

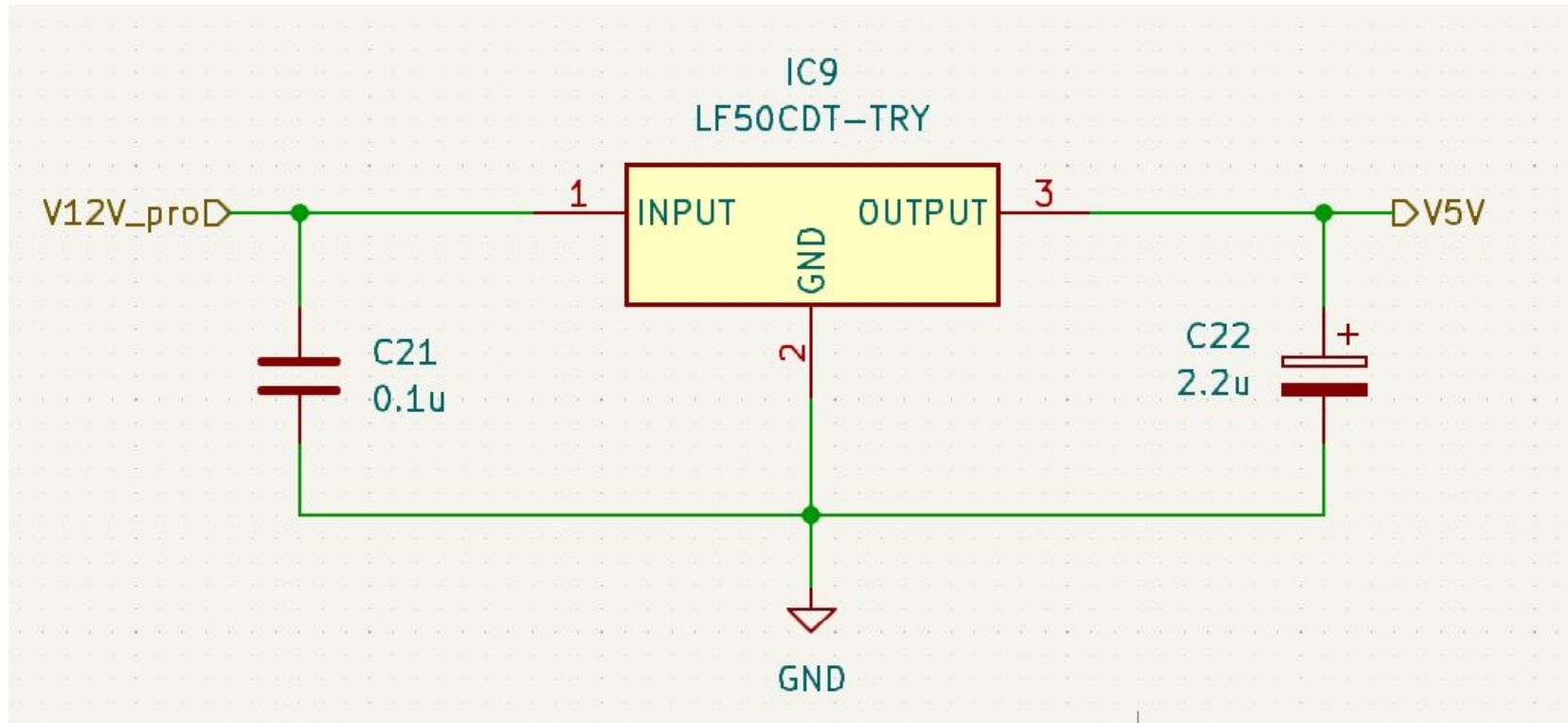
ECU SW CONTROL PROJECT

DESIGN FINDINGS

LF50CDT-TRY LDO

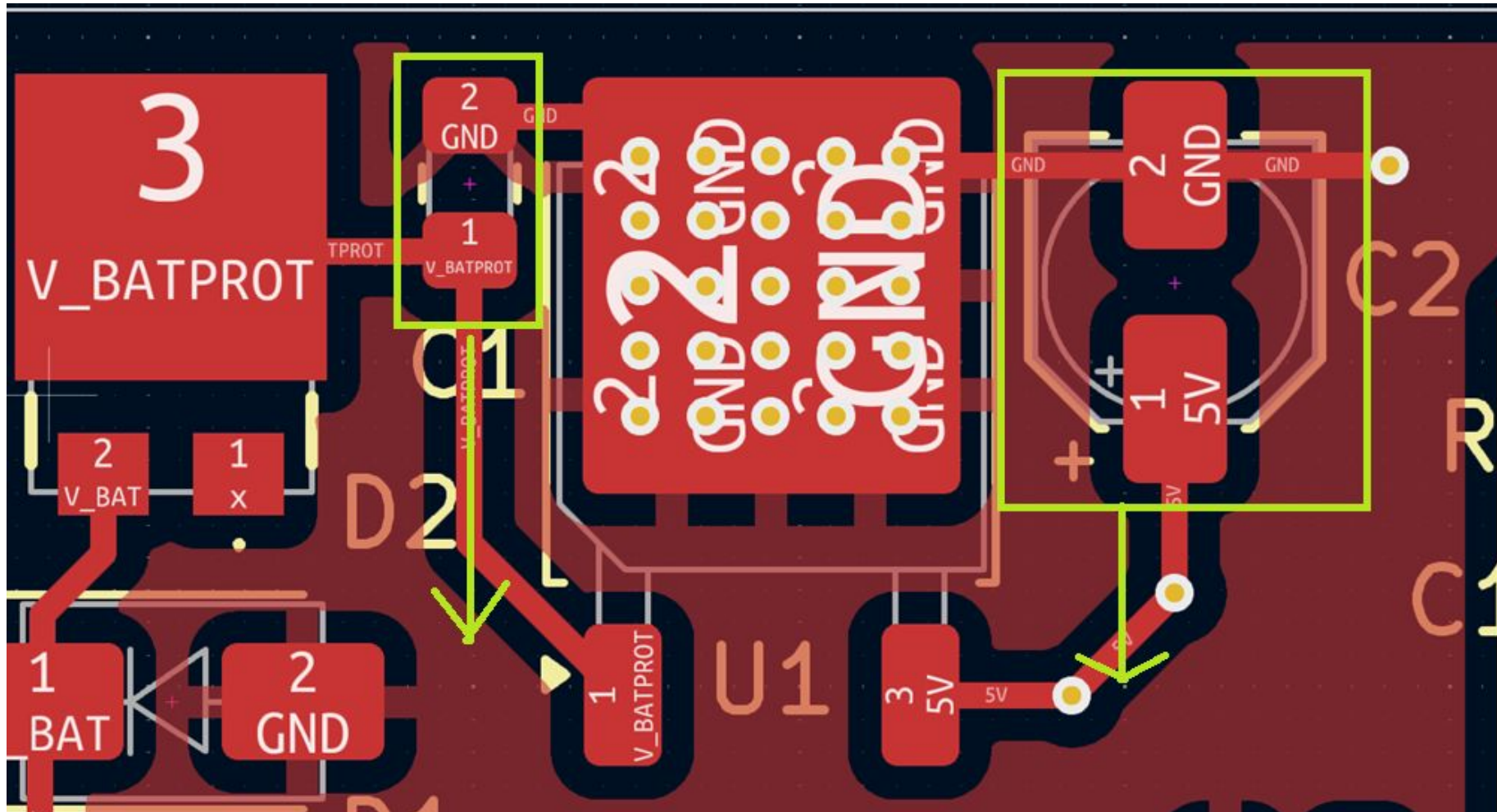
Finding 1: Thermal Issue

- An LDO with 12V input voltage and 5V output voltage was used. Issues such as power dissipation and thermal analysis should be evaluated by considering parameters such as ambient temperature and output current. I suggest you research this topic.



