Department of Engineering Cybernetics

TTK4550 - Specialization project

Sensor data acquisition board for nRF52840 maze mapping robots

Author:
Emanuela Tuong Vi Thi Tran

Autumn, 2022

Preface

This project thesis is part of the evaluation for the TTK4550 course at the Norwegian University of Science and Technology (NTNU) in the Department of Cybernetics. As part of the evaluation for this 15-credit course, I was given a robot from the SLAM project and tasked with creating a sensor data acquisition board to improve the robot's capabilities.

I would like to thank my to my supervisor, Professor Tor Onshus, for his guidance and interesting discussions throughout this thesis. I would also like to thank the mechanical workshop at the Cybernetics Department for lending equipment and helping me with soldering and testing.

It has been a great experience working with the other students on the SLAM project. They have been a valuable resource for discussions and helping me to fully understand the project. I am grateful for their cooperation.

I have found this project to be both valuable and rewarding. It has given me the chance to create a system from scratch, which has been a unique and interesting experience as a student in the Department of Electronics Systems. I have also had the opportunity to work with students from the Department of Cybernetics, which has allowed me to gain a different perspective on embedded systems.

Problem Description

In this project, a sensor data acquisition board will be developed for maze-mapping robots. The board will be based on Nordic Semiconductor's nRF52840 System on Chip(SoC). This project involves the design and development of new embedded hardware, specifically a printed circuit board(PCB). The development of the new PCB will take into account both the current robots and future Simultaneous Localization and Mapping (SLAM) implementation.

A PCB featuring two SoCs, one dedicated as a SLAM handler and one dedicated for steering a servo tower on the robot.

The specific tasks outlined for the project are as follows:

- Review earlier hardware of the robots and set specifications for the new embedded system.
- Develop schematic and layout for the PCB.
- Assembling and testing of the PCB

Software implementation is not within the scope of this project.

Summary and Conclusion

The purpose of this project is to create a new system that will enhance the hardware capabilities of maze mapping robots. Currently, these robots are using the Nordic Semiconductors development kit, nRF52840-DK, and the goal of the new system is to distribute certain tasks onto multiple processors. There are four tasks that are running on the nRF52840-DK and it is desired that two of those four tasks will be running on the new system. Due to the design space of the robots it was desired to develop a new system that would also substitute the current custom shield that is set on top of the development kit. The objective of this new system is to provide the same benefits as the custom shield while also adding additional functionality.

A printed circuit board has been developed based on two nRF52840 system on a chip. One serving as the servo tower handler and one as the SLAM handler. The design of the PCB has involved designing circuits, schematic and layout design, production and assembling of the printed circuit board, and testing. PCB design for a new systems usually takes multiple revisions and as the new system is part of a larger robot project it was desired to design the PCB as a prototype, specifically for debug and testing.

The PCB design was created using the software tool Altium Designer. JLCPCB was the production site used to manufacture the PCB and assemble some of the components. The remaining components that were not in stock at JLCPCB were purchased from other component suppliers and hand soldered. The soldering and testing of the PCB was carried out in the mechanical workshop at the Department of Cybernetics.

The PCB has four main circuits: one for stepping 12 V to 5 V, one for stepping down 5 V to 3.3 V, one for controlling the servo tower using the nRF52840, and one for handling SLAM using the nRF52840. It has four layers: a top layer, a ground layer, a power layer, and a bottom layer, and has a copper thickness of 1 oz. The PCB can be used alone or together with the nRF52840-DK. It has stackable headers that fits on top of the development kit such that the pin-outs of the nRF52840-DK are available for the new system. The new system can also deliver power to the nRF52840-DK. The PCB includes connectors such that the new system and nRF52840-DK can connect to the sensors and actuators on the robot.

The test results shows that both regulators delivers the desired voltages. There are connections between the PCB and the development kit with only one of six connectors having the wrong size. The fault can be modified physically as a short term modifications and it is recommended to have the right size for a new revision. Visually no other faults have been detected. Due to limited time further testing of the board is left for future work.

This project has laid the foundation for the new system, and the structure has been established for future development and testing to proceed.

Abbreviations

ADC - Analog to Digital Converter
BOM - Bill of Material
CAN - Controller Area Network
DAC - Digital to Analog Converter
DC - Direct Current
ESD - Electrostatic Discharge
GPIO - General Purpose Input Output
I2C - Inter-Intergrated Circuit
IMU - Intertial Measurement Unit
IR - Infrared
LED - Light Emitting Diode
LIDAR - Light Detection and Ranging
OLED - Organic Light Emitting Diode
PCB - Printed Circuit Board
PWM - Pusle Width Modulation
SCL - Serial Clock
SDA - Serial Data
SLAM - Simultaneous Localization and Mapping
SMD - Surface Mount Device
SoC - System on a Chip
SPI - Serial Peripheral Interface
UART - Universal Asynchronous Receiver Transmitter
USB - Universal Serial Bus

Table of Contents

List of Figures

List of Tables

1 Introduction

The SLAM robot project involves the use of small, autonomous vehicles to communicate with a central server in order to map an unfamiliar area. These robots, which are based on the nRF52840 SoC, are responsible for navigating to specific coordinates provided by the server without relying on external localization technology. They also collect and transmit measurements of their surroundings, including distances to nearby objects. By combining these measurements from multiple robots, the central server is able to build a detailed map of the area. This enables the robots to explore and map new environments independently.

1.1 Motivation

For future development of the SLAM robot, it is desired to separate the software on the development kit and have desired nRF52840 SoC for specific tasks. As the system has become more complex over time, dividing it into smaller parts will make it easier to iterate, debug, and further develop the system without encountering major setbacks. Dividing each task or module will enable the development of new solutions without hindering the progress of the project.

By creating new hardware with two nRF52840 chips to perform two tasks, we have taken the first steps towards dividing the software for the project. The robot will now have access to three System on a Chip units to run its software. The initial step in this process is to assign the task of running the servo tower on the robot to one nRF52840, while using the other to run SLAM algorithms.

1.2 Outline

The goal of this thesis is to both recreate and comprehend the topic at hand. The structure of the thesis is meant to assist in the reconstruction process, while the content is intended to provide a thorough understanding of the subject matter. The purpose of this report is not only to provide the reader with step-by-step instructions for reconstruction, but also to provide them with the understanding and knowledge needed to make modifications and improvements to it.

Chapter 2 describes the hardware background on the current robot and related previous work.

Chapter 3 covers the theory related to this project.

Chapter 4 covers the methods and steps toward designing embedded hardware.

Chapter 5 presents the final results in two parts: physical design results and results from testing.

Chapter 6 covers the discussion around the results.

Chapter 7 presents the suggestions for future work.

2 Background

In this chapter of the nRF52840 robot will be introduced in this chapter as well as the related previous work.

2.1 Robot hardware components

In this section, the hardware of the nRF52840 robot will be introduced. Table 1 provides an overview of the robot's hardware components. Figure 1 shows an illustration of the robot's hardware layout from above. In addition to the components shown in the figure, the robot also has an IMU located under the bottom layer and an OLED display located on top of the shield.

Component	Quantity	Description
Development kit	1	nRF52840-DK
Shield	1	Custom shield for nRF52840-DK
DC motor	2	Machifit 26GA370 DC 12V 110 RPM
Motor driver	1	L298 H-Bridge Dual Bidirectional Motor Driver
Inertial Meausurement Unit	1	ICM20948
Servo motor	1	S05NF STD
IR range sensor	4	Sharp GP2Y0A21YK
Battery	2	Ansmann 10.8Vm 2.6Ah
OLED display	1	PEMENOL 128 x 64 LCD with SSD1306 driver

Table 1: nRF52840 Robot Hardware Overview

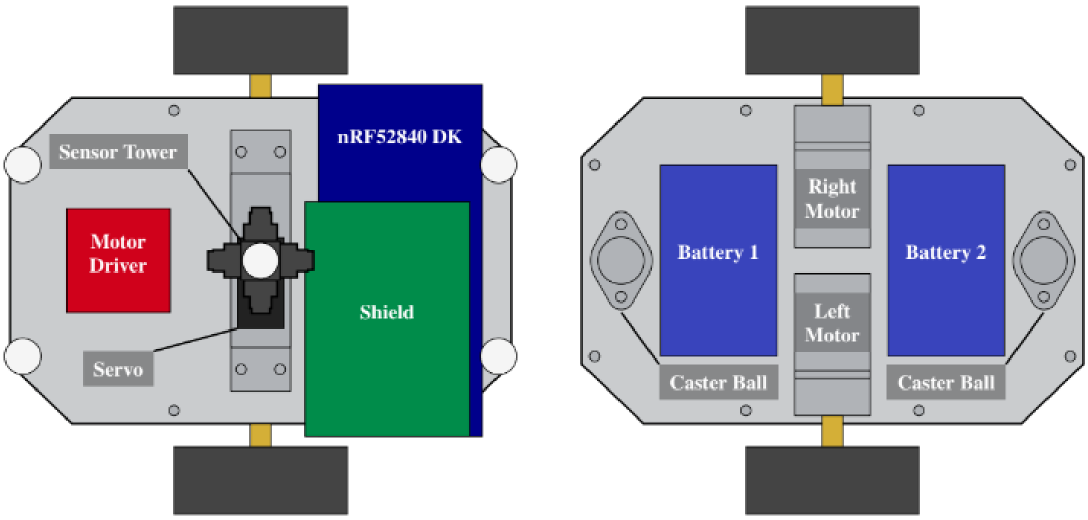


Figure 1: Hardware layout of the nRF Robot[1, p.5]

2.1.1 nRF52840 DK

The nRF52840-DK [3] development kit from Nordic Semiconductor serves as the primary computing module for the robot. The nRF52840 System on a Chip on the development kit supports Bluetooth low energy, Bluetooth mesh, NFC, Matter, Thread, and Zigbee. It is compatible with Arduino UNO Revision 3, making it easy to use third-party shields. The development kit comes equipped with an on-board SEGGER J-Link debugger that allows for programming and debugging of both the on-board SoC and external targets through the debug header. The nRF52840 DK can be powered through a USB connection, but it also has the capability to be powered by a wide range of external sources within the 1.7 to 5 volt supply range. In addition to USB, it has a CR2032 battery holder and a Li-Po battery connector, giving it the flexibility to be powered by various sources depending on the needs of the project.

2.1.2 nRF Shield

The nRF52840 shield was custom made by Jølsgard [2] and is attached on the top of the nRF52840 DK. The purpose of the shield is to connect all peripherals to the GPIO headers of the development kit.

2.1.3 DC Motors with Rotary Encoders

The robot uses two 12V DC-motors from Machifit [4]. Which are rated up to 100 rpm. Each motor has built-in quadrature encoders for measuring wheel angle and speed.

2.1.4 L298N Motor Driver

The L298[5] motor driver is a bi-directional motor driver based on the L298 Dual H-Bridge Motor Driver Integrated Circuit. According to its data sheet, it is ideal for robotic applications and can be easily connected to a microcontroller, requiring only a few control lines per motor. It is capable of controlling two motors at up to 2 A each in both directions.

2.1.5 ICM-20948 Inertial Measurement Unit

The ICM-20948[6] Inertial Measurement Unit is made up of a 3-axis MEMS-based gyroscope, accelerometer, and compass. It is located underneath the robot's chassis and is equipped with a digital motion processor that is used for simple filtering of sensor measurements and power management.

2.1.6 Servo Tower

The servo tower is made up of four 2YA21[7] Sharp infrared sensors. These sensors are mounted on a S05NF[8] servo motor and arranged radially with respect to each other, allowing the set of IR sensors to rotate. The sensor tower has a maximum rotation angle of 90° and each of the IR sensors have a valid measurement range of 0.1 to 0.8 m.

2.2 Previous Work

This section will present a review of the related previous work as the functionality of the nRF custom shield will be taken into account in the development of the new system.

2.2.1 nRF52840-DK-Shield

In 2020 E.Jølsgard designed a custom shield [2] for the nRF52840-DK shown in figure 2. The purpose of the shield is to connect the nRF52840-DK to peripheral units as sensors, actuators and a power supply. Figure 3 shows the shield on top of the nRF52840-DK. The custom shield was designed using the software tool KiCad[9] where the shield has a layer stackup of 2 layers: top and bottom, and automatic routing has been used for layout.

