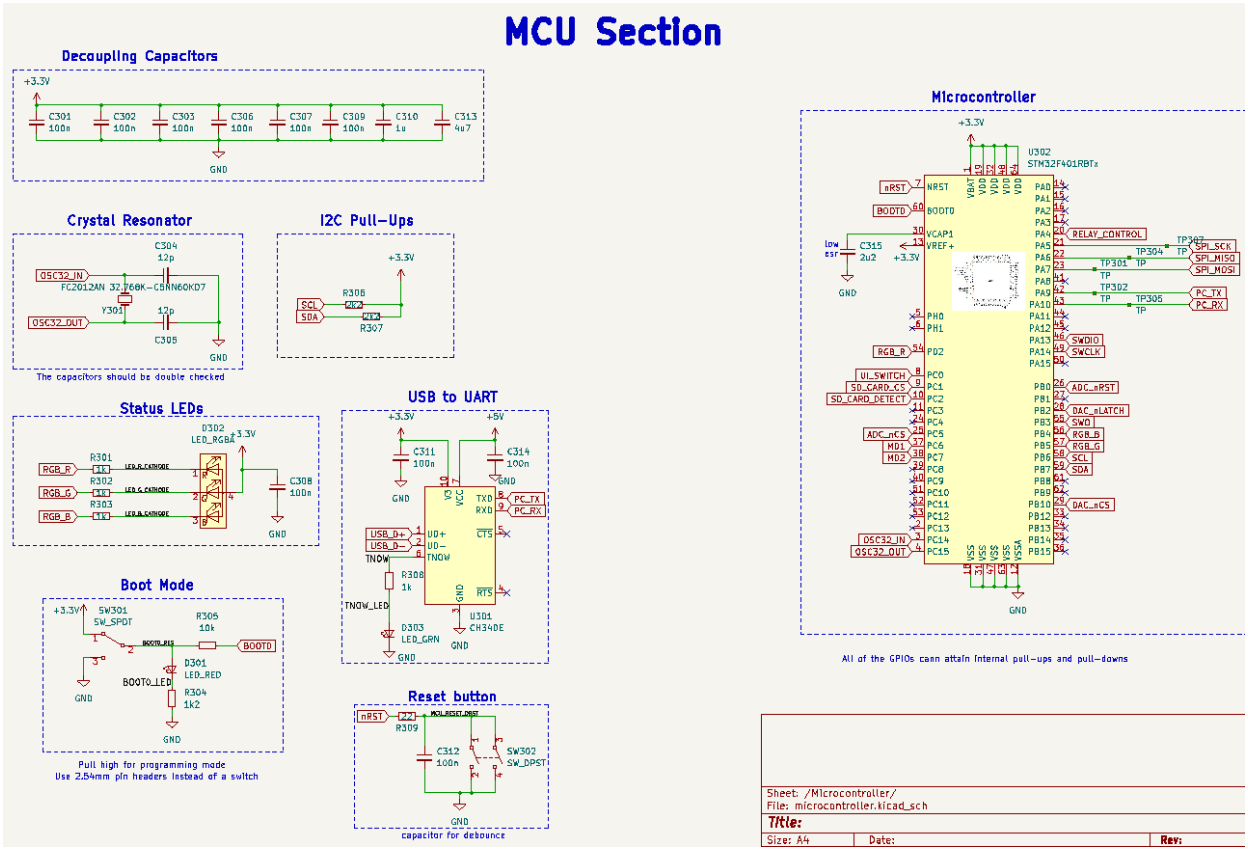
Asked 1 year, 2 months ago Modified 1 year, 2 months ago Viewed 311 times