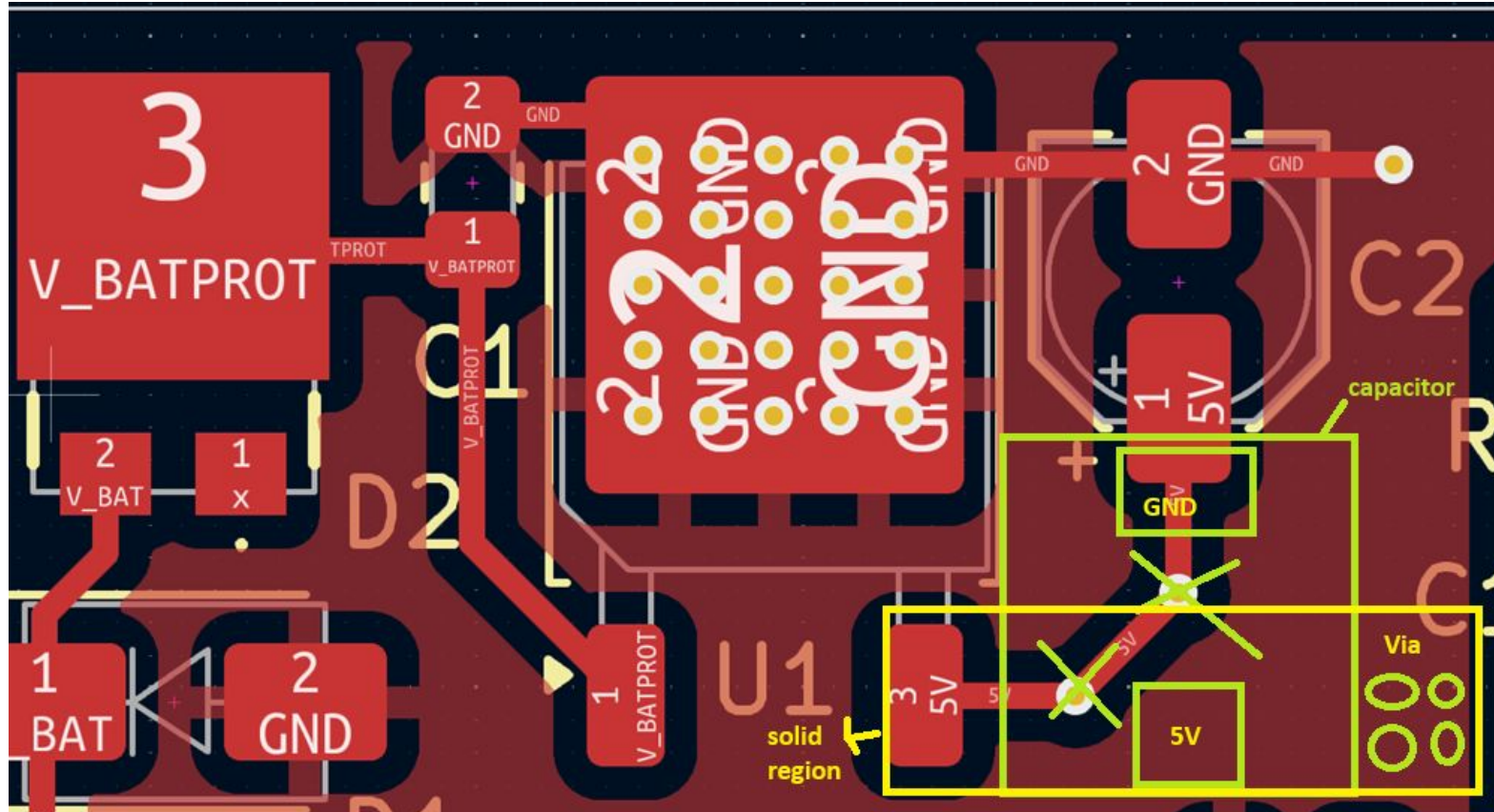
Finding 2: Capacitor Placement

- Capacitors can be placed closer to the LDO input and output pins.



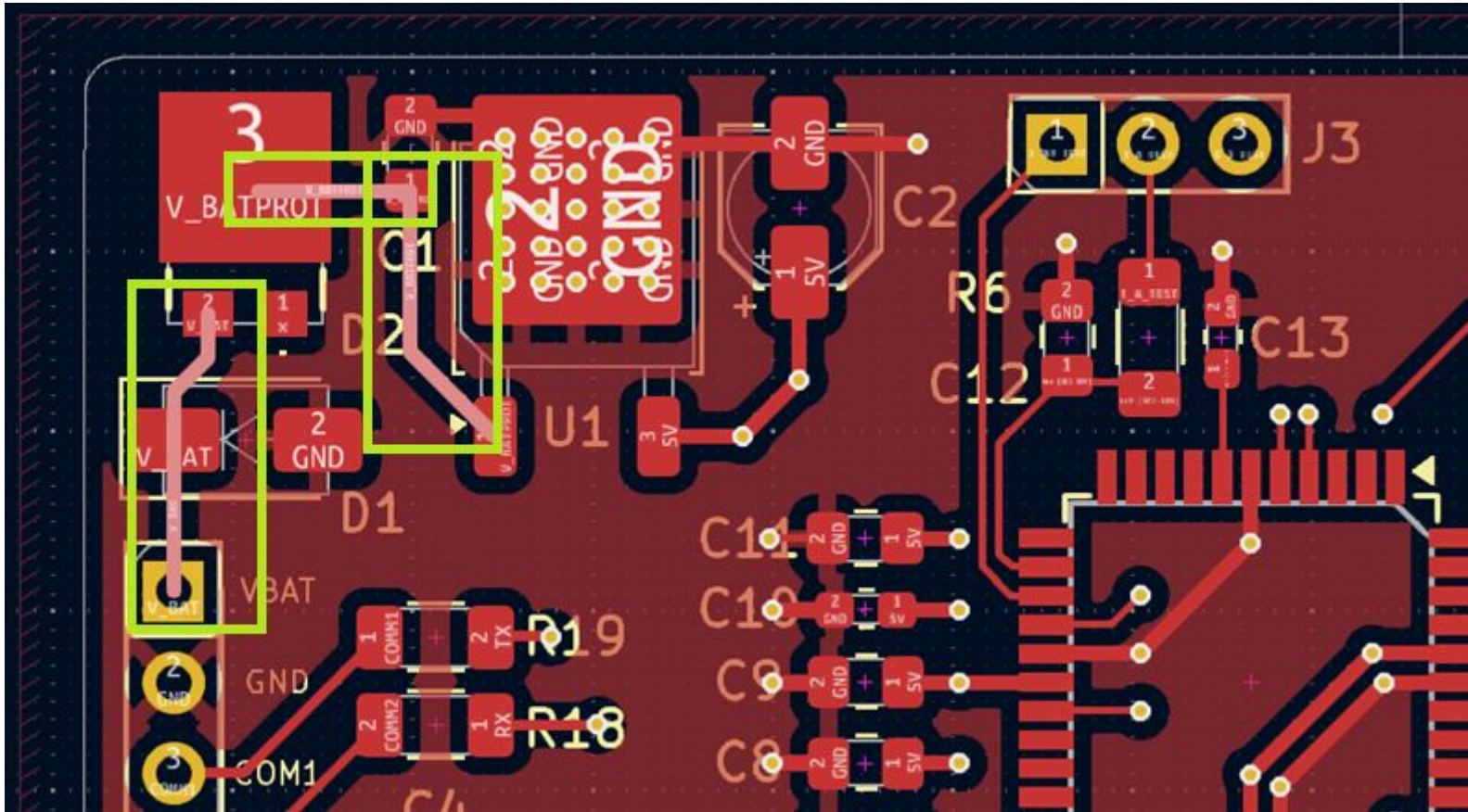
Finding 3: 5V Voltage

- 5V can enter the Electrolytic capacitor after the LDO output pin and then be distributed to the board from the 3rd layer with vias.
- Via numbers are related to current capability. I suggest you to investigate.