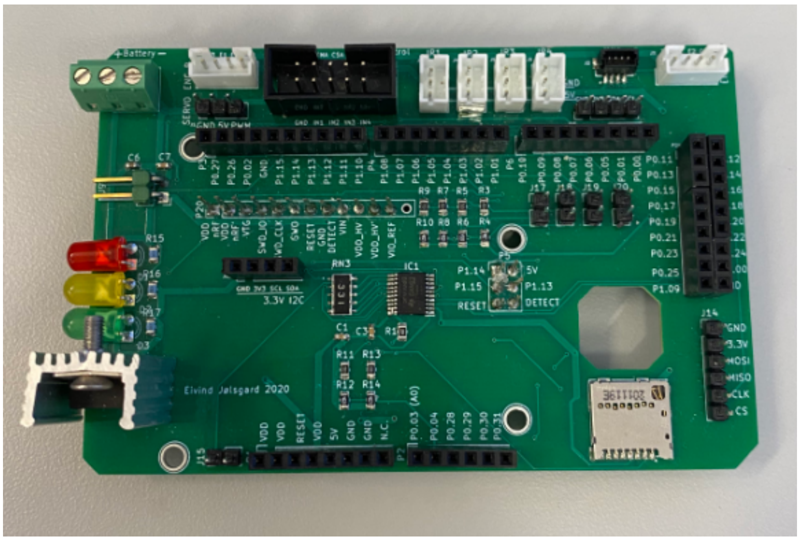


Figure 2: Shield to nRF52840-DK

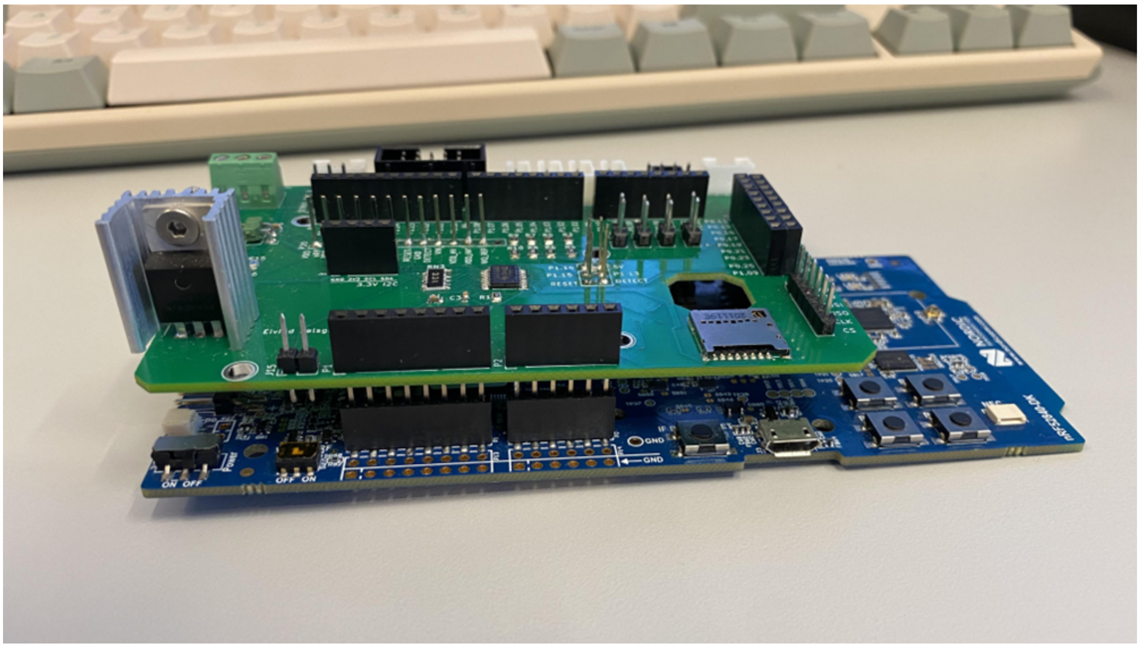


Figure 3: nRF52840-DK shield stacked on top of nRF52840-DK

The nRF52840-DK-shield receives power from the robot's batteries and steps down the voltage to the desired voltages such as 5 V and 3.3 V. The choice fell on to a linear voltage regulator for 5 V and a logic level converter for 3.3 V due to Jølsgaard previous experience with those components. In addition to power circuits the shield includes connectors, listed in table 2, three LEDs, microSD card reader. The schematics is done in one sheet and Jølsgaard has sorted it with boxes around the different parts of the shield. Figure 4 shows the schematics sheet for the custom shield in KiCad.

In his project thesis Jølgaard concludes that the shield works as intended with the exception for the µSD card.

Connector	Quantity
IR - sensor	4
Encoder	2
Motor driver	1
IMU	1
Servo motor	1
nRF52840 DK	6
I2C	2
SPI	1
Battery	1

Table 2: Connectors to the custom shield

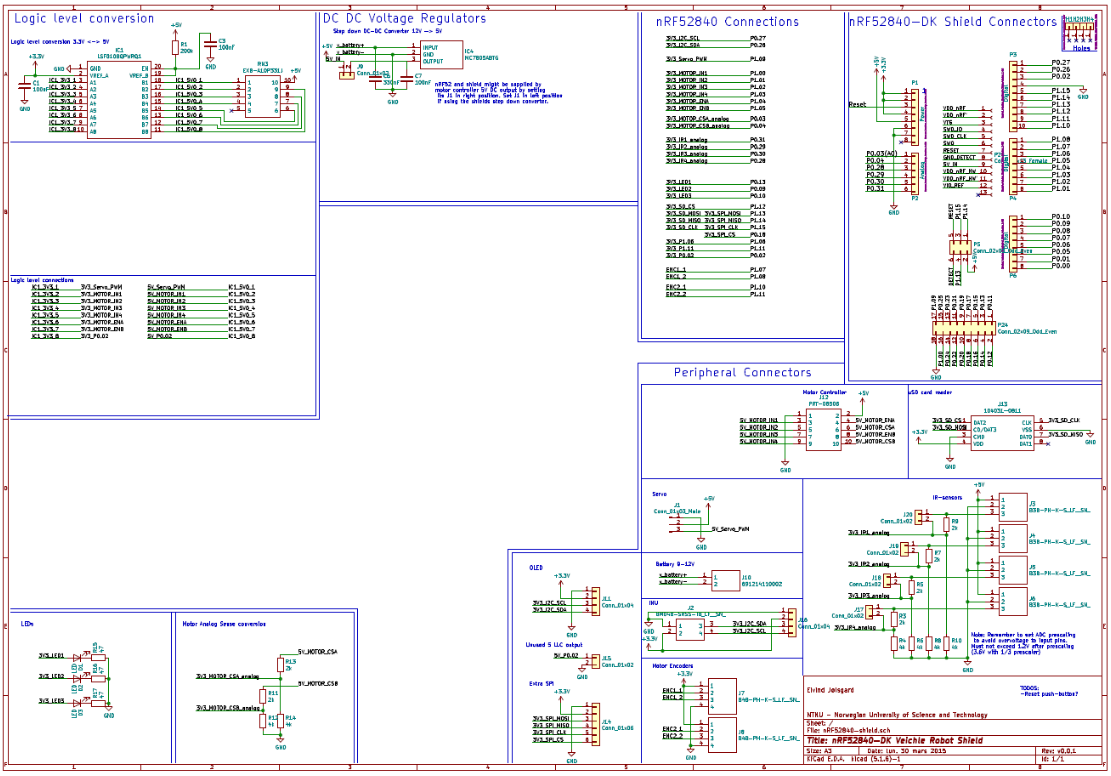


Figure 4: Schematic of shield [2, p.41]

2.2.2 Review of previous work

While the custom shield functions as intended, there is room for improvements when designing a new system. The following points are suggestions for consideration.

- The IR-sensor connectors are difficult to connect and disconnect due to its placement on the board.
- The board cutout for accessing the debug header on the development kit is too small.
- It is difficult to attach and detach the shield from the development kit.
- The connectors on the custom shield are labeled with text describing their purpose, but these labels are often covered by wires from the robot. In order to identify the different connectors, it is necessary to unplug them, which can be difficult due the connectors position.
- The layout of the PCB is not straightforward and requires referencing both the schematics and the layout in order to understand how the shield functions. This can be especially challenging for those on the project who are not familiar with hardware.
- The placement of the various modules in the schematics is not intuitive, making it difficult to understand and navigate.
- The circuits in the schematics are very compact, making it challenging for developers to easily review and understand them.

3 Theory

This chapter will introduce general theory about PCB design and core components for circuit design.

3.1 Printed-Circuit Boards

Printed-circuit boards are epoxy-bonded fiberglass sheets plated with copper. The copper plating is etched away, leaving tracks(traces) that from the interconnections of the circuit [10, p. 140-141]. To design a PCB you will need to use a software program. There are several programs for designing a PCB and the choice often relies on the difficulty of the project, what your company/school uses or just personal preferences.

Usually the programs comes with several tools, allowing schematic entry, netlist generation (a list of what need to be connected to what, PCB layout, manual routing (making the connections), and autorouting. Autorouting simplify the process of generating the PCB as it does the hard work of routing, but one thing to be certain about is that it does not necessarily make the right design choices for your system. Autorouting can sometimes result in routing solutions that are not optimal for signal integrity. Signal integrity refers to the quality of the electrical signals on a PCB, and it can be affected by factors such as the geometry and spacing of the signal lines, and the presence of noise.

When designing a PCB you will need to define the layer stackup. PCBs can either be single-sided (one layer), double-sided (two layers) or 4-layered, 6-layered, 8-layered, 12-layered or more. The more layers you have the easier it is to route your interconnections. It might be tempting to add more layers, but the cost of fabrication go up considerably with extra layers. An advantage of using additional layers dedicated to power and ground planes is that your system will have greater noise immunity.

The five types of objects that can be placed on a copper layer are tracks, individual pads, components (array of pads grouped together), vias and fills.

Tracks are used to interconnect components on a PCB. The thickness of the tracks can vary, and a PCB may have different track widths for different functions. Thicker tracks are capable of carrying more current, but may be more difficult to work with in tight spaces. When deciding on track width, it is important to consider the capabilities and tolerances of the PCB fabrication chosen.

Pads are used to mount component pins on a PCB and can be round, rectangular, or oval in shape. A pad can be multilayer, meaning that it appears on all copper layers with a hole drilled through the entire PCB, or it can appear on only one layer. Figure 5 illustrates the two different types of pads.

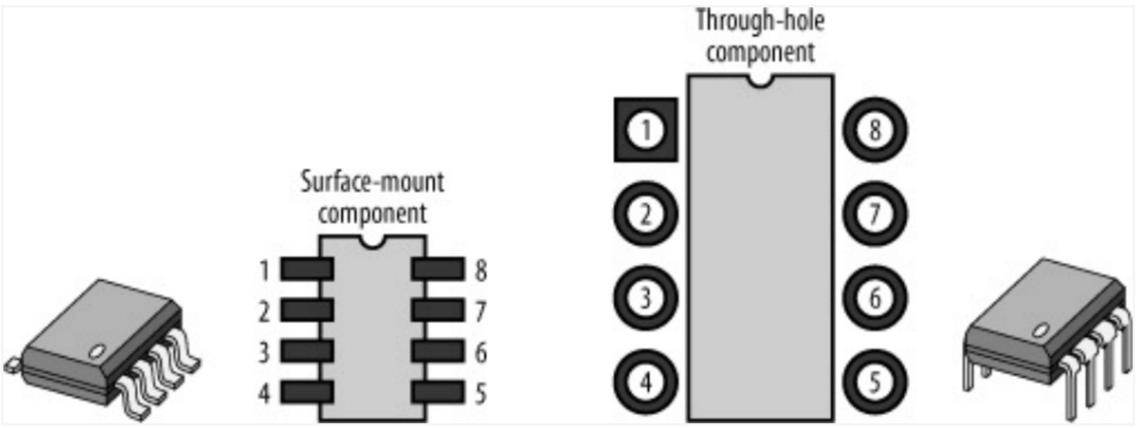


Figure 5: Footprints of surface-mount and through-hole (multilayer) component [10, p.143]

Vias are used to connect tracks on different layers together as shown in figure 6. They can either be through-hole, appearing on all layers, or blind/buried vias, appearing only on the layers they are interconnecting and on intermediate layers.

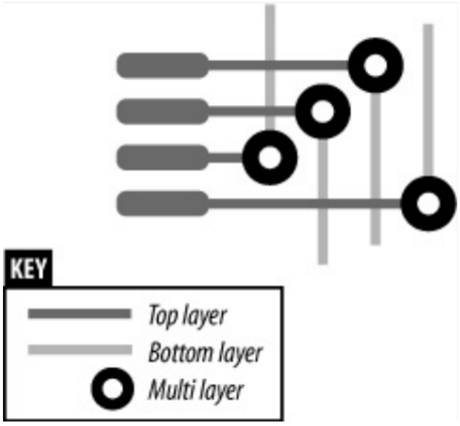


Figure 6: Vias connecting top and bottom layer tracks together [10, p. 145]

Fills are used to provide shielding to certain sections of the PCB, and also for circuit paths that carry a lot of current. It is common to place ground fills in and around analog sections of the circuit to isolate them from digital cross talk.

3.2 Laying out a PCB

Listed below are general things to consider when doing layout for a PCB:

- Tracks should aim directly for the pad center, as shown in figure 7 and not as in figure 8
- Avoid stubs as shown in figure 9a and try to place the components such that the pad lies on the primary signal track as shown in figure 9b
- Keep buses parallel to minimize skew as shown in figure 10. Skew is the difference in delay between the signals on a bus, which can occur due to factors such as differences in track length and track geometry.
- For high-speed signals, make sure there is ground return path close to track so that the current-loop area is minimized.
- In high-speed systems, power and ground planes must be continuous. Must cover the entire PCB with no breaks.
- When routing buses (data or address), keep the tracks running parallel if possible.
- All power and ground traces should be as thick as possible. Power and ground should also be on different layers.