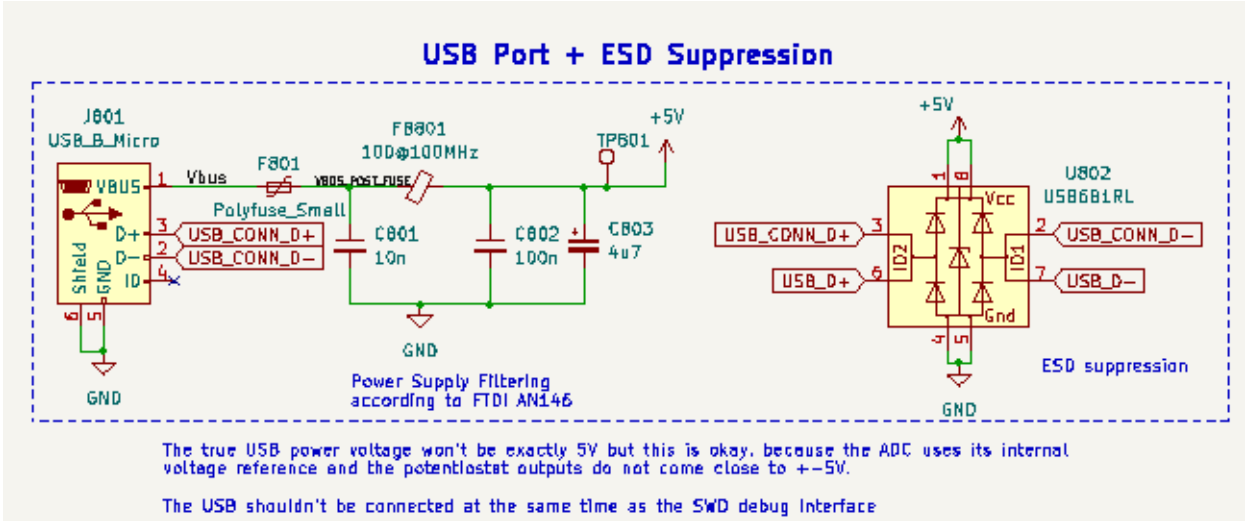
5

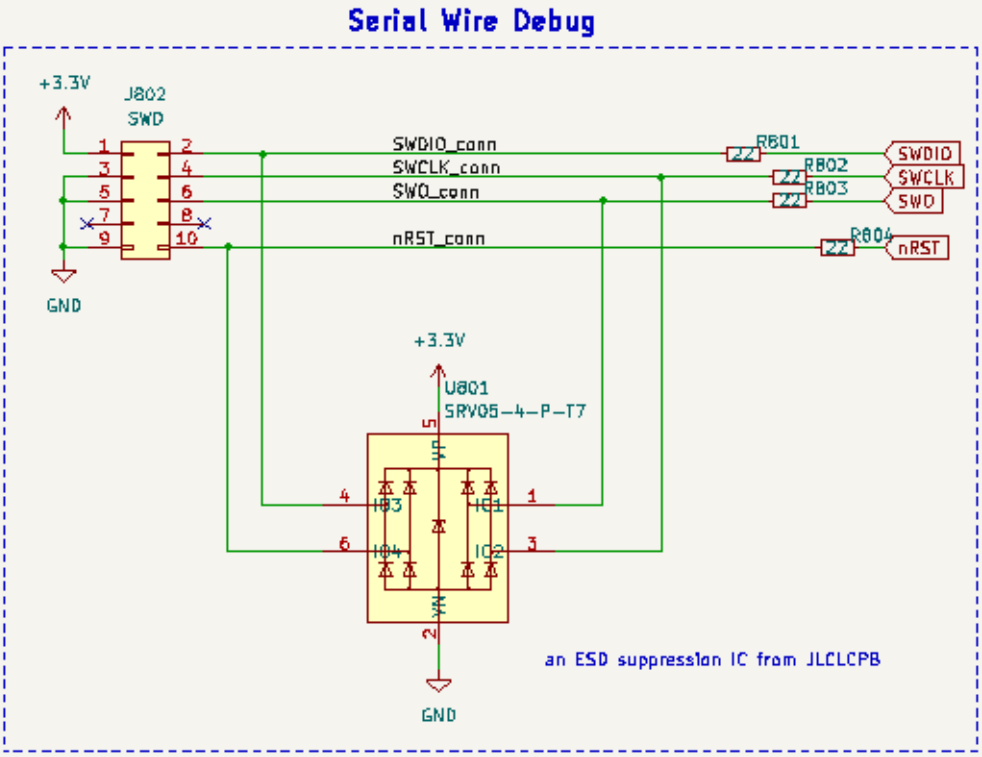
For a school project I've designed a PCB on KiCAD around STM32F401RBT6. The design is based on what I designed before and the application notes. It is powered through the USB port and should be programmable via SWD but it is not. I am using an ST-Link V2 and the STM32Programmer application to communicate with the microcontroller but it is not found by the application. The PCB was ordered from JLCPCB with the passive components already assembled. I am hand-soldering the microcontroller by the method of drag-soldering (320C soldering iron temperature). I clean off the excess solder using flux and a solder wick. As soon as I solder the microcontroller on the PCB and power it up, the power supply shows me that the current draw is ~160mA. I verified by observing the PCB with a thermal camera and by desoldering the microcontroller that it certainly is the microcontroller that is dissipating all the heat. The interesting thing is, I tried tweaking all sorts of things and it is always the same symptom and almost exactly the same amount of current (within +-5mA). Things I've tried:

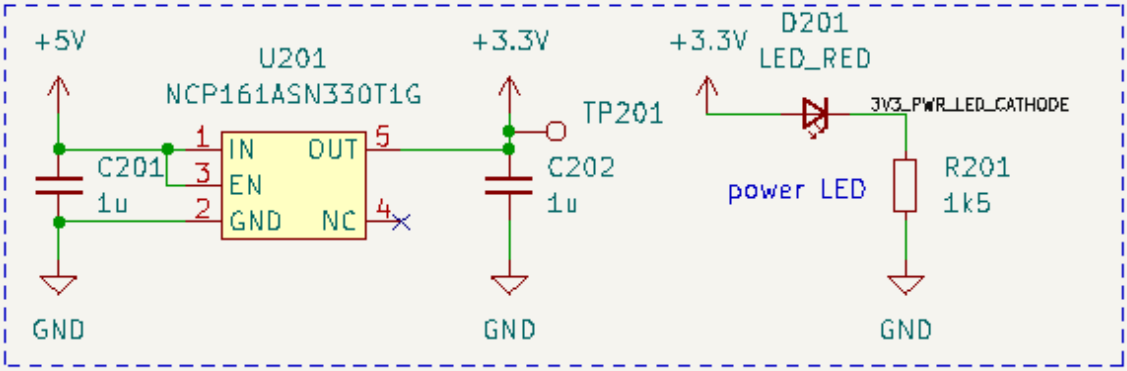
- Leaving all irrelevant components unassembled, leaving only the SWD header and the LDO (5V to 3V3)
- Trying out pristine microcontrollers (I have already fried 9 of them). This leads me to believe that it is not a short circuit problem.
- Cleaning the solder flux thoroughly with isopropyl alcohol and a toothbrush
- Looking at the connections under the microscope and verifying that there's no short circuit (although I am aware that there might be a short-circuit but it always being the same current draw and that the microcontroller package gets warm, I conclude that the error is within)
- Pullup resistors on SWIO and SWCLK
- Checking the orientation
- the debugger works, I see the pulses on the oscilloscope
- there's no overvoltage since I am powering the board with an LDO. I've also observed the 3V3 rail during power-up and it seems to not jump up exceeding 3V3.
- replacing the ESD protection IC at SWD
- NRST normally high, and is asserted by the debugger
- BOOT0 is not floating.
- KiCAD DRC check
- the pinout is correct (I compared with the datasheet and STM32CubeMX)