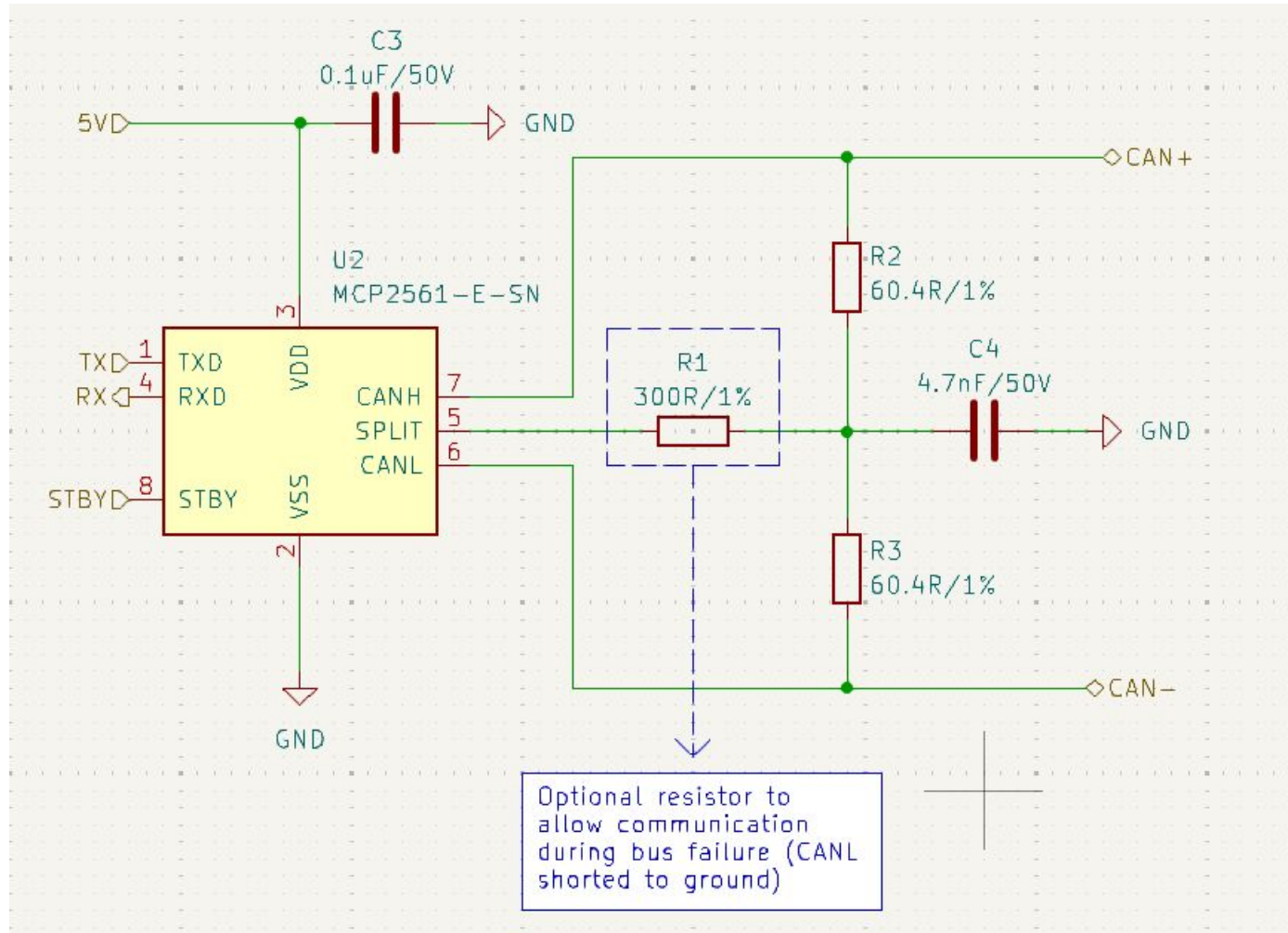
Finding 4: Current Capability

- Battery voltage tracks can be thickened if necessary, according to the current requirement of the system.
You can download and check the PCB Saturn Tool for current calculation.



CAN INTERFACE

Finding 5: Use of TVS and CMC



- The following documents about the use of TVS (Transient-voltage-suppression diode, CMC (Common Mode Choke) for CAN Communication interfaces should be examined. You can also get information about the layout placement of these components.

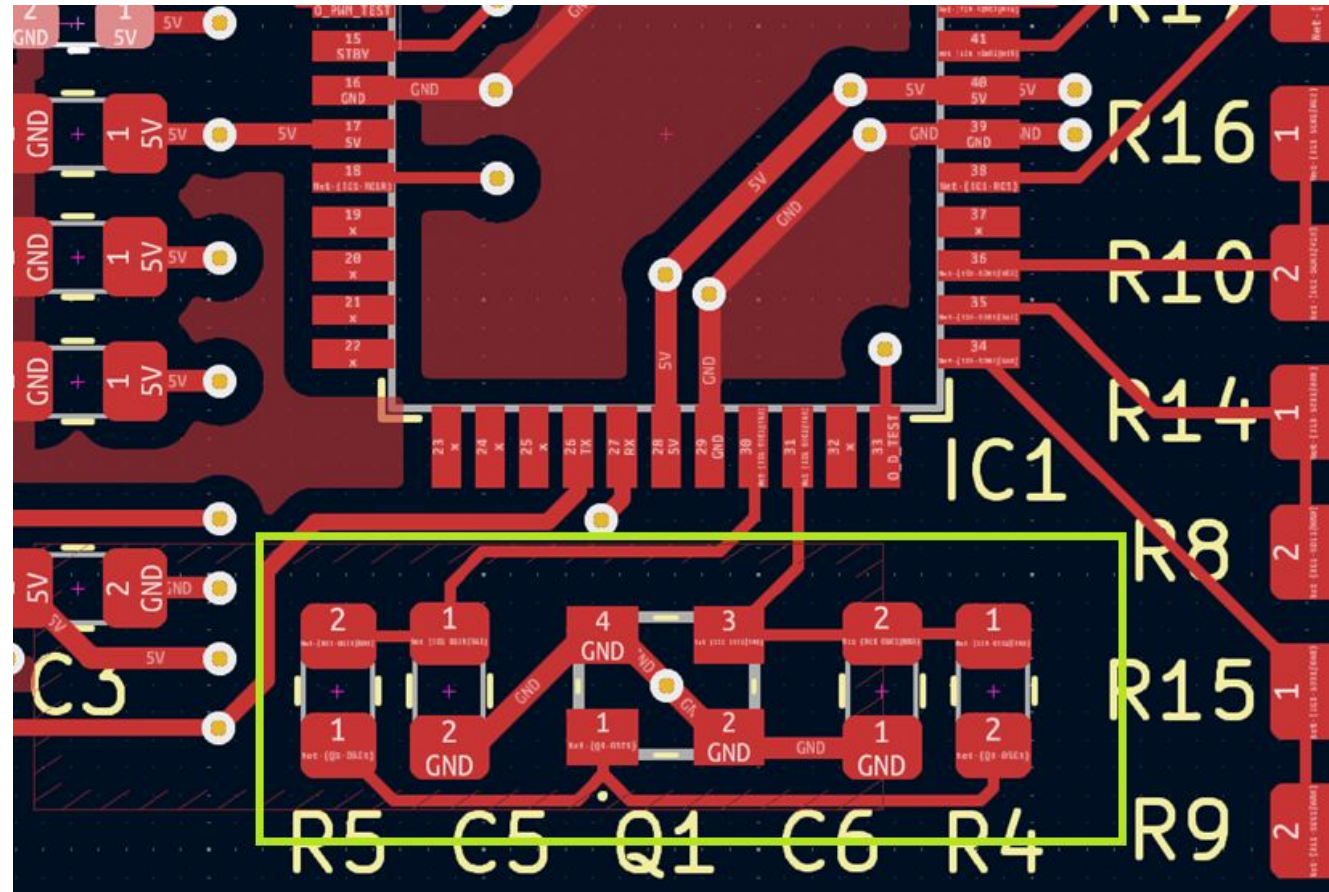
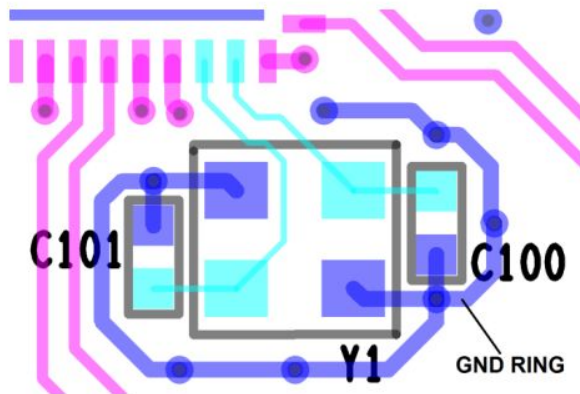
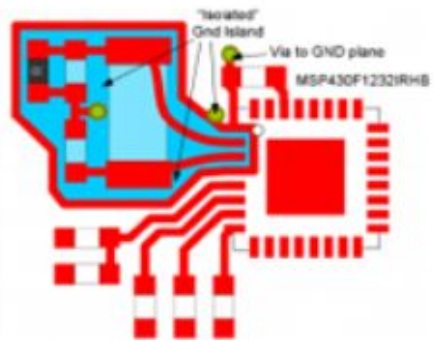
<https://www.onsemi.com/pub/Collateral/AND8169-D.PDF>