Figure 7: Surface-mount and through-hole pads [10, p. 144]

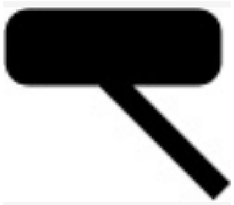
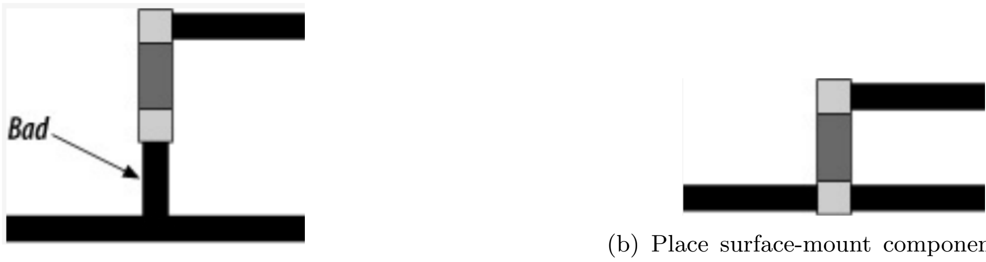


Figure 8: Incorrect way for a track to enter a surface-mount pad [10, p. 144]



(a) Stubs [10, p. 148]

(b) Place surface-mount components directly onto tracks[10, p. 148]

Figure 9: Avoid stubs when routing

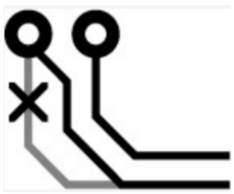


Figure 10: Keep busses parallel to minimize skew[10, p. 148]

3.3 System on Chip (SoC) vs Microcontroller

An SoC, or System on a Chip, is an integrated circuit that contains all the components of a computer or electronic system on a single chip. It is a highly integrated solution that combines the microprocessor, memory, and other peripherals in a compact and cost-effective package.

A microcontroller is a type of computer that is designed to be integrated into small, standalone devices. It is a single chip that includes the microprocessor, memory, and input/output peripherals needed for basic computing functions.

Although an SoC and a microcontroller have some similarities, there are some key differences between the two. An SoC is a more advanced and complex solution that combines all the components of a computer onto a single chip, while a microcontroller typically includes only the essential components. An SoC is often used in applications that require high levels of integration and performance, while a microcontroller is a more basic and cost-effective solution that is suitable for simple computing tasks. Additionally, microcontrollers are often used as components within SoCs.

3.4 Regulators

A voltage regulator is a semiconductor device that converts an input DC voltage to a fixed output DC voltage. They are used to provide a a constant supply voltage within a system. [10, p.112]

The type of regulators that will be introduced are called DC-DC converters. This type of regulator takes an unregulated DC voltage and provide a constant DC voltage output of a fixed value. There are three types of DC-DC regulators:

- linear regulators
  - produce lower voltages than the supply voltage
- switching regulators
  - step up, step down or invert the input voltage
- charge pumps
  - same as the switching regulator but with limited current-drive capability

Linear regulators are small, inexpensive, and have low noise compared to other types of regulators. They function by filtering both the input and output with decoupling capacitors. In contrast, switching regulators, also known as buck or boost regulators, operate by switching a power transistor at their output. These regulators are more efficient than linear regulators as they waste less power during the conversion process. However, they require more external components and generate more noise. Charge pumps, which are not as widely used due to their limited ability to supply voltage, will not be discussed in this chapter.

3.5 Communication protocols

There are many different types of communication protocols that can be used in embedded systems, each with its own set of rules and procedures for transmitting and receiving data. In this project these communication protocols for embedded systems will be introduced:

- SPI(Serial Peripheral Interface)
  - Full-duplex communication protocol that is used for connecting devices to a microcontroller or computer.
- I2C (Inter-Integrated Circuit)
  - Half-duplex communication protocol that is used for connecting multiple devices to a microcontroller or computer using only two wires
- UART (Universal Asynchronous Receiver/Transmitter)
  - Serial communication protocol that is used for transmitting and receiving data asynchronously between devices.
- CAN (Controller Area Network)
  - Message based communication between microcontrollers and devices.

3.5.1 Serial Peripheral Interface (SPI)

Serial Peripheral Interface is a synchronous protocol in which all transmissions are referred to a common clock to synchronize its acquisition of the serial bit stream. SPI provides a simple interface between microcontrollers and peripheral chips such as memory, ADC, DAC, sensors and other processors. It is possible to connect several chips to the same SPI Master. A Master will select a slave to receive by asserting the slave's chip select input. If a peripheral is not selected it will not take part in the SPI communication.

SPI allows for high-speed data transfer with data rates than can range from a few hundred kilobits per second to several megabits per second. This communication protocol is sometimes known as a four-wire interface where it has these signals:

- MOSI - Master out slave in
  - Generated by the master and is received by the slave
- MISO - Master in slave out
  - Produced by the slave, but its generation is controlled by the master.
- SCLK or SCK - Serial clock
- CS - Chip select
  - The chip select to the peripheral is normally generated by using an I/O pin of the master.

3.5.2 Inter-Integrated circuit(I2C)

I2C, also known as Inter-Integrated Circuit uses two wires to connect multiple devices in a multi-drop bus. The bus is bidirectional, low speed, and synchronous to a common clock. Devices may be attached or detached from the I2C bus without affecting other devices. The data rate of I2C is slower than SPI, where the date rate is the range of a few hundred kilobits per second. The two wires used to interconnect with I2C are SDA (Serial Data) and and SCL (Serial Clock).

Each device connected to the I2C bus has a unique address and can operate as either a transmitter (a bus master), a receiver (a bus slave) or both. I2C is a multi-master bus, meaning that more than one device may assume the role of bus master.

3.5.3 Universal Asynchronous Receiver/Transmitter (UART)

In UART communication, data is transmitted and received one bit at a time over a single communication line or channel. The communication protocol allows data transmission without a fixed clock, which means that the sender and receiver do not need to be synchronized. Instead, each device uses a separate clock signal to control the transmission and reception of data. UART is often used for applications that require low-speed data transfer or for debugging and testing purposes.

3.5.4 Controller Area Network (CAN)

CAN is a broadcast-based protocol, meaning that all devices on the network receive all messages, but only the device with the matching identifier will accept and process the message. The communication protocol has a data rate up to 1 megabit per second. It is implemented over a 2-wire serial network and supports multiple masters on the network. Each master is responsible for local sensing and control within the distributed system as seen in figure 11. From here on each master is called a CAN node.

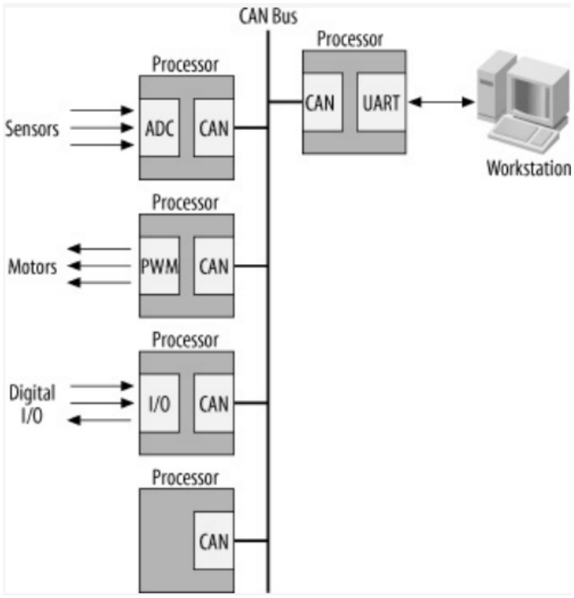


Figure 11: Example on a CAN distributed system[10, p. 332]

The two bus lines are called CAN high (CAN_H) and CAN low (CAN_L). To make a device available to the CAN Bus the CAN node require a processor, a CAN controller and a CAN transceiver.

A CAN controller is a hardware device that facilitates communication between devices on a CAN network by implementing the CAN protocol. It is responsible for sending and receiving CAN messages and often has a register interface that can be accessed by the processor for configuring the controller and sending/receiving messages. Some CAN controllers are integrated into microcontrollers or SoCs.

A CAN transceiver is a hardware device that converts the digital signals from the CAN controller into physical differential signals that can be transmitted over the CAN bus. It is responsible for driving the bus signals and detecting/arbitrating bus errors. The transceiver often has a register interface that can be accessed by the microcontroller for configuring the transceiver and monitor its status.

Together, the CAN controller and transceiver form the hardware interface that allows a device to communicate over a CAN network. The microcontroller uses the register interface to interact with the CAN controller, while the CAN controller and transceiver handle the low-level aspects of sending and receiving messages on the CAN bus.

4 Method

This chapter will cover the various methods used in the development of a printed circuit board. The following steps are typically involved in this process:

1. A schematic is drawn to define the electrical connections and components of the system. This serves as a blueprint for the PCB design.
2. PCB design software is used to create a layout that determines the physical placement of the components on the PCB, as well as the shape and size of the PCB.
3. A test plan is created for testing the PCB.
4. A bill of materials is created, listing all components to be used on the PCB, including their specifications and quantities.
5. The PCB is fabricated using a PCB manufacturing service, based on the design and bill of materials.
6. The PCB is assembled by soldering all components to the PCB according to the design and bill of materials.
7. The PCB is tested to ensure it functions properly and meets the specifications of the design.

Before discussing these steps in more detail, it may be helpful to provide an overview of the system and its core components.

4.1 Breakdown of the system

Before beginning the schematic and layout design phase for a PCB, especially for a new system, it is important to break down the system into its individual components in order to determine the necessary requirements. There are several benefits to breaking down a system into its individual components such as:

Improved understanding: By decomposing a system into its components, it is easier to understand how the various parts work together to achieve the desired functionality.

Enhanced design flexibility: Breaking down a system allows for the individual components to be designed and optimized independently, which can lead to improved performance and increased design flexibility.

Facilitated troubleshooting: If a problem arises during development or testing, having a clear understanding of the individual components can make it easier to identify and fix the issue.

Enhanced modularity: Decomposing a system into its individual components can facilitate the development of modular designs, which can make it easier to maintain and update the system over time.

Improved documentation: Detailing the individual components of a system can improve the quality and clarity of documentation, making it easier for others to understand and work with the design.

The context diagram in figure 12 is a representation of a system that shows the system's relationship to its external environment. The purpose of a context diagram is to provide a clear and concise overview of a system and its interactions with its surroundings.

In comparison to the previous custom shield, the new system includes some parts and excludes others. As shown in figure 4, the mikroUSB connector, connector P20, and connector P5 have been excluded in order to simplify the design. However, the new system includes connectors whose pins are currently being used in the current system. The intention was to focus on including these connectors in the new design.

Most of the peripherals shown in the context diagram are already present on the custom shield, with the exception of the UART. The UART has been added in order to allow for the exploration of using LIDARs in future development. However, the use and development of LIDARs is outside the scope of this project.

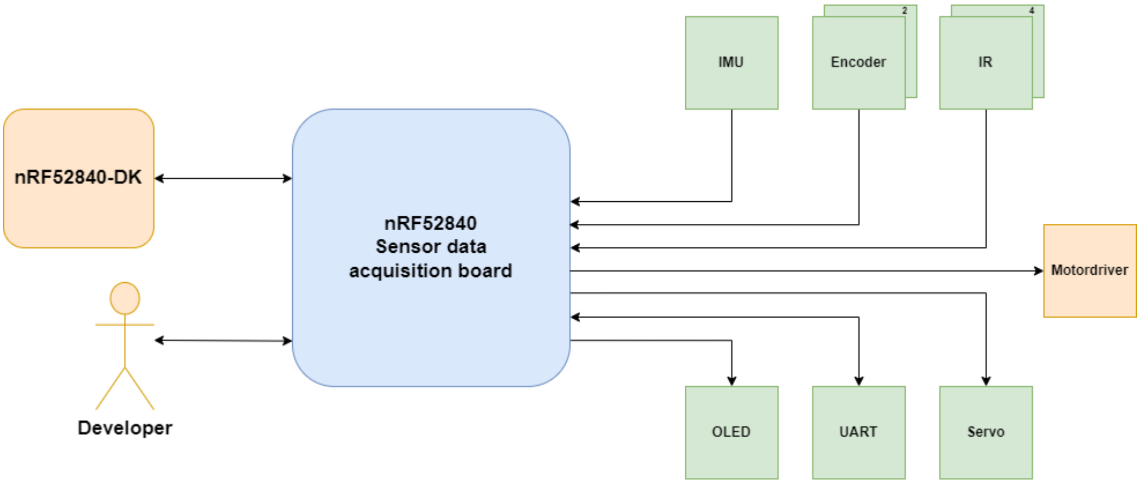


Figure 12: Context diagram of the system

As shown in figure 13, the system structure and the relationships between its various components are illustrated. It is important to note that the communication between the two nRF52840 the SoC on the nRF52840-DK is done using the SPI protocol, but the master and slave roles must be determined in software. The system is designed so that either of the SoCs can be chosen as the master or slave. How power is distributed is not included in this figure. It is therefore important to map out the power requirements for each component, shown in figure 14.