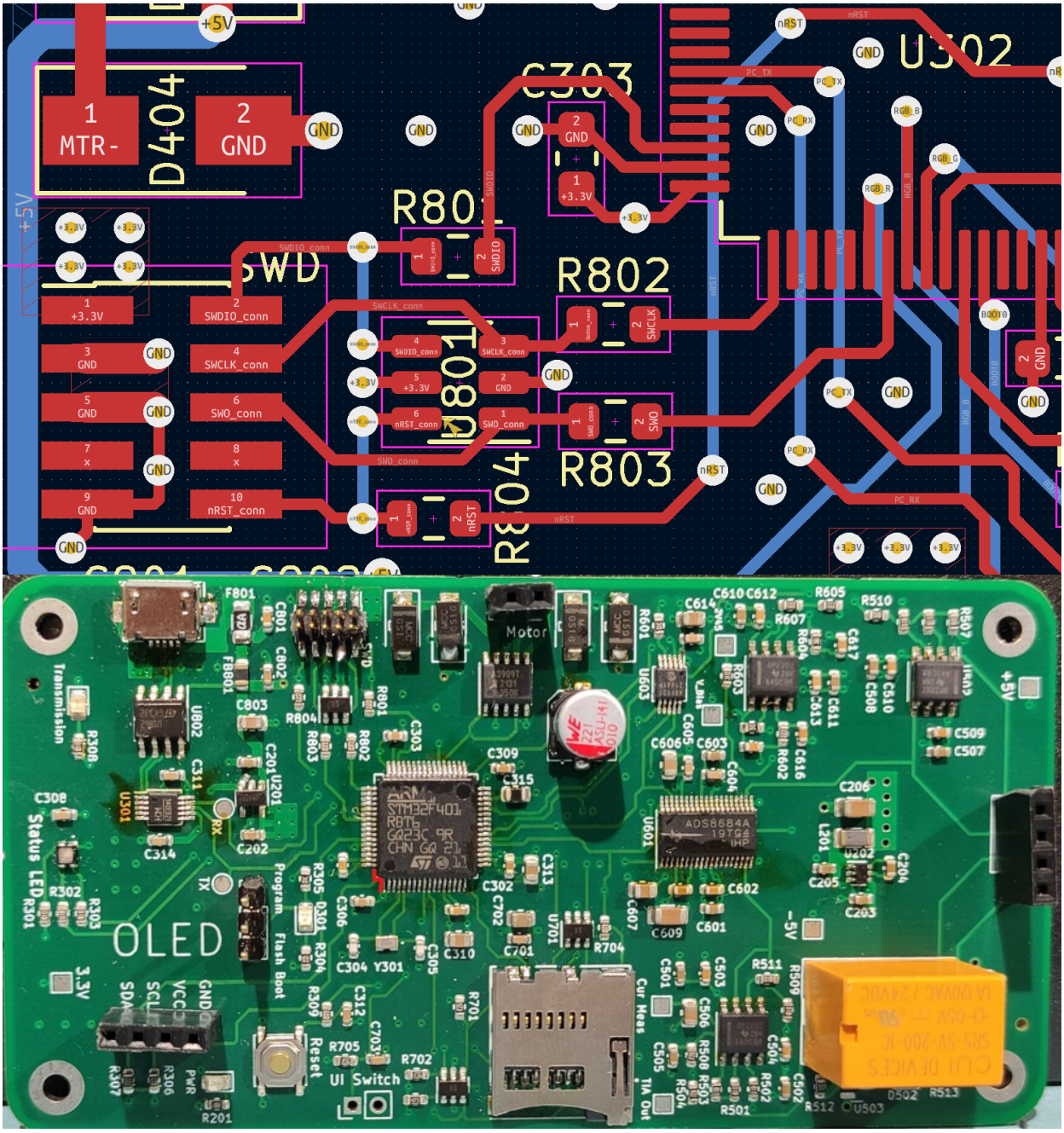
- Ramping the current up slowly
- All VDD and **VDDA** caps are according to what is advised in the application note. So is VCAP
- I am using this adapter (https://www.olimex.com/Products/ARM/JTAG/ARM-JTAG-20-10/) I am sure that the pinout is correct
- The voltage at the 3V3 rail seems to be correct


MCU Section
Decoupling Capacitors
Crystal Resonator
I2C Pull-Ups
Microcontroller
Status LEDs
USB to UART
Boot Mode
Reset button
The capacitors should be double checked
Pull high for programming mode
Use 2.54mm pin headers instead of a switch
capacitor for debounce
All of the GPIOs cann attain internal pull-ups and pull-downs
Sheet: /Microcontroller/
File: microcontroller.kicad_sch
Title:
Size: A4
Date:
Rev:


USB Port + ESD Suppression
J801
USB_B_Micro
VBUS
D+
D-
ID
Shield
GND
Vbus
F801
Polyfuse_Small
VBUS_POST_FUSE
USB_CONN_D+
USB_CONN_D-
FB801
10Ω@100MHz
C801
10n
C802
100n
C803
4u7
+5V
TP801
GND
Power Supply Filtering
according to FTDI AN146
U802
USB6B1RL
Vcc
Gnd
IO1
IO2
USB_D+
USB_D-
ESD suppression
The true USB power voltage won't be exactly 5V but this is okay, because the ADC uses its internal voltage reference and the potentiostat outputs do not come close to +-5V.
The USB shouldn't be connected at the same time as the SWD debug interface