<https://www.onsemi.com/pub/Collateral/AND8253-D.PDF>

dsPIC33EV128

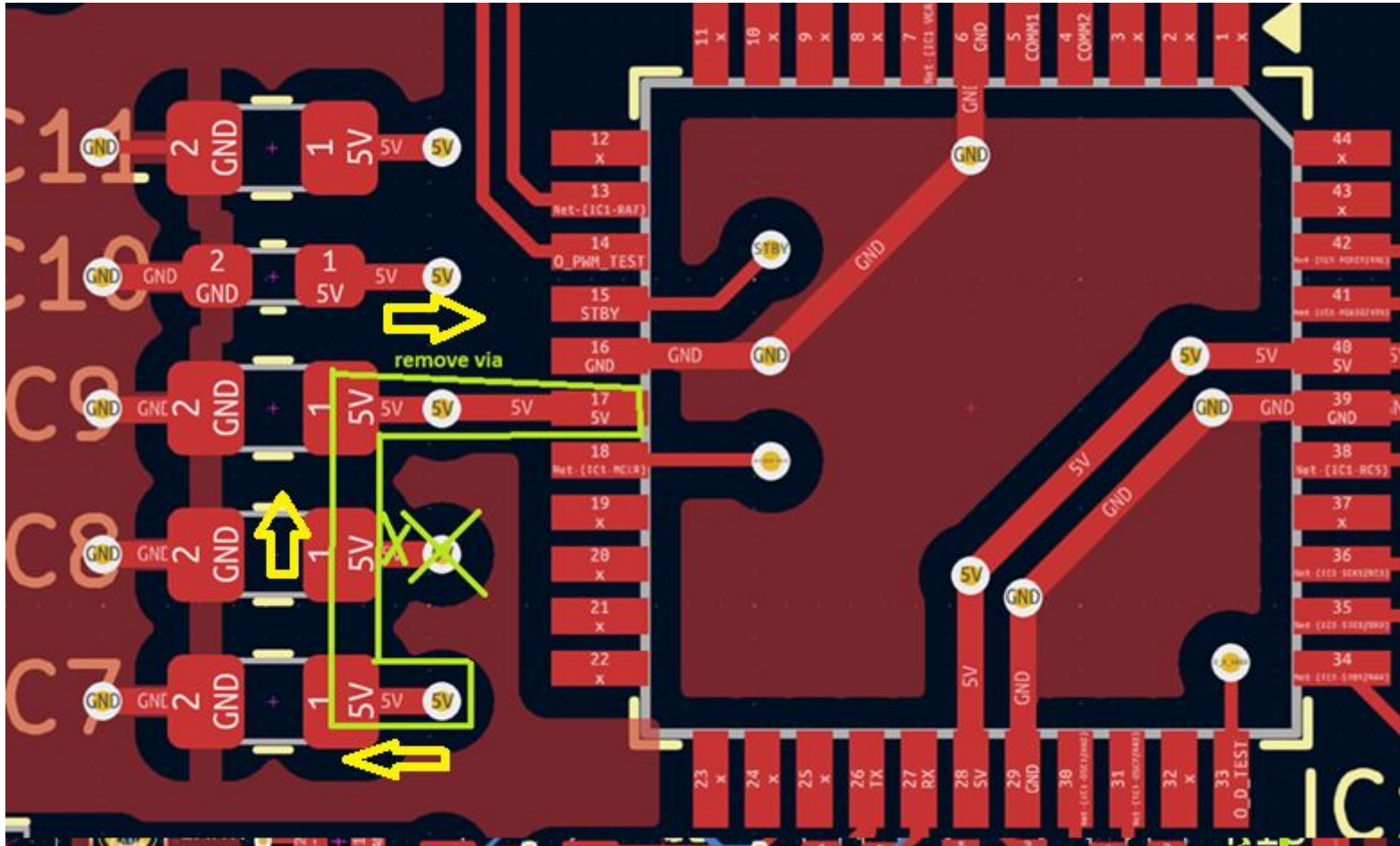
Finding 6: Oscillator

- The oscillator layout is not bad, but I recommend you to search for oscillator layout recommendations in order to make a better layout. (Especially Ground connection of the oscillator)

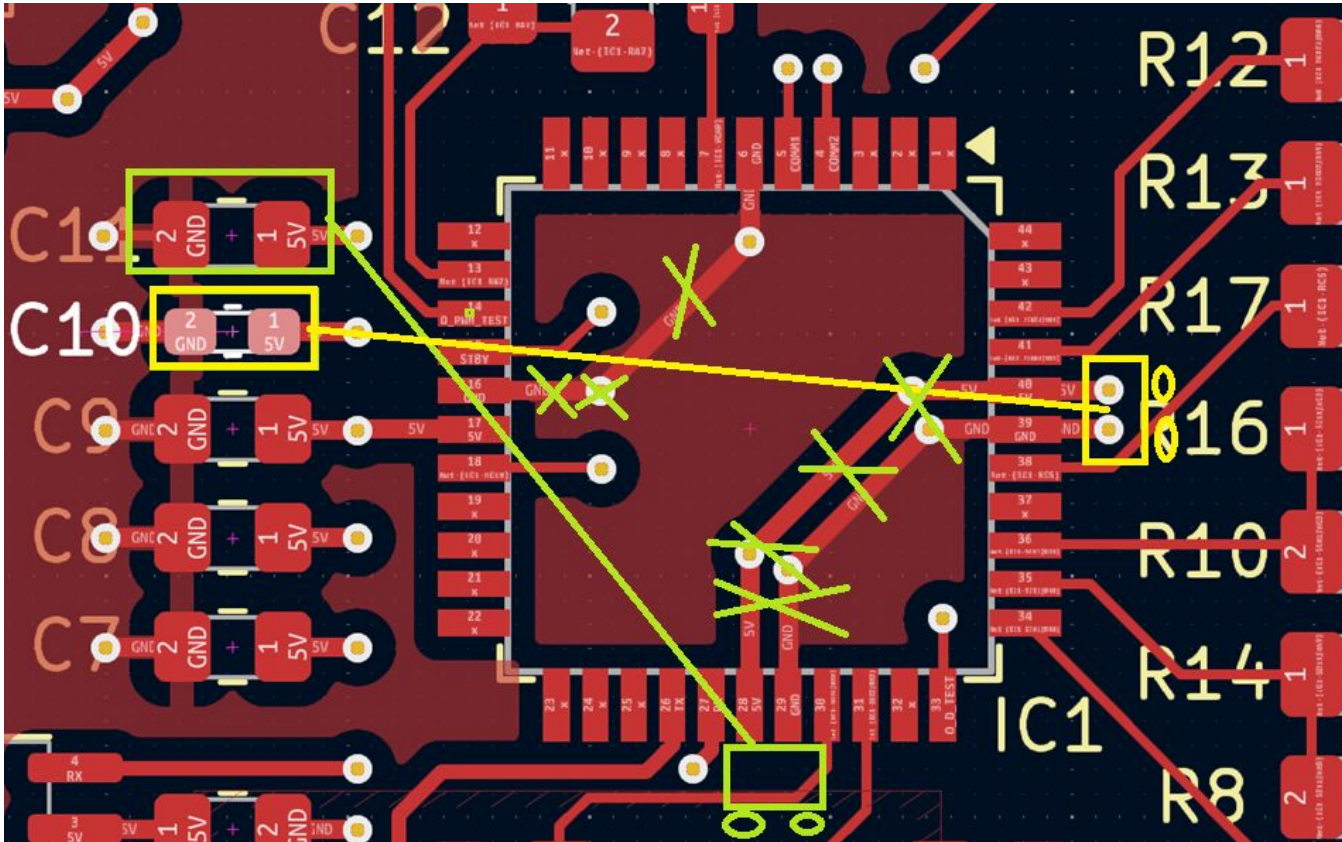


Finding 7: 5V Voltage Flow Direction

- 5V voltage should pass through the capacitors respectively and input to pin 17.