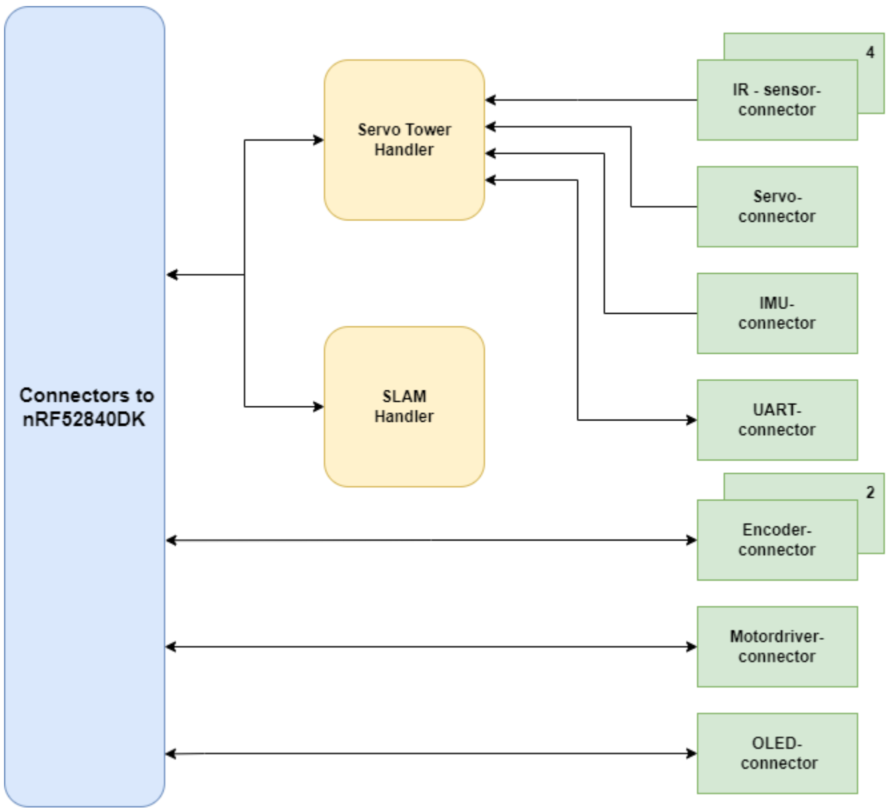


Figure 13: System structure

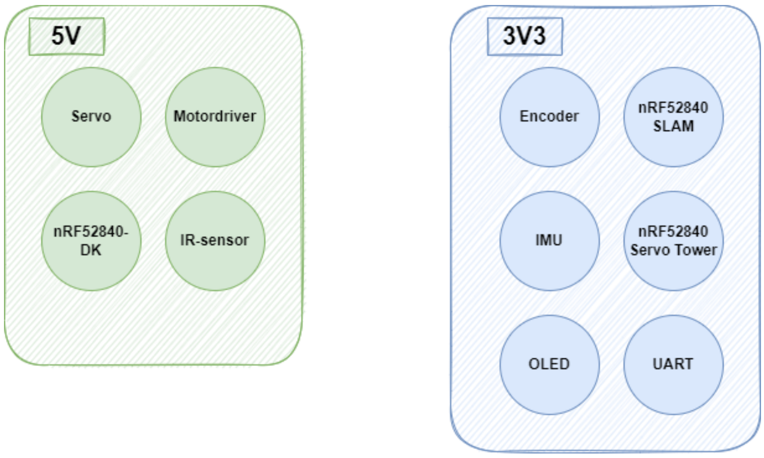


Figure 14: Overview of power requirements

4.2 Choice of components

This design includes three key components: a buck regulator for 5 V, a linear voltage regulator for 3.3 V, and a nRF52840[11] SoC. The SoC was chosen because it has a familiar development environment as the robot project, and the other regulators were chosen to meet the power needs of the peripherals. For voltage regulation, a buck regulator was selected to deliver 5 V to the development kit and several peripherals, while a linear regulator was chosen to provide a stable 3.3 V to the nRF52840 SoC, which does not require a high power draw.

When choosing the buck regulator, table 3 was used to map out the different regulators on the market. Unfortunately, due to the still remaining component crises from the Covid19 pandemic, there were very few buck regulators available. ADP2303[12] shown in figure 15a from Analog devices was the only available and suitable for the design of the system.

MPN	Manufacturer	Input V-range	Output V-range	Max cont.output current
TPS56637	Texas Instruments	4.5 - 28 V	0.5 - 13 V	6 A
TPS5440	Texas Instruments	5.5 - 36 V	Adjustable	5 A
LM22679	Texas Instruments	4.5 - 42 V	Adjustable	5 A
ADP2303A	**Analog Devices**	**3 - 20 V**	**Adjustable**	**3 A**

Table 3: Overview of buck regulators

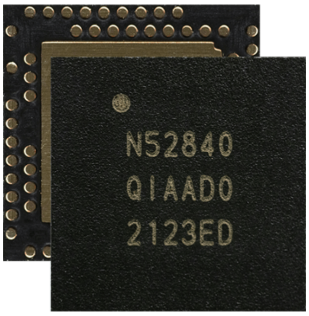
The linear voltage regulator that was chosen for this system was LM3940[13], shown in figure 15b. As stated earlier the component crises made it difficult to find components, but LM3940 was luckily available and in stock. When searching for 3.3 V linear voltage regulators the LM3940 was the only available on the market at the time (September 2022).

(a) Buck regulator ADP2303

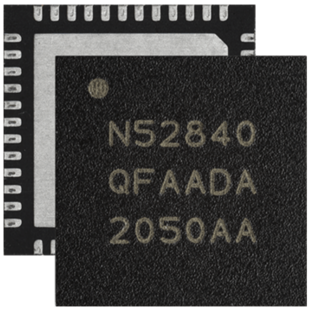
(b) Linear voltage regulator LM3940

Figure 15: 5 V and 3.3 V regulators

The nRF52840 used in this project can be seen in figure 16a. The SoC comes in three different package types: QFN, aQFN and WLCSP. The QFN package, shown in figure 16b is the most suitable package as you can solder the SoC by hand. It was however not available on the market (September 2022) and the aQFN package was chosen.



(a) nRF52840 aQFN



(b) nRF52840 QFN

Figure 16: nRF52840 packages

4.3 Circuit design - Schematic

The program used to draw the schematic was Altium Designer and for this project, four sheets were used compared to the previous project where one sheet was used. The main sheet that covers the overview of the whole system, while the three others cover the internal circuits for power delivery and the two SoCs nRF52840. To facilitate future modification, each module has been isolated from the others. In this context, a module is a circuit with a specific task and has a blue striped box surrounding the circuit. LED-indicators and test points are included with the thought for testing and debugging.

All connectors that connect the system to other peripherals are shown in the main sheet. Connectors that are related with the development kit are placed to the left, while the others are placed to the right. The green boxes, called sheet symbols, represent the three other sheets; the circuit for power, the circuit for Servo Tower nRF52840, and the circuit for SLAM nRF52840. The main schematic can seen in appendix A.

For each circuit around the three key components, their datasheet has been used as a reference. How each circuit is drawn will be further explained below. In addition to the three key components, this system will continue using the same connectors as the custom shield and circuits with regards to the four IR-sensors. The remaining signals that are not transferred to one of the nRF52840 SoCs will remain the same. This means that signals regarding OLED, encoder and motor driver will have the same pin mapping as before.

4.3.1 Regulators

The datasheet for ADP2303 comes with reference circuits for different appliances. For this project's system the input voltage is 12 V and the desired output voltage is 5 V. Figure 17 shows the reference circuit for this application. LM3940 also has its reference circuit shown in figure 18. Both circuits are realized and shown in figure 19.

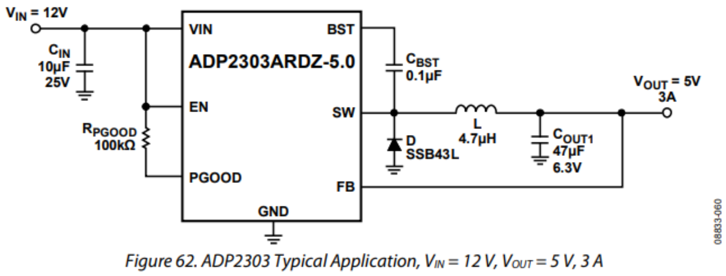


Figure 17: Reference circuit for ADP2303

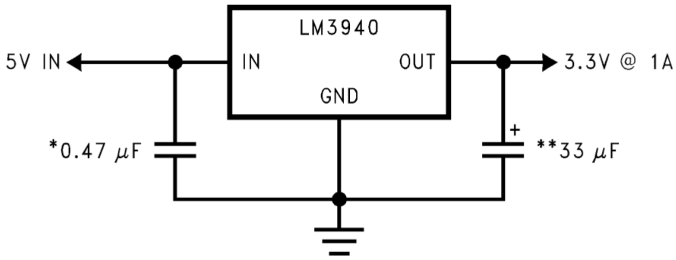


Figure 18: Reference circuit for LM3940

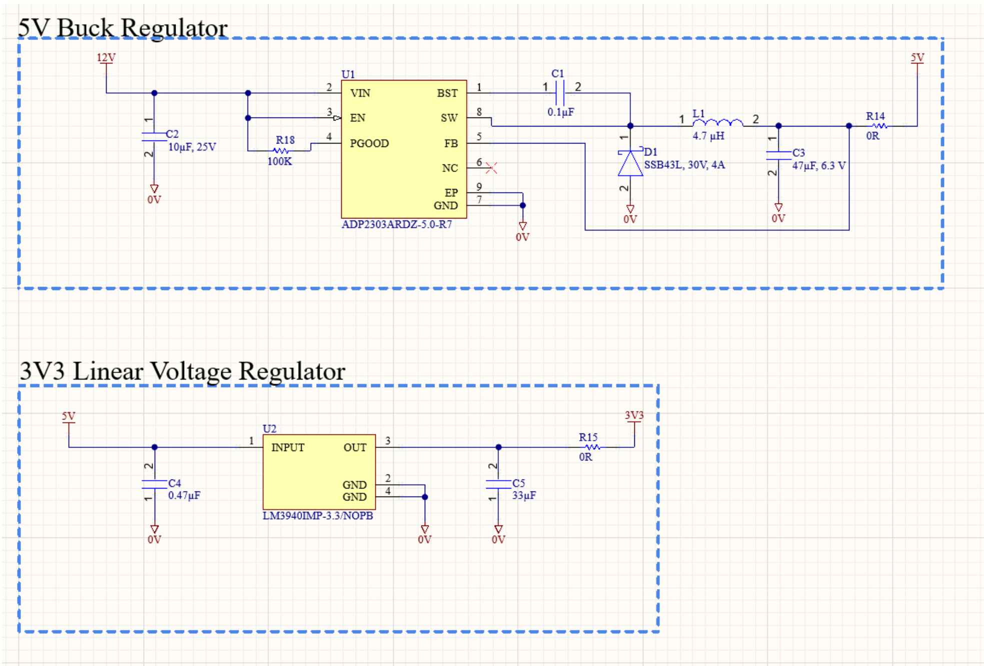
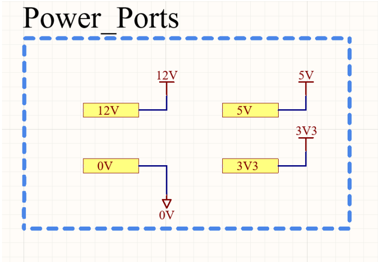


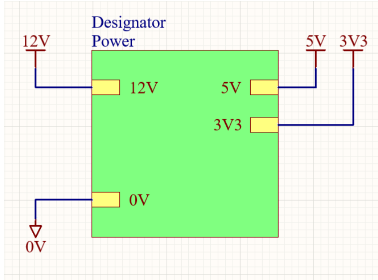
Figure 19: Schematic of power circuit

To connect the sheet with its power circuits to the main sheet, ports must be set. It is similar to defining input and output of a system. The input of this sheet is 12 V and the outputs are 5 V and 3.3 V. Ground is also included as it is wanted to have the same ground for the whole circuit.

Figure 20a shows the ports of the power sheets and figure 20b show how the ports are made available in the main sheet. The sheet symbol connects the ports to the main sheet and 5 V and 3.3 V are now available voltages to the rest of the main sheet. The full schematic of the power sheet is shown in appendices A, B, C and D.



(a) Power ports in power sheet



(b) Sheet symbol for power sheet in main sheet

Figure 20: Connecting sheets with ports

4.3.2 nRF52840 System on Chip

The reference circuit for nRF52840 has been used for both SoCs. Figure 21 shows the reference circuit. The datasheet has up to 6 types of configurations for the SoC, and this project has chosen configuration number 3. This configuration was closest to the projects needs. See figure 22.

