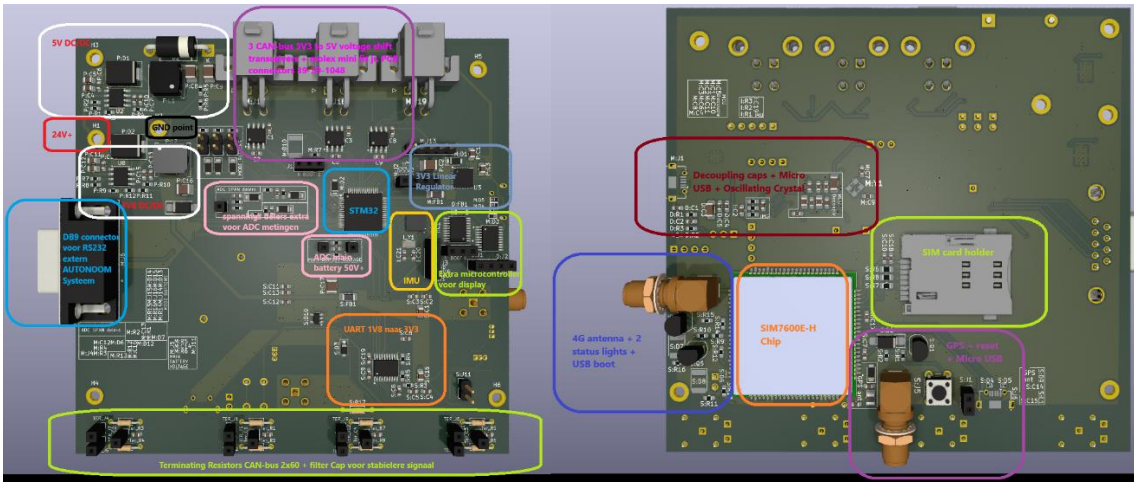
MCU PCB User Guide

Required:

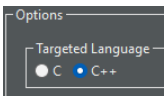
- ST-Link for programming
- Power supply
- MCU PCB

Legend:



Programming:

1. Download STM32CubeIDE.
2. Connect the ST-Link programmer to the MCU PCB.
3. Create a project for the STM32G474RET6 chip.
4. Select these options, then click Next and Finish:



5. Copy the pinout file from the GIT or Teams/
   FAT Minor 2024-2025 / Dennis Boekholtz / CODE
   (File: RTR_PINOUT_STM32)

   If the pinout looks the same as in the picture,
   then the import is correct,
   and you can start programming.

Figure 1 SWD pins for stm

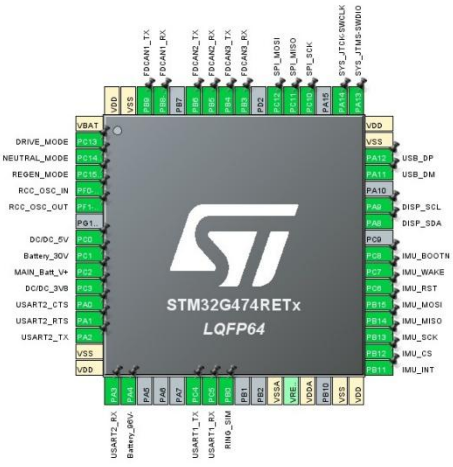
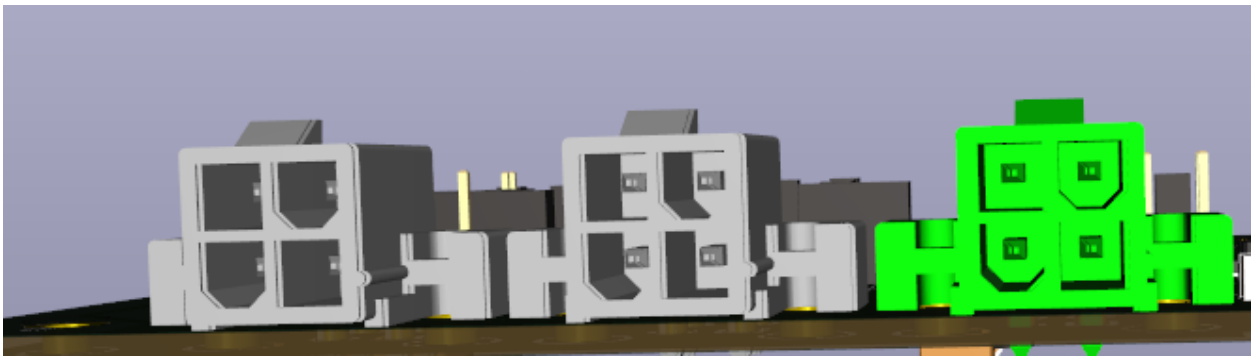


Figure 2 MCU pinout

Notes:

1. If you have never worked with the STM32 workspace, look on YouTube for additional information.
2. You can also work in VS Code with the STM32.
3. To use the PCB in real-world conditions, connect the 24V+ to a 24V power supply and connect the GND to ground.
4. For CAN-bus, use the MOLEX 4-pin connector. The pins are ordered from top right to bottom left.
   - CANlow - CANhigh
   - GND -5V+

   
5. There is a design flaw in the 4G module, so it is not usable. (The problem has not been found yet.)
6. There is a possibility that the CAN-ICs have been blown up, so a new PCB needs to be resoldered. There should be enough spare parts for a second PCB.
7. If you only want to test the STM32G474RET6 chip, this is still possible because it still works.