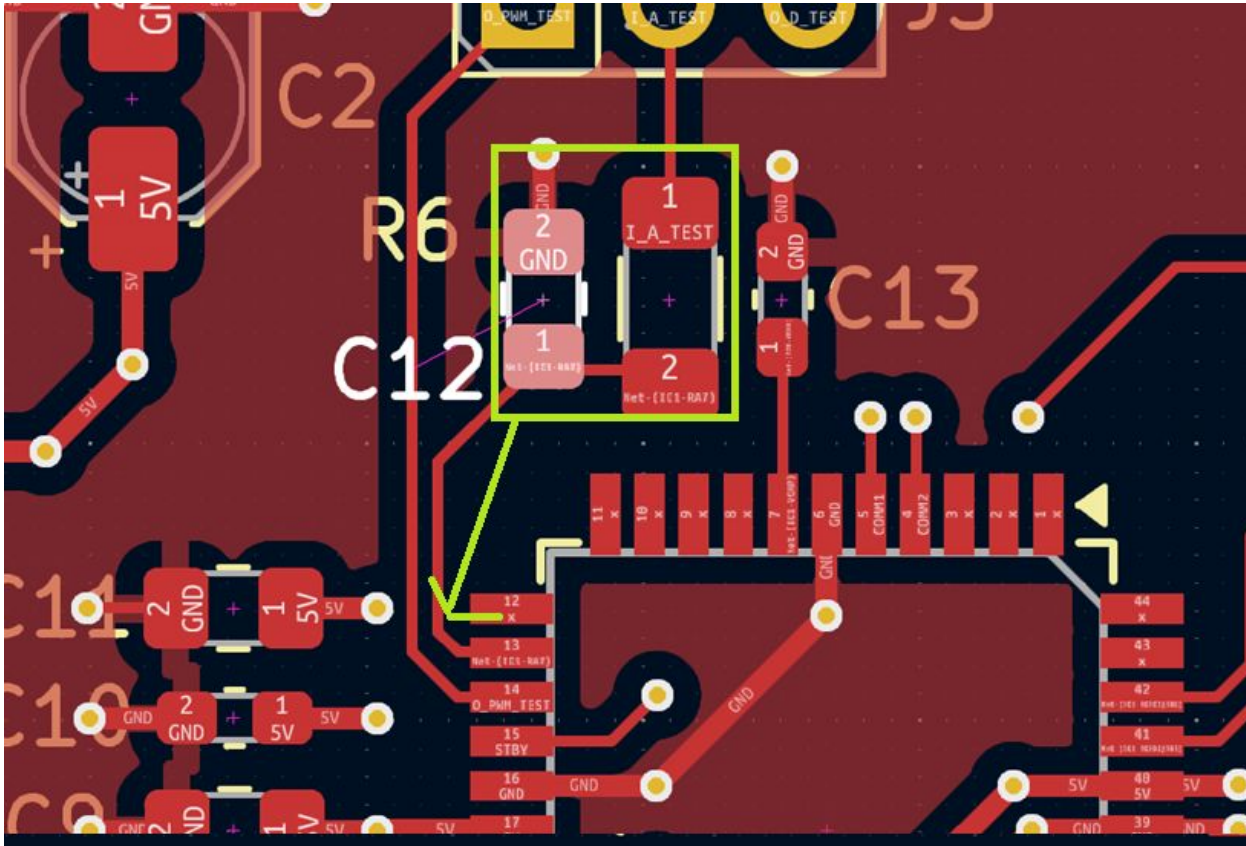


Finding 8: Capacitor Placement



- Capacitors C10-C11 must be placed close to the corresponding pins.
- No need for intermediate connections and vias. After moving the capacitors to the right places, you will be able to get the 5V and GND signals from the 2nd and 3rd layer with the vias.

Finding 9: Low Pass Filter Placement

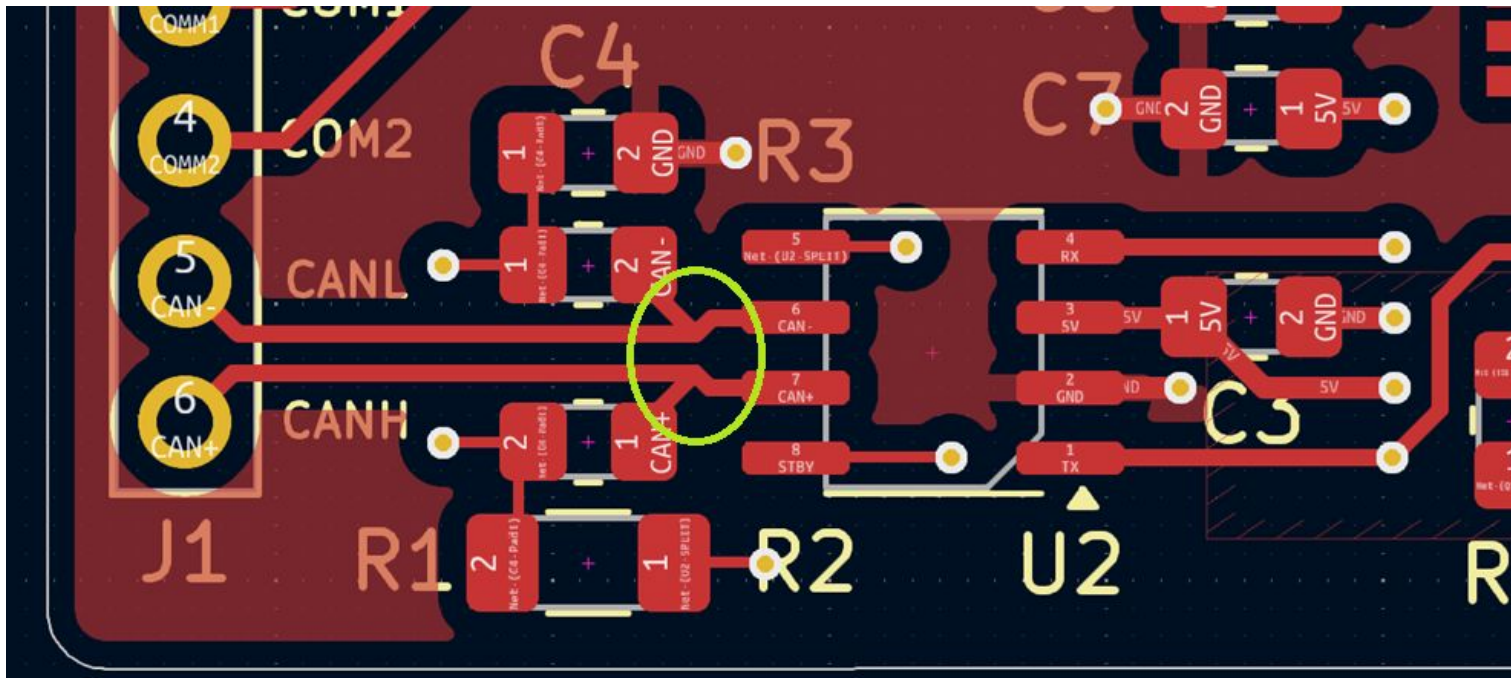
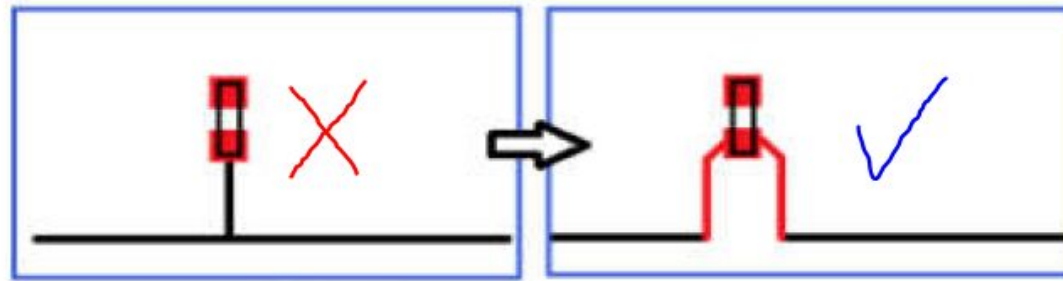
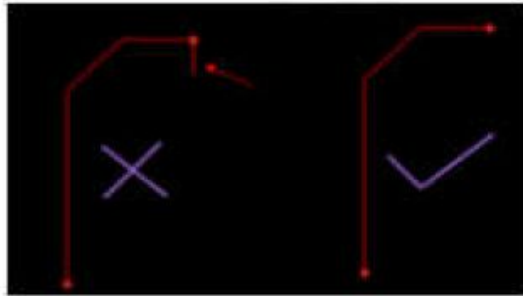


- R6-C12 low pass filter components should be placed as close as possible to the relevant pin. (Tip: Since C10 and C11 capacitors will be moved up, these components can be shifted to the free space).