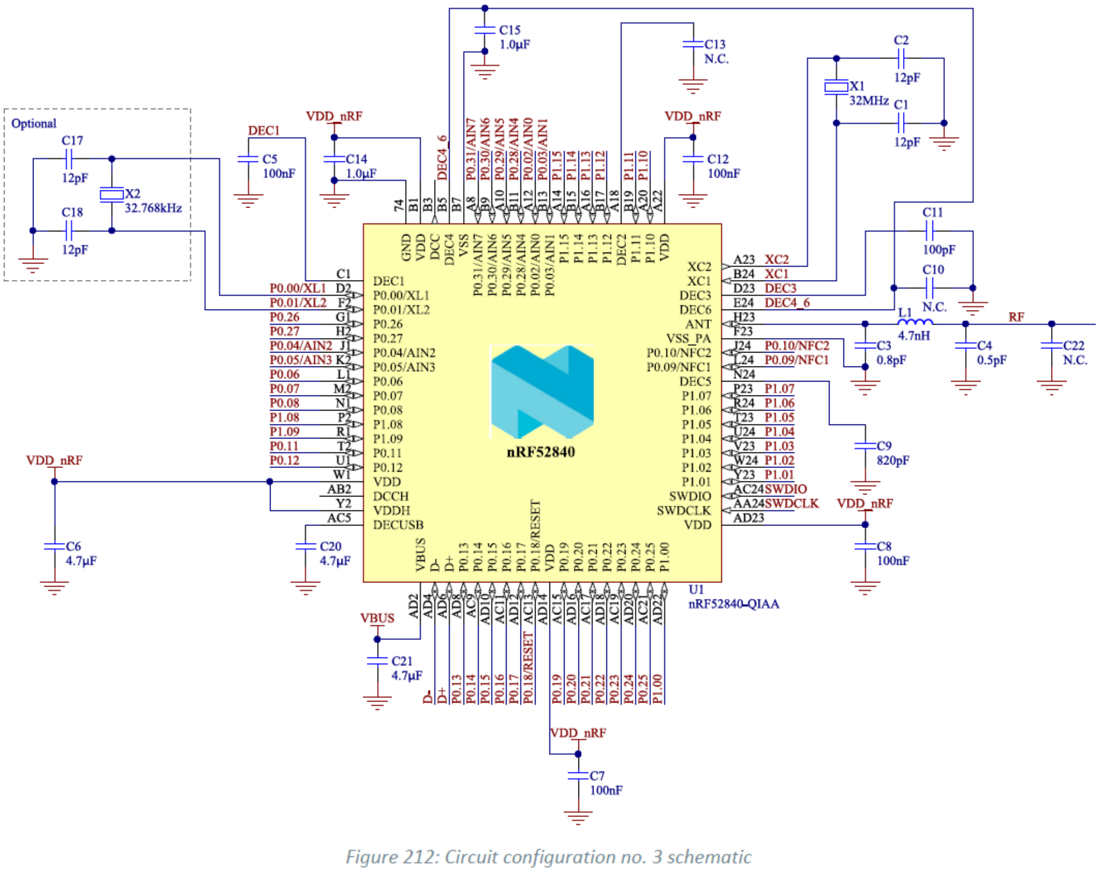


Figure 21: Reference circuit for nRF52840

Config no.	Supply configuration		Enabled features				
	VDDH	VDD	EXTSUPPLY	DCDCEN0	DCDCEN1	USB	NFC
Config. 3	N/A	Battery/Ext. regulator	No	No	No	Yes	No

Figure 22: Circuit configuration

The recommended bill of materials for this project is included in the datasheet and shown in Figure 23. For this project, 0603 package size was chosen for the passive components, even though 0402 package size was recommended. The 0603 package size was selected because the system is in the prototype phase and it is anticipated that most of the components will be hand-soldered during testing and debugging.

Designator	Value v1.0	Value v1.1	Description	Footprint
C1, C2, C17, C18	12 pF		Capacitor, NP0, ±2 %	0402
C3	1 pF	0.8 pF	Capacitor, NP0, ±10 %	0402
C4	1 pF	0.5 pF	Capacitor, NP0, ±10 %	0402
C5, C7, C8, C12	100 nF		Capacitor, X7R, ±10 %	0402
C6, C20	4.7 µF		Capacitor, X7R, ±10 %	0603
C9	820 pF		Capacitor, NP0, ±5 %	0402
C10, C13, C22	N.C.		Not mounted	0402
C11	100 pF		Capacitor, NP0, ±5 %	0402
C14, C15	1.0 µF		Capacitor, X7R, ±10 %	0603
C21	4.7 µF		Capacitor, X7S, ±10 %	0603
L1	3.9 nH	4.7 nH	High frequency chip inductor ±5 %	0402
U1	nRF52840-QIAA		Multi-protocol *Bluetooth*® low energy, IEEE 802.15.4, ANT, and 2.4 GHz proprietary system-on-chip	AQFN-73
X1	32 MHz		XTAL SMD 2016, 32 MHz, Cl=8 pF, Total Tol: ±40 ppm	XTAL_2016
X2	32.768 kHz		XTAL SMD 3215, 32.768 kHz, Cl=9 pF, Total Tol: ±50 ppm	XTAL_3215

Figure 23: Recommended bill of material for nRF52840

Both SoCs on the PCB are similar and follow the reference circuit. The only difference is in how the pins of each SoC are connected to the various peripherals in the system. The nRF52840 dedicated to the servo handler must have available pins for four IR sensors, PWM signals, and communication protocols such as SPI, I2C, and UART. In contrast, the nRF52840 dedicated to SLAM only needs to have available pins for the SPI communication protocol.

Electrical connectivity between component pins can be created by physically placing wire between those pins or created logically by using suitable net identifier, such as net labels. To draw a neatly schematic net labels have been used and can be seen in figure 24.

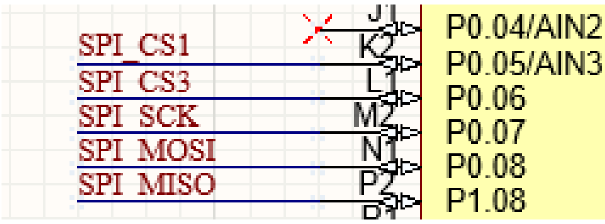


Figure 24: Net label for SPI

The SPI pins must be available to the main sheet and instead of making five ports, one harness entry concerning SPI has been used and can be seen in figure 25.

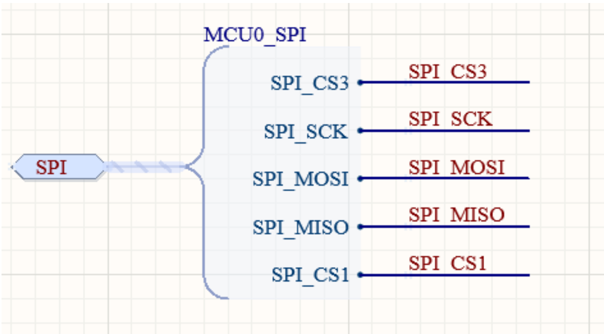


Figure 25: Harness entry for SPI

The harness entry works like a port but instead of making one signal available, it makes all signals connected to the harness available. The SPI harness entry can now be placed in the sheet symbol, making those signals available to the main sheet. Figure 26 shows how SPI is connected and made available between the two sheet symbols. Appendices X and Y show the full schematics for both nRF52840 circuits.

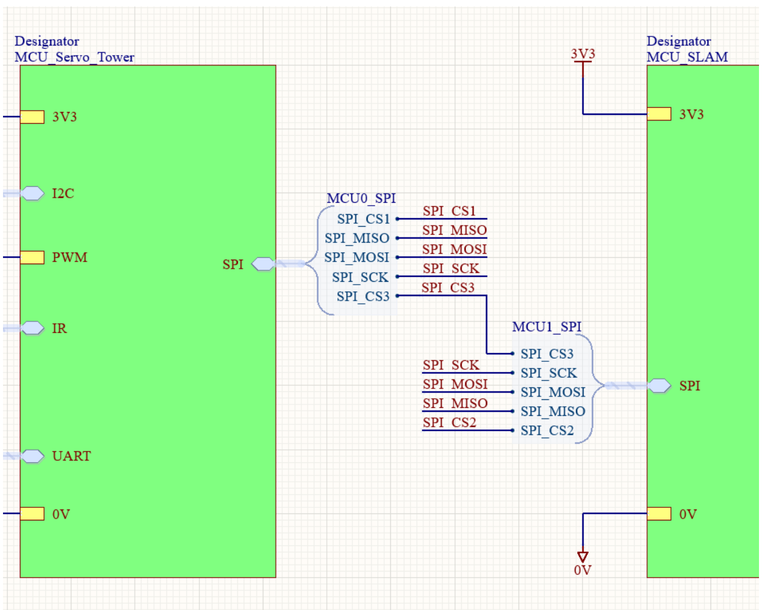


Figure 26: Connection between harness entries in main sheet

4.3.3 Circuits from the custom shield

The connectors used in this system are the same as those used on the custom shield. Figure 27 shows the schematic for the custom shield pin mapping to the nRF52840-DK, and Figure 28 shows the pin mapping for the system to the nRF52840-DK. The numbering of each connector remains consistent, meaning that connector P1 on the custom shield is the same as P1 on the current system.

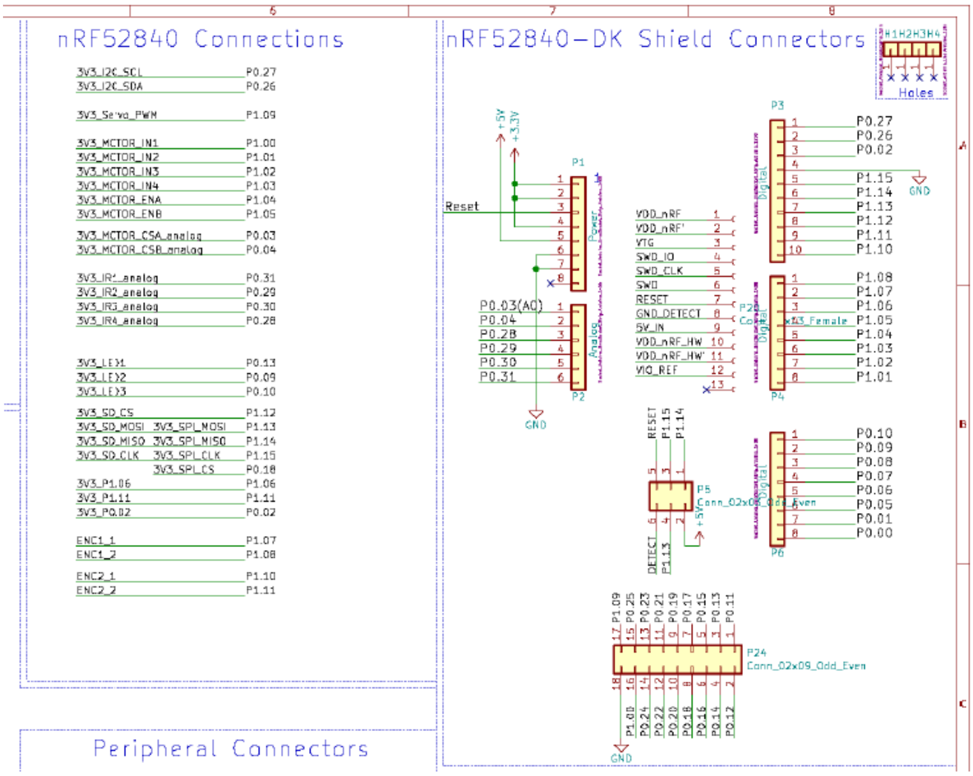


Figure 27: Pin mapping for nRF52840-DK on custom shield

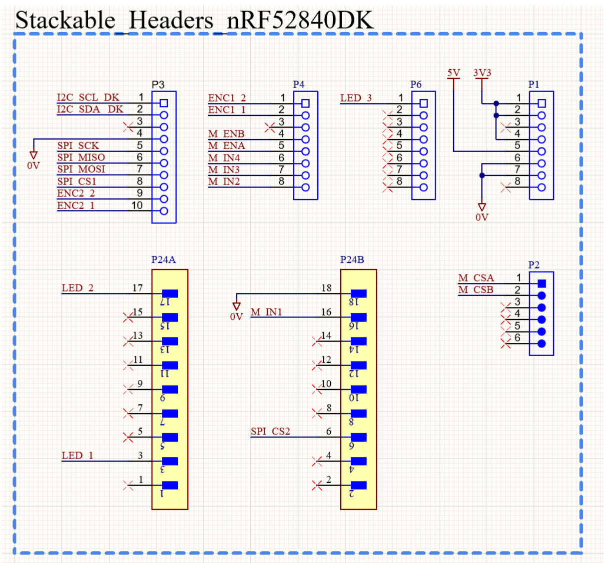


Figure 28: Pin mapping for nRF52840-DK from main sheet

The circuit for the four IR sensors is logically the same, but are modified to isolate each IR sensor for easier viewing. Figure 29 shows the custom shield's circuit for the IR sensors, and Figure 30 shows the redesigned circuit.