Serial Wire Debug
+3.3V
J802
SWD
SWDIO_conn
SWCLK_conn
SWO_conn
nRST_conn
R801
R802
R803
R804
22
SWDIO
SWCLK
SWD
nRST
GND
U801
SRV05-4-P-T7
IO1
IO2
IO3
IO4
VN
VP
an ESD suppression IC from JLCLCPB


LDO Regulator to 3V3
+5V
U201
NCP161ASN330T1G
IN
EN
GND
OUT
NC
C201
1u
C202
1u
+3.3V
TP201
D201
LED_RED
3V3_PWR_LED_CATHODE
power LED
R201
1k5
GND



I am completely stuck and out of ideas out here. Does anyone have an idea as to what is going on? Has anyone had the same problem before?

microcontroller pcb-design stm32 debugging

Share Cite Follow

edited Nov 21, 2022 at 12:11

asked Nov 13, 2022 at 9:05

Ciguli
81 5

1 Mind posting a photo of the actual board so we can double check orientation etc? Have you removed the MCU and measured what voltages are on each of the pins? The problem may not be this part of the schematic as the connections to elsewhere are unknown. – Justme Nov 13, 2022 at 9:58

1 No use saying 'you're sure' - I've proven wrong many times. What have you done to check and remove any doubt? Erase your brain and start with the datasheet. Are you referring to the pinout of the correct

package? If the footprint numbered correctly? Correct rotation of the part? Pin 1 correct? Beep the pcb out with a multimeter. Maybe use a blackpill board as a reference? – Kartman Nov 13, 2022 at 10:32

@Kartman I compared the pinout and packaging information given in the datasheet. The photo is included in the post now. Do you think the orientation is correct? I referred to AN4488. – Ciguli Nov 13, 2022 at 10:52

Is there a specific reason why you chose not to have all pins of your ADC in your schematic? Nevertheless, assuming the placement of the STM is correct, start by removing other ICs and see if the power consumption goes down. Remove parts until only the STM is left. Then you can be sure if it's the STM or some other connection that's causing you troubles. I have a feeling it might have something to do with a 5V path through your STM's ESD diodes but I don't really see an issue because most pins are 5V tolerant. – Tom L. Nov 16, 2022 at 21:19

2 Please post your edit as an answer to your own question and accept it so that this question can be closed. – winny Nov 20, 2022 at 10:49

1 Answer

Sorted by: Highest score (default)

3

UPDATE I was able to solve the issue. While measuring the voltages at the pins, I noticed that the multimeter was telling me that it is a short circuit betweens the two pads of the VCAP capacitor. Then I noticed that instead of a capacitor at VCAP, I placed an inductor. . So anyway, placing a 4u7 capacitor at VCAP solved the issue.

Share Cite Follow

answered Nov 21, 2022 at 12:11

Ciguli
81 5

1 Hi, Thanks for coming back with an answer to your question. In order to effectively mark the topic as solved, please consider "áccepting" your choice of the best answer (i.e. click the "tick mark" next to an answer - your answer or another one, if one is written - to turn the relevant tick mark green). This shows that you don't need more help and future readers can quickly see which was the confirmed solution. Thanks. – SamGibson ♦ Nov 21, 2022 at 12:26

1 It's good habit to probe the power pins of the PCB *before* you start soldering to check if there is a problem with the board. And then periodically do so as you solder in case you do something like a fry an ceramic cap (they fail short). But certainly, the first thing you should have done after the you removed a dead MCU the first time due to a short was to probed the power pins while the MCU was not there there to find out whether the issue was the MCU or something else. – DKNguyen Nov 21, 2022 at 13:17

Also, I have seen cases where the bare PCB was fine, but right after the MCU was soldered there was a short, and then after the MCU was removed the short persisted even though the pads were clean. I still don't know what the cause of this was, and it was repeatable between all ten boards. – DKNguyen Nov 21, 2022 at 13:21