Layout Issues

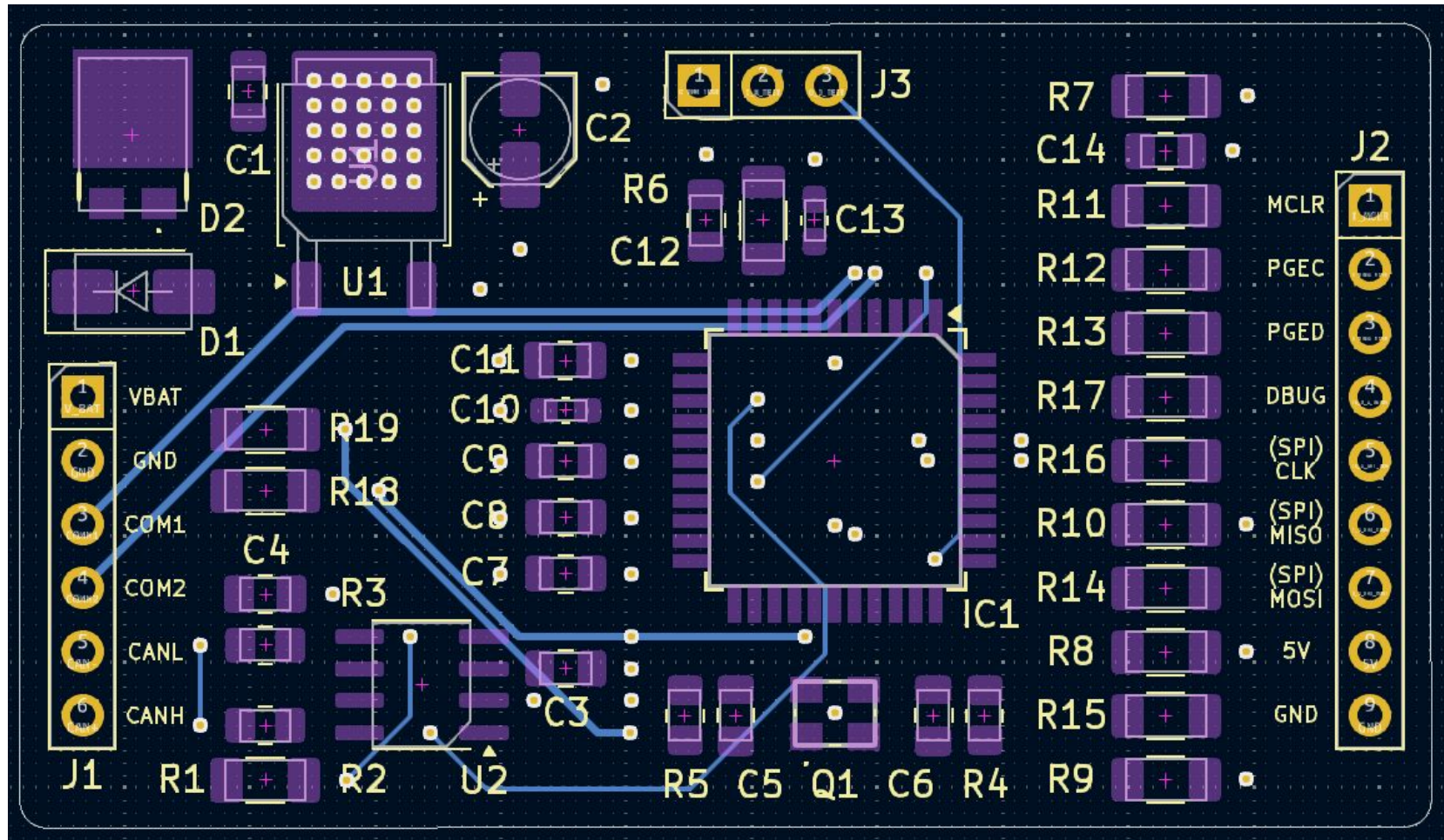
Finding 10: EMC and EMI Improvement

- This is not critical. This type of connection is called a stub. It can cause reflection in signals, distortion of the signal. It is an undesirable situation. You can also search as PCB Layout track stub.



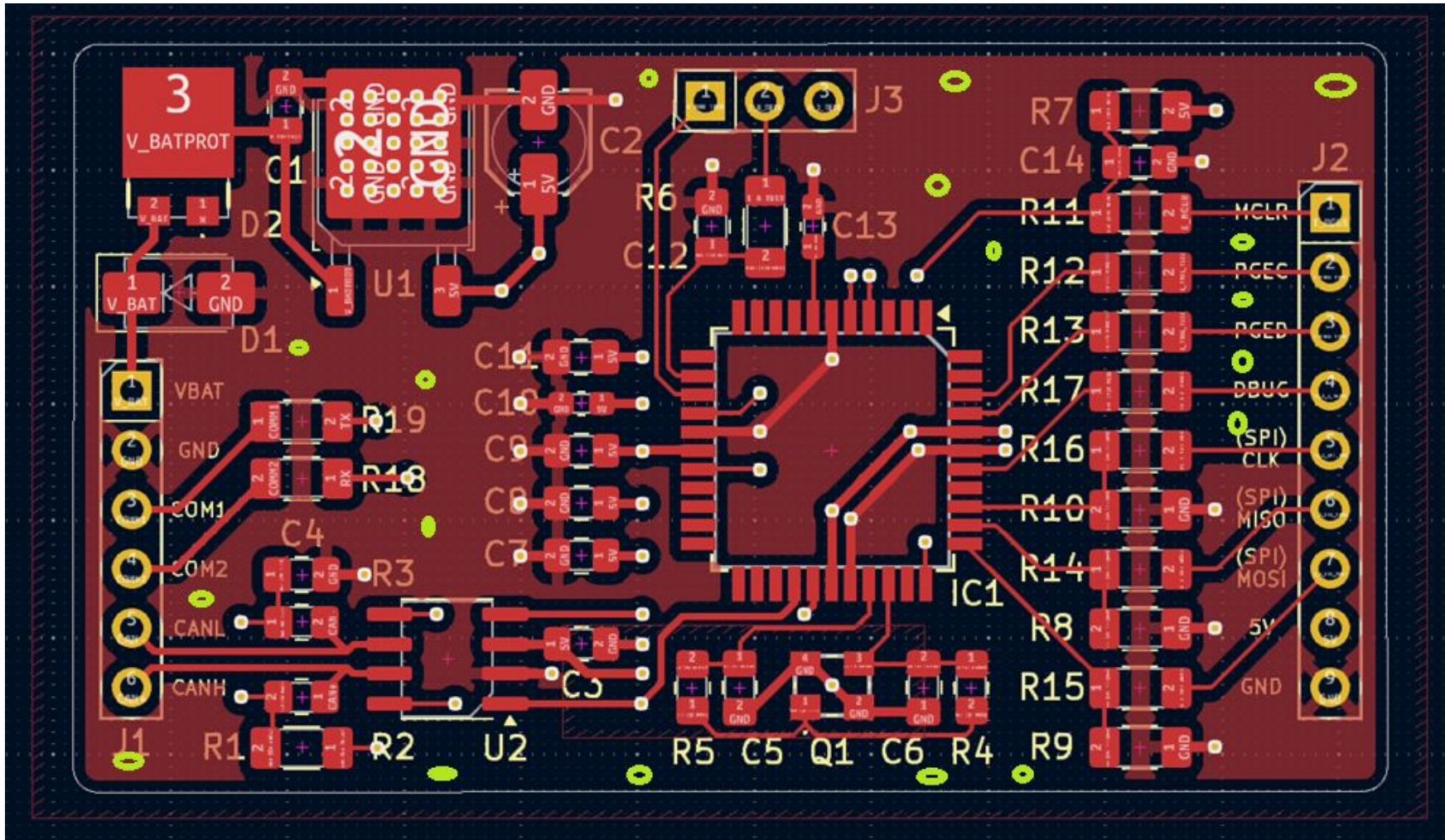
Finding 11: EMC and EMI Improvement

- Improvements in EMC and EMI can be achieved by improving ground connections and obtaining low impedance connections by adding GROUND polygon to the bottom layer. Thus, noise and interference can be reduced.