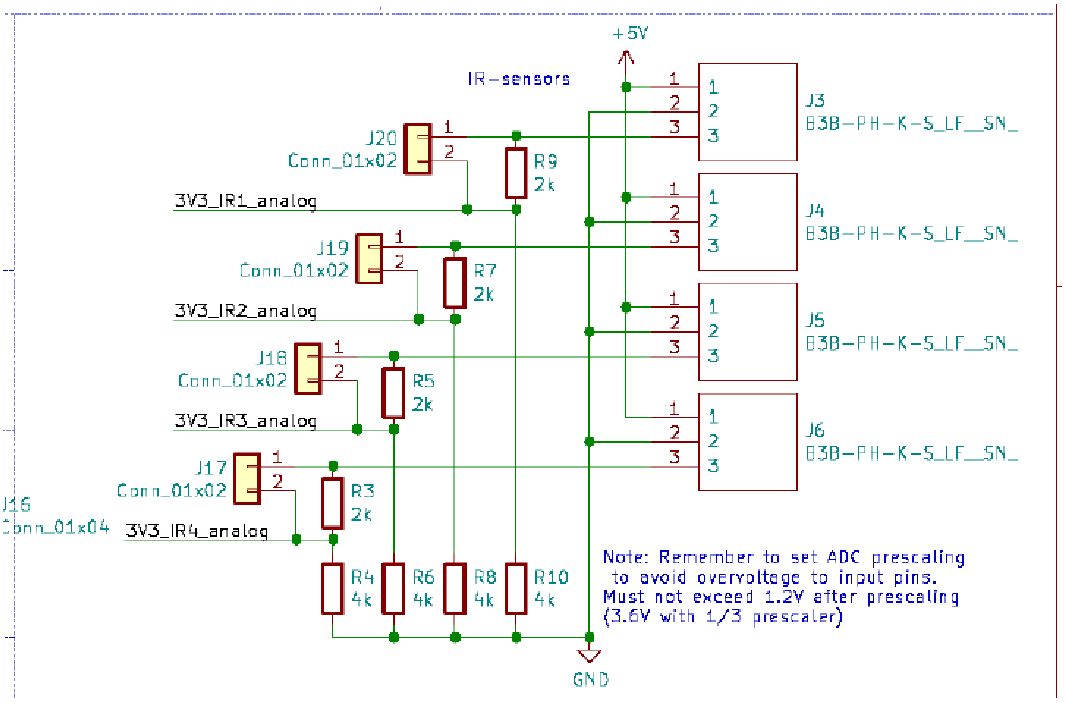


Figure 29: IR-sensor schematic from custom shield

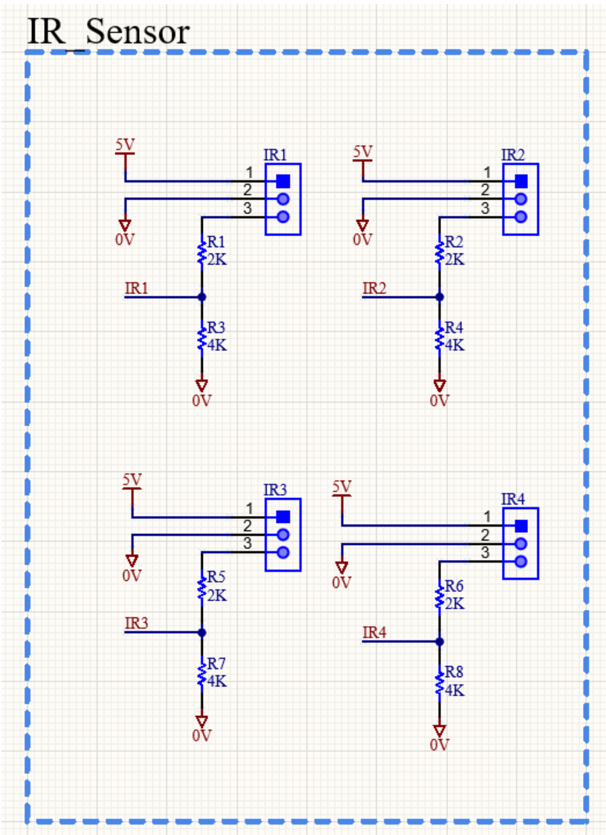


Figure 30: IR-sensor schematic from main sheet

4.4 Circuit design - Layout

In contrast to the custom shield, the layout for this PCB has been done manually, due to the complexity of the PCB which includes layout around two SoCs. Manually designing the layout can be very time-consuming, so careful planning has been prioritized. The components on the PCB have been divided into individual modules or parts, such as the circuit for converting 12 V to 5 V being its own module. After the components in each module have been placed, the location of the modules is determined. The maximum size of the PCB board shape, which is 130 mm x 67 mm as shown in Figure 31, has been taken into consideration when placing the modules. Cutouts have also been added to allow access to the nRF52840-DK debugging connectors.

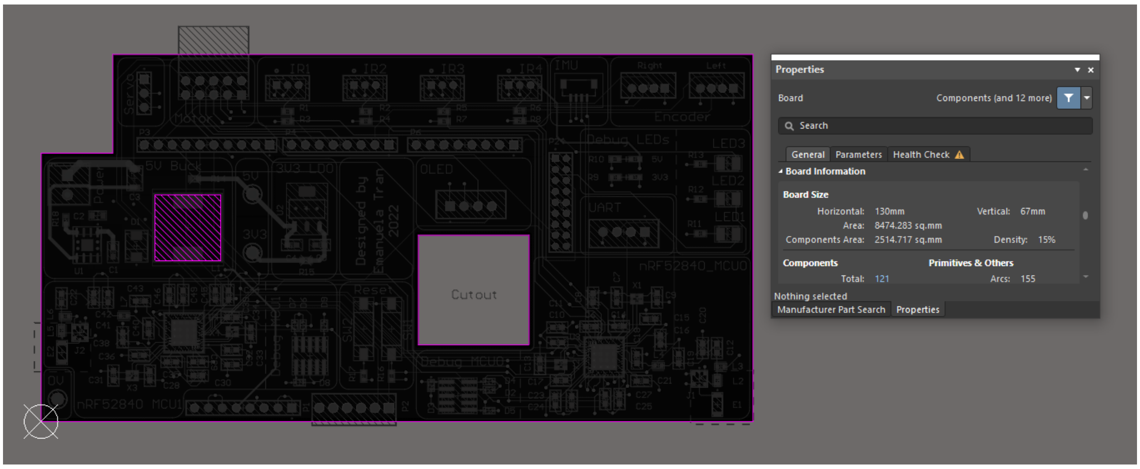
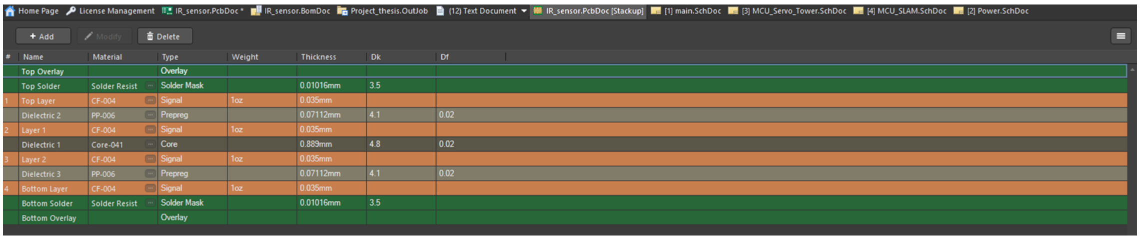


Figure 31: Board shape

The layer stack-up is set to be four layers with 1 oz copper thickness, as shown in figure 32. The four layers are the top layer, layer 1, layer 2, and bottom layer. Layer 1 serves as the ground layer, while layer 2 serves as the power layer. The addition of two layers was necessary for the layout around the nRF52840. The nRF52840 datasheet includes recommended layout design guidelines, which utilize four layers and can be seen in figure 43.



#	Name	Material	Type	Weight	Thickness	Dk	Df
	Top Overlay		Overlay				
	Top Solder	Solder Resist	Solder Mask		0.01016mm	3.5	
1	Top Layer	CF-004	Signal	1oz	0.035mm		
	Dielectric 2	PP-006	Prepreg		0.07112mm	4.1	0.02
2	Layer 1	CF-004	Signal	1oz	0.035mm		
	Dielectric 1	Core-041	Core		0.889mm	4.8	0.02
3	Layer 2	CF-004	Signal	1oz	0.035mm		
	Dielectric 3	PP-006	Prepreg		0.07112mm	4.1	0.02
4	Bottom Layer	CF-004	Signal	1oz	0.035mm		
	Bottom Solder	Solder Resist	Solder Mask		0.01016mm	3.5	
	Bottom Overlay		Overlay				

Figure 32: Layer stackup

For layer 1, fills have been used to cover almost the entire board with a ground plane, while fills have been used on layer 2 to create planes for 5 V and 3.3 V. The placement of each module has been arranged with consideration for the power planes. In figure 33, the power plane is divided such that most components using 5 V are located at the top of the board, while components using 3.3 V are at the bottom part. Figure 34 shows how layer 1 is covered with fills for the ground plane. If observed closely, some routing has been done on both layers, but the tracks have been kept minimal and as short as possible to reduce the risk of cuts in the planes. Layer 1 is not fully covered with the ground fills as some components require the ground fills not to surround them.

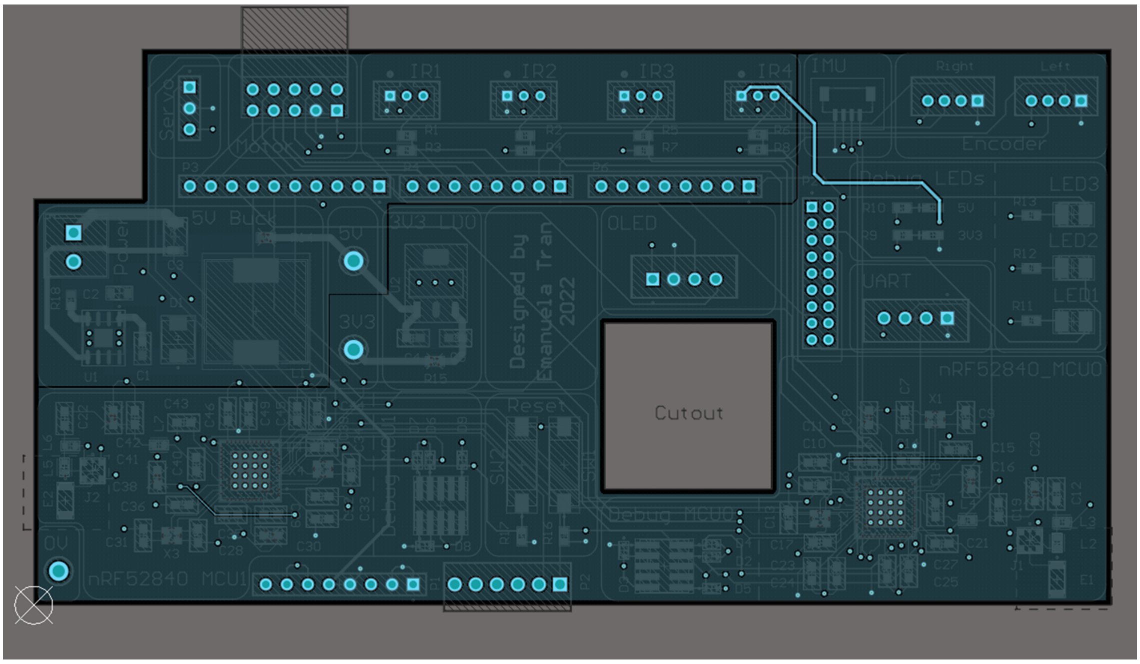


Figure 33: Layer 2 power

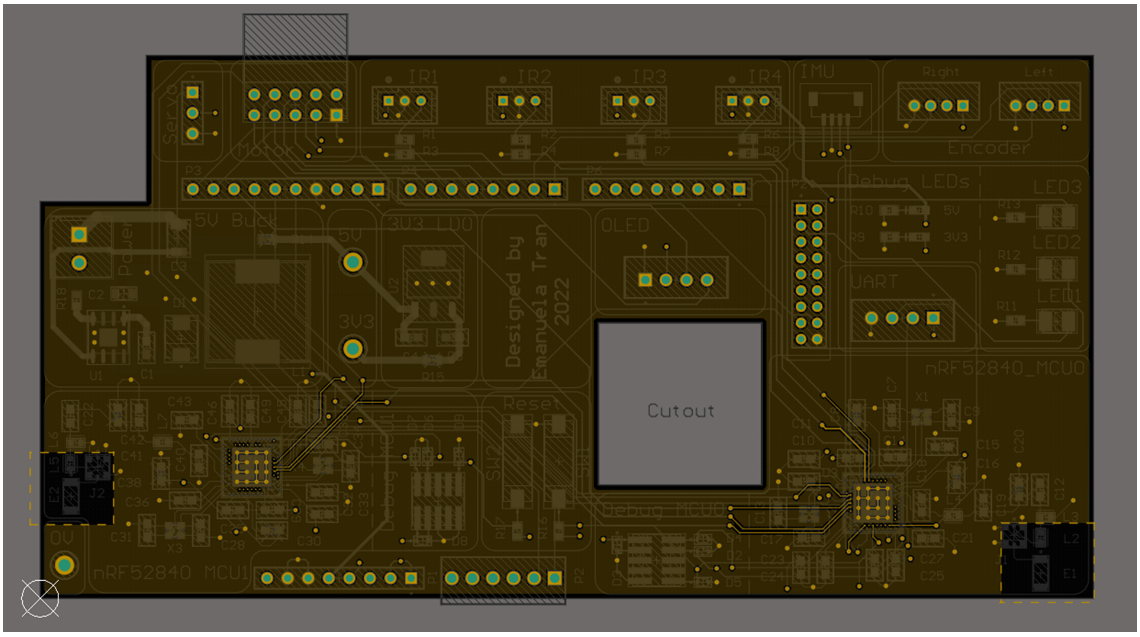


Figure 34: Layer 1 ground

Figure 35 and 36 show the layout for the top and bottom layers. Here the tracks are routed directly from the pads and busses have been routed parallel to the best degree. The minimum track width is set to be 0.15 mm.

The remainder of this section will provide information on how the layout has been specifically designed for certain components, such as the regulators and the nRF52840.

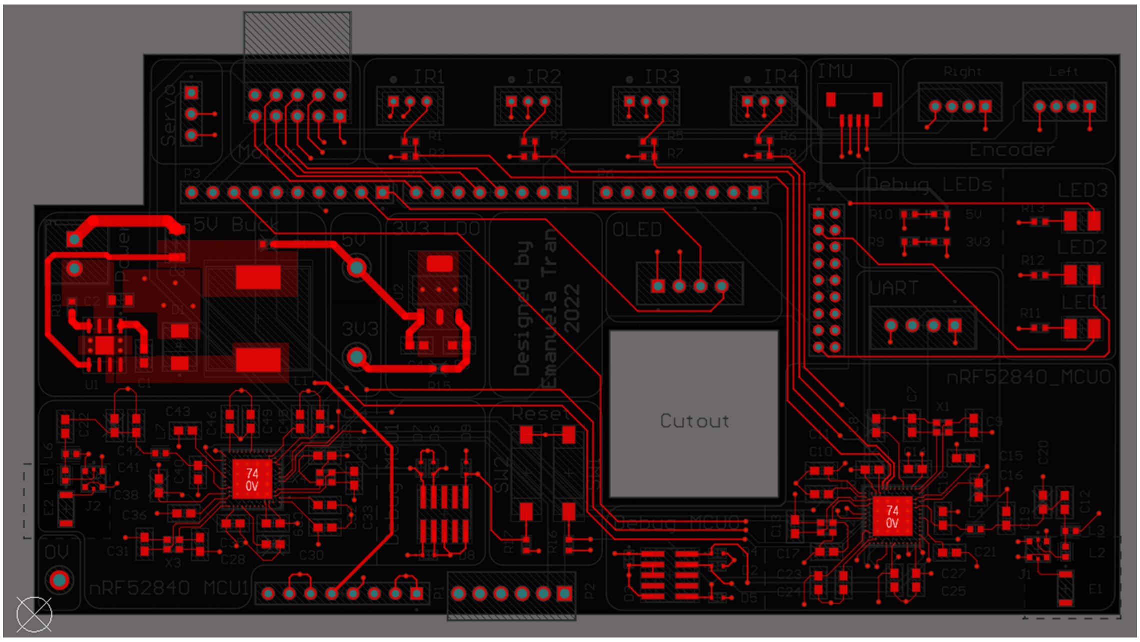


Figure 35: Top Layer

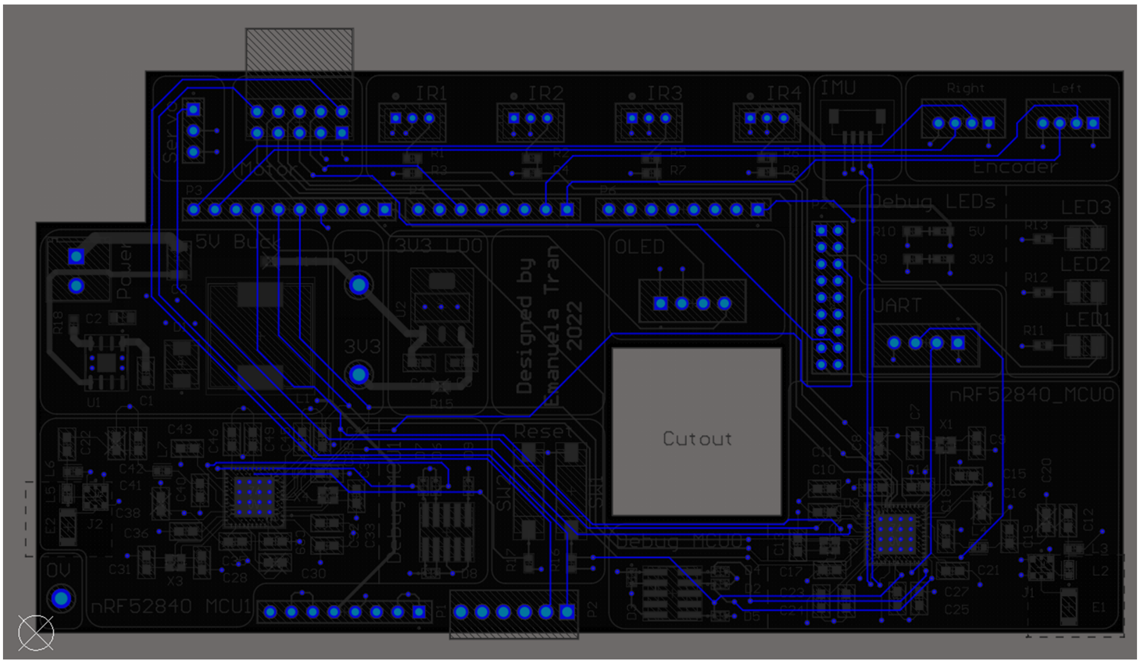


Figure 36: Bottom layer

4.4.1 Regulators

The layout for the ADP2303 and LM3940 regulators has been designed based on the recommended layout provided in their respective datasheets. Figure 37 shows the recommended layout for the ADP2303, while figure 38 shows the attempted layout. Fills have been used where possible or tracks have been widened.

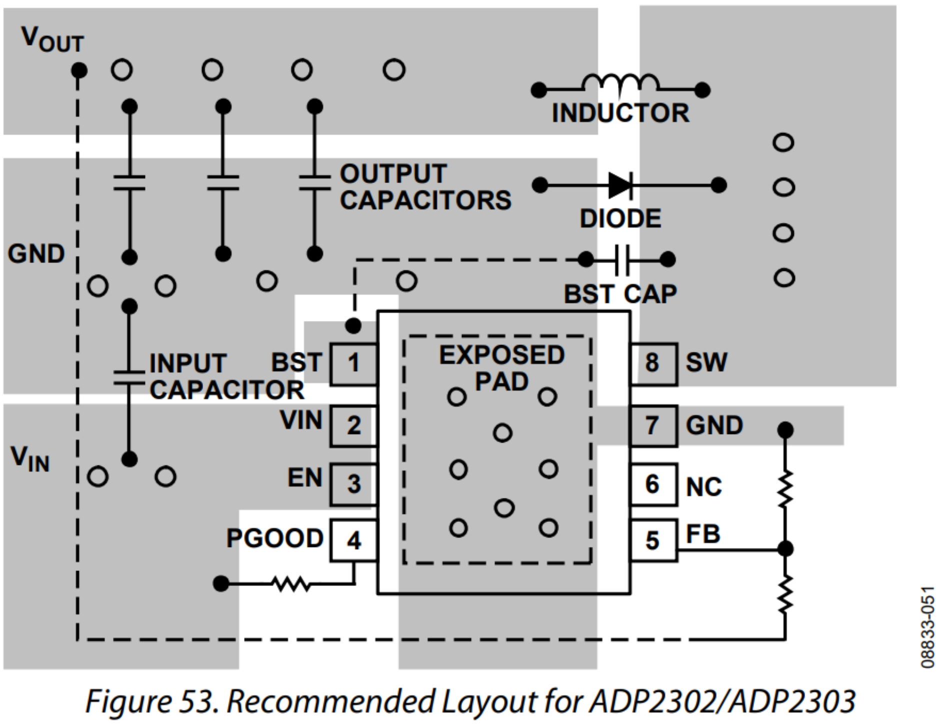


Figure 37: Recommended layout for ADP2303

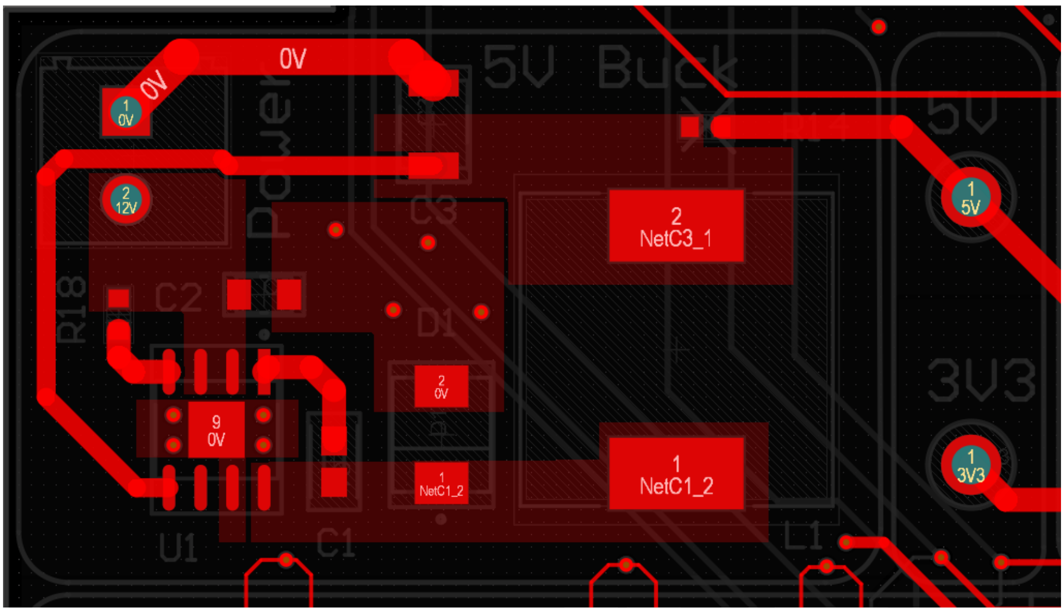


Figure 38: Layout around ADP2303

Figure 39 shows the recommended layout for the LM3940 and figure 40 shows the attempted layout.

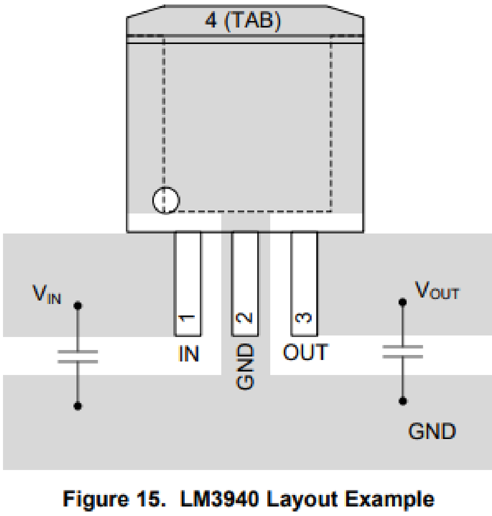


Figure 39: Recommended layout for LM3950

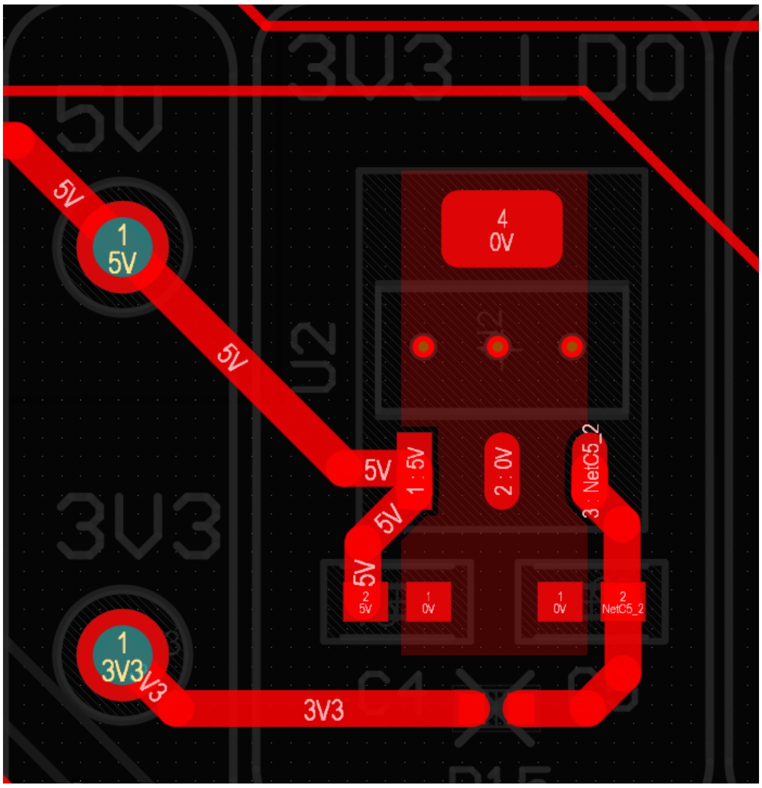


Figure 40: Layout around LM3940

4.4.2 nRF52840

Routing around the nRF52840 can be challenging if multiple layers are not included. Figure 41 shows the layout for the SoC dedicated to the Servo tower. All layers are included and can be seen in this figure. The nRF52840 datasheet recommends using the top and ground layers for the pins of the SoC, but the bottom layer has also been used in this design.

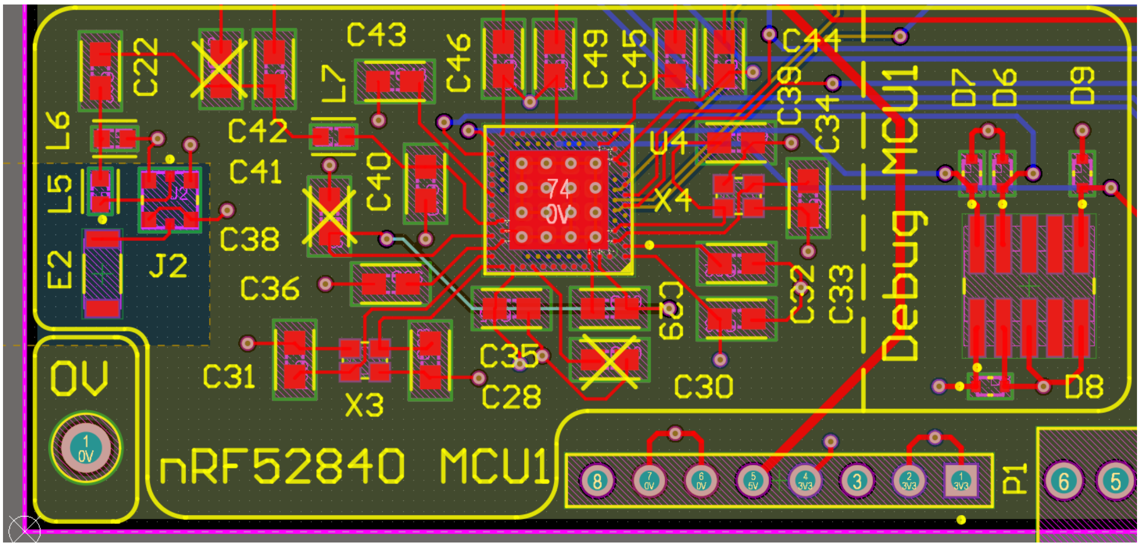
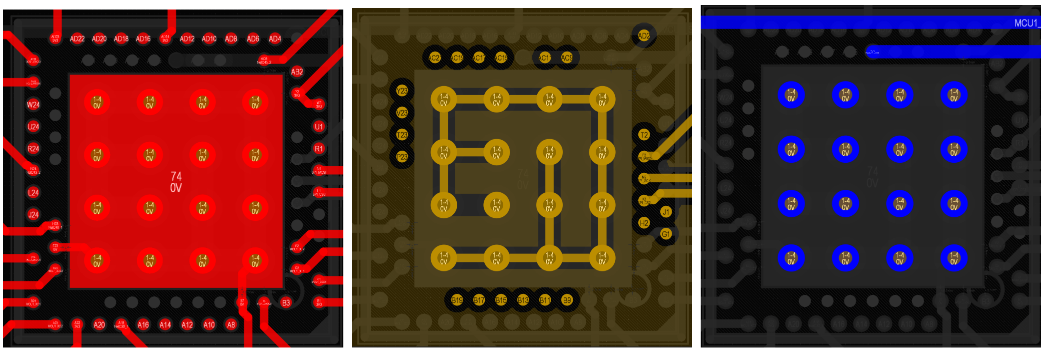


Figure 41: Layout for nRF52840 where all layers are visible

The layout follows the recommendations from the datasheet as closely as possible, with a few modifications. The datasheet states that the minimum recommended trace width is 0.145mm, and in order to achieve the recommended layout, this trace width is necessary for routing around the nRF52840. It is important to note that the recommended layout in the datasheet utilizes all pins, while this system does not.

Most of the traces around the SoC have the width 0.2mm, where the thinnest track width that has been used are 0.15mm. Figure 42 shows how the pins for the nRF52840 has been assigned to the different layers. The reccomended layout for the nRF52840 from the datasheet can se seen in figure 43.

Most of the traces around the SoC have a width of 0.2mm, with the thinnest track width being 0.15mm. Figure 42 shows how the pins for the nRF52840 have been assigned to different layers. The recommended layout for the nRF52840 from the datasheet can be seen in figure 43



(a) Top layer (b) Ground layer (c) Bottom layer

Figure 42: Pin assigment around the nRF52840

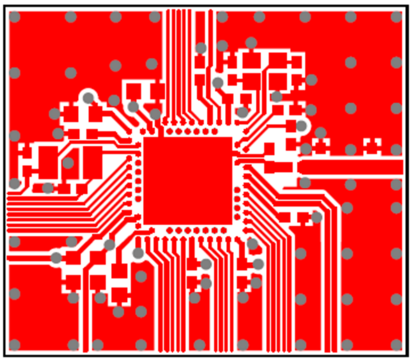


Figure 223: Top layer

(a) Top layer

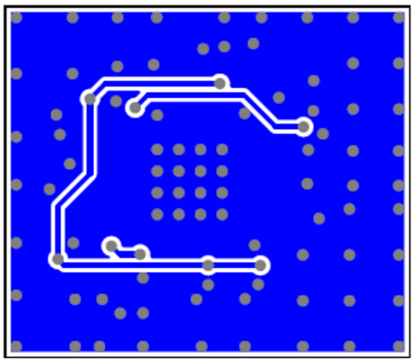


Figure 226: Bottom layer

(b) Bottom layer

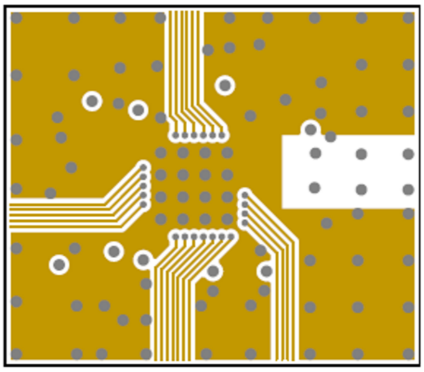


Figure 224: Mid layer 1

(c) Layer 1: Ground

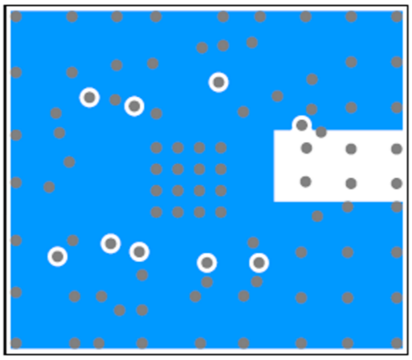


Figure 225: Mid layer 2

(d) Layer 2: Power

Figure 43: Recommended layout for nRF52840

During routing around the nRF52840, care has been taken to route away from sensitive components such as the crystal X4, which is one of the external clocks for the nRF52840. This can be seen in Figure 44, where most of the blue lines are located above the crystal X4.

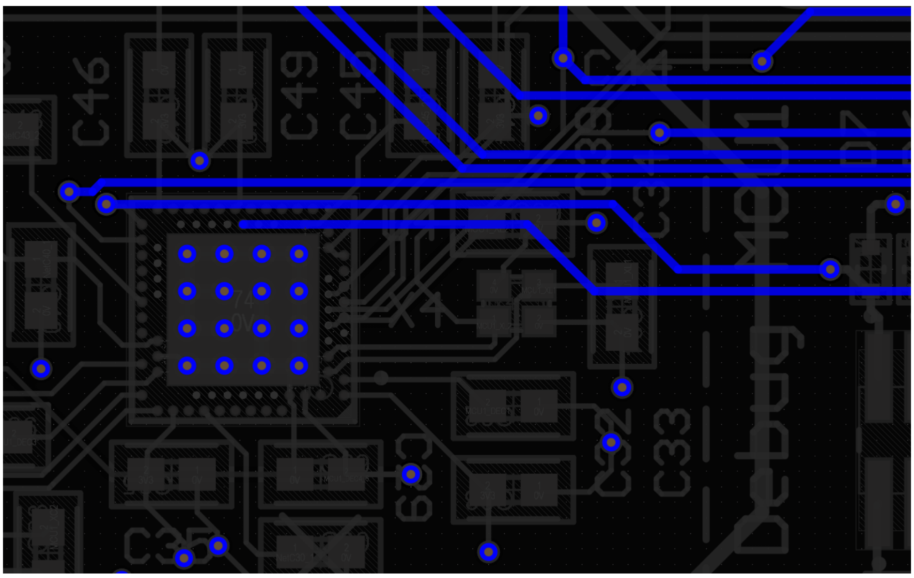


Figure 44: Signals have been routed away from sensitive components

4.4.3 Silkscreen

After completing the routing between the components, the silkscreen design was created. The silkscreen is a text layer that can be located on the top or bottom of a PCB and is commonly used for annotation. Figure 45 shows the silkscreen for the PCB. A cross has been added over components that should not be soldered. Boxes are used to separate all the modules, and a descriptive title has been added for each box.

The purpose of the silkscreen for this PCB is to provide students working on the project with a clear overview of the system and its modules. It is also designed to facilitate hand soldering and testing. If debugging the buck regulator becomes necessary, the silkscreen will show the placement of all its components and provide a starting point for debugging, as they are all located within the 5 V buck regulator box.

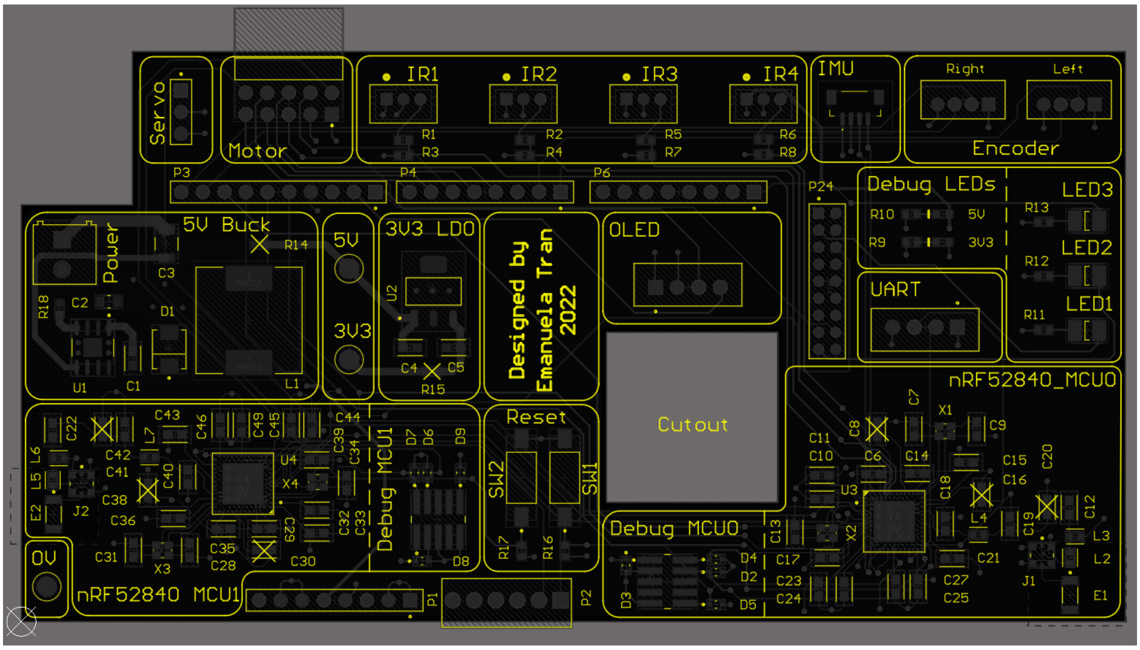


Figure 45: Silkscreen

4.5 Test plan

A test plan was developed to outline the testing that needs to be done and in what order. The plan was to divide the testing of the PCB into six parts, as shown in Figure 46. It is important to note that some tests cannot be started until others have been successfully completed. For example, the test for the 3.3 V linear regulator cannot be initiated until the test for the 5 V buck regulator has been passed, as it relies on a 5 V supply. The same is true for the tests labeled MCU0 and MCU1. However, if the MCU0 test fails, it is still possible to start the MCU1 test, as they do not depend on each other.

The test plan also includes sections as: how, equipment, comments and pass/fail. The how-section is a description on what needs to be done before you test, what you are testing and how you wrap up the test. The equipment-section tell you all equipments needed for you test. The comment-section are for documenting while testing. This section can be very important and used later when developing a new version of the PCB.

The test plan also includes sections for: how, equipment, comments, and pass/fail. The how section includes a description of what needs to be done before the test, what is being tested, and how to conclude the test. The equipment section lists all the equipment needed for the test. The comments section is for recording observations during the testing process. This section can be very important and can be used later when developing a new version of the PCB.

What	How	Equipment	Comments	Pass/Fail
5 V buck regulator	Solder on components for the 5V buck regulator. Deliver 12 V to input. Measure stable 5 V from output. Solder on 5V jumper.	Soldering Iron, microscope, solder, flux, multimeter		
3v3 linear regulator	Solder on components for the 3v3 regulator. Deliver 12 to input. Measure stable 3.3V from output. Solder on 3V3 jumper.	Soldering Iron