Project #3, STM32\_RF <https://github.com/ravindsk/STM32_RF_PCB_KiCAD>

Why?

* To learn PCB design for Radio Frequency chip NRF24L01 (2.4Ghz Radio Transceiver)

A close-up of several towers

Description automatically generated

How?

* Designed a custom PCB for nRF24L01 and STM32L432KBUx using KiCAD

Voltage regulator (+5VUSB to +3V3)

Microcontroller (STM32L)

Debugger

nRF24L01 transceiver

Antenna Impedance matching (50 ohm)

SMA connector

A close-up of a circuit board

Description automatically generated