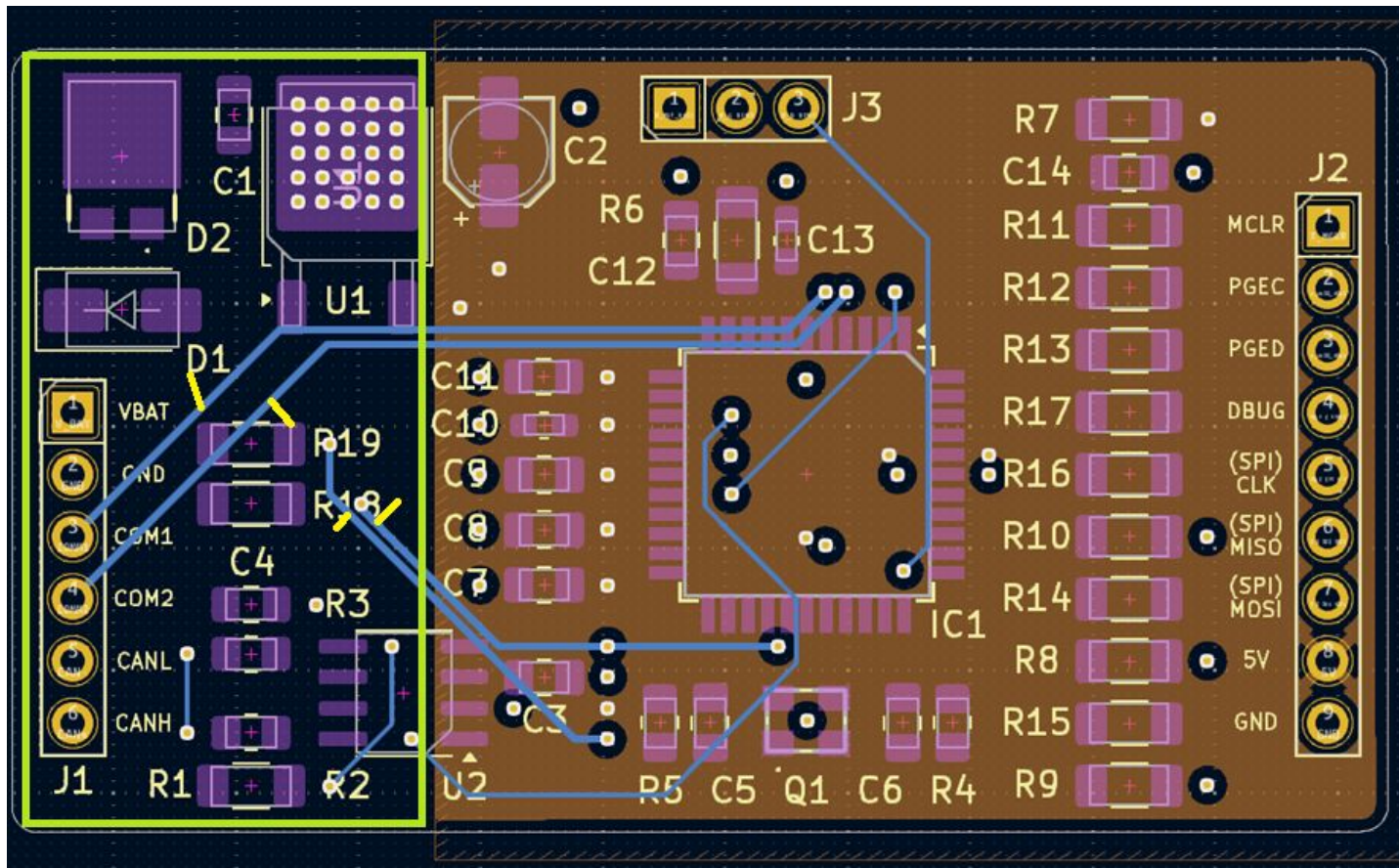
Finding 12: EMC and EMI Improvement

- The number of ground via seems to be a bit low. The GROUND vias can be placed as green marked and GROUND copper can be filled. Stitching ground vias is essential for high quality ground connections, lower noise, more stable performance and a more reliable design.



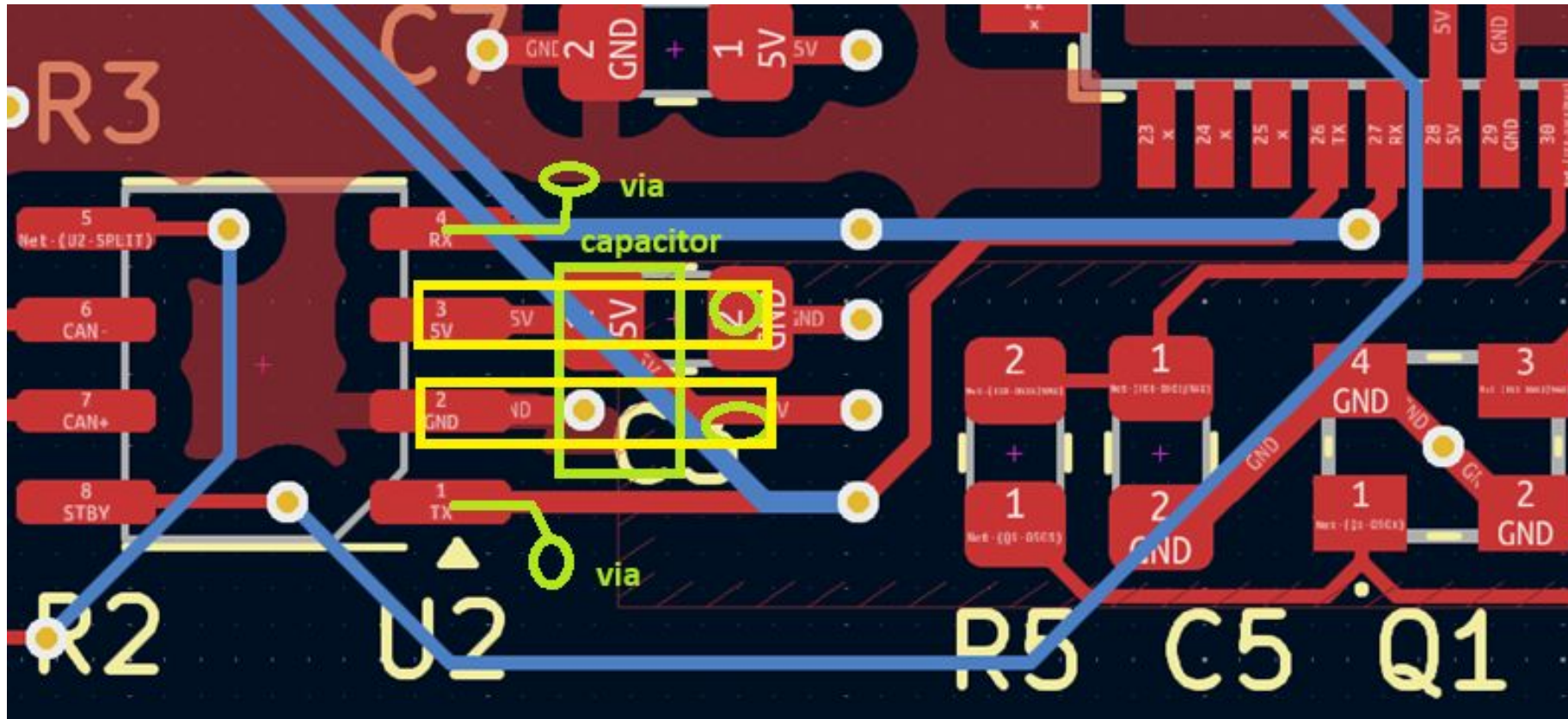
Finding 13: EMC and EMI Improvement

- For the references of the signals in the bottom layer, the free space in layer 3 can be filled with the GROUND polygon.
- In this way, the signals in the bottom layer get their ground references from the ground plane in layer 3 instead of the ground plane in layer 2 which is far away. This improves signal integrity.



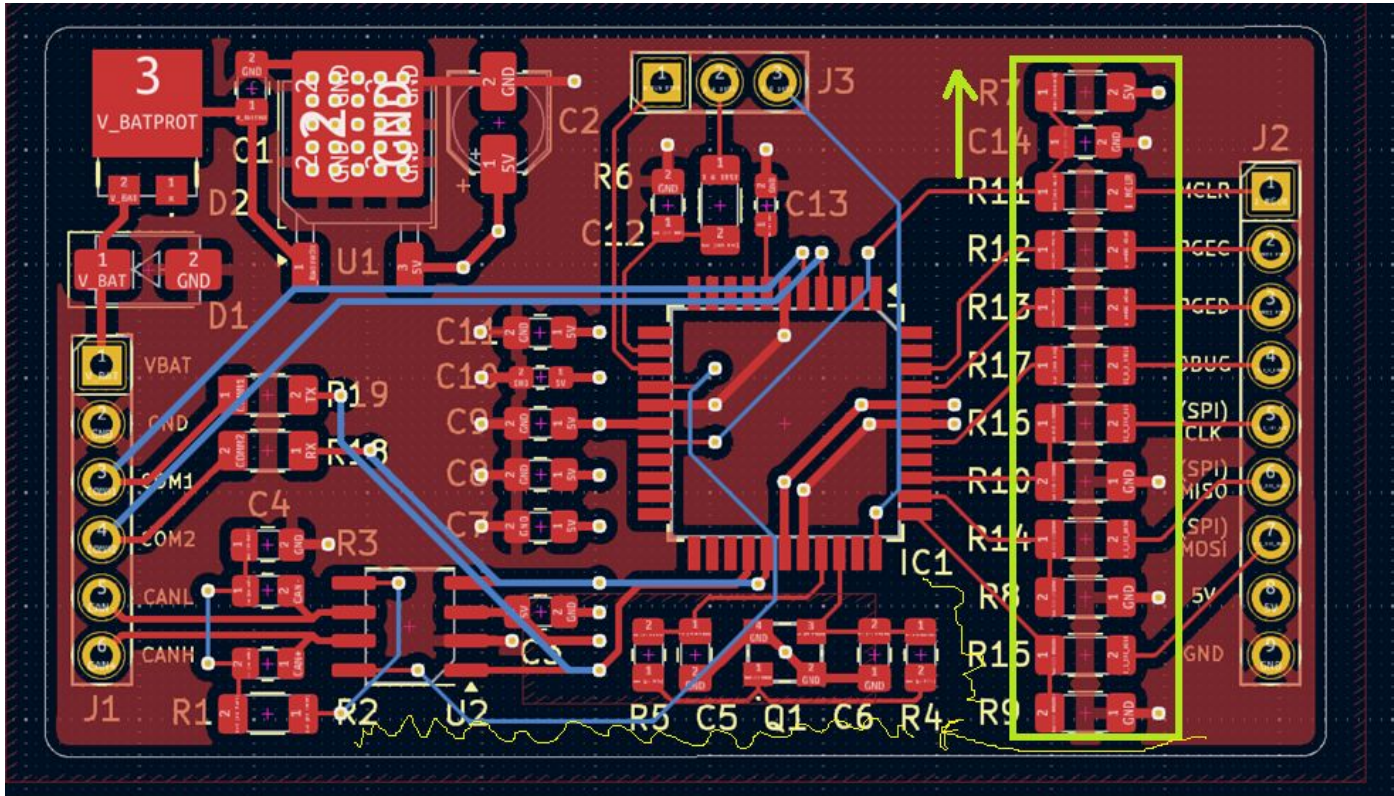
Finding 14: Capacitor Placement

- Not too critical, just a tip for future projects, if you place the TX-RX vias as shown in the image, you can place the decoupling capacitor closer and neater to the 5V and GND pins. This will give you a smoother voltage signal.

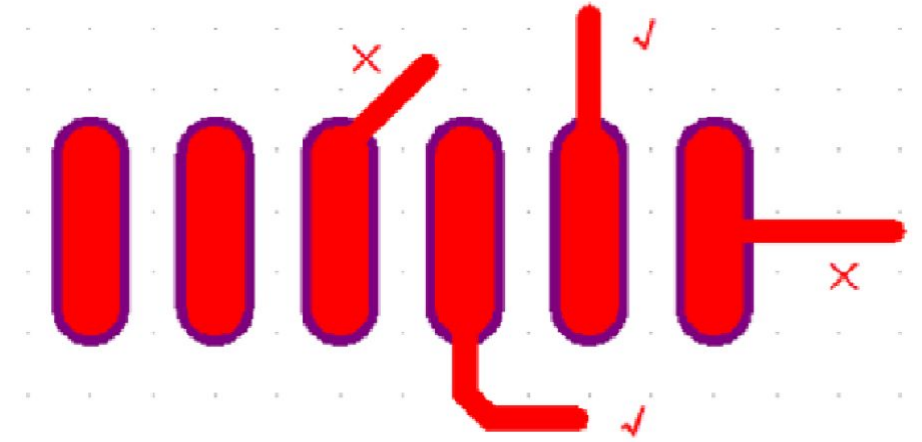
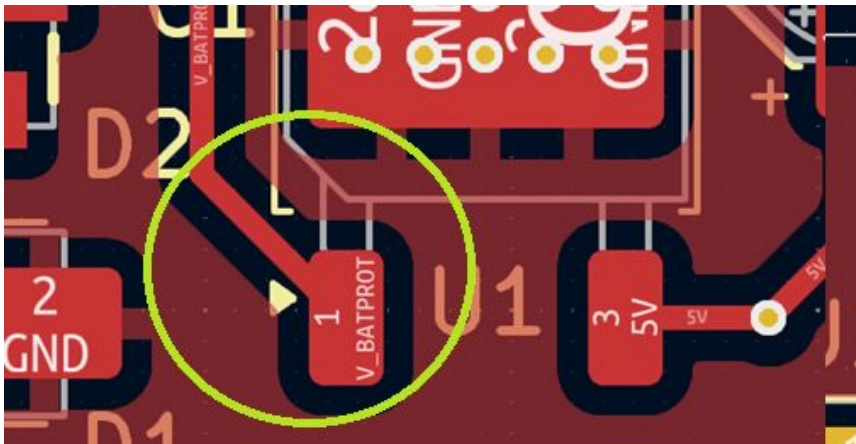
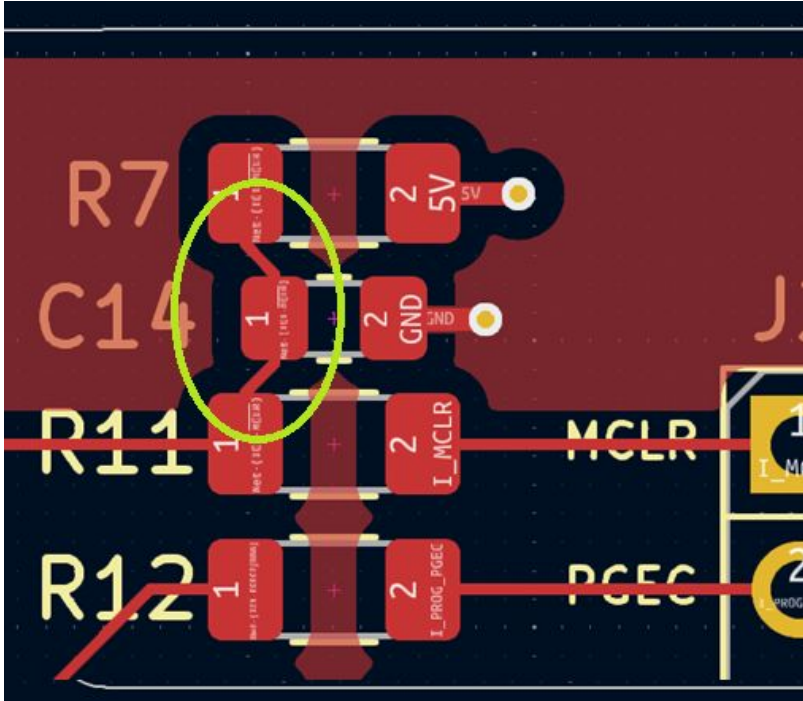


Finding 15: EMC and EMI Improvement

- This is not critical, but it can give you a hint for future projects. You can improve your design with small improvements. For example, if you can move the components on the right up a little bit, the GROUND polygon can be formed from the space you have created below and fill the empty spaces on the left (**yellow marked**). This is just an example, you can examine the whole board and make such improvements.
- Remember, improvements are always possible to improve your design. You can achieve a better design with small touches.



Finding 16: Layout Tips



- Although it is not a problem for the moment, I will give you some tips to help you when you layout a more comprehensive project in the coming years.
- The cross (X) marked connections you see in the image are undesirable for pcb production, weakening the strength of the connection between the pad and the track. Therefore, connections marked with a tick (✓) will be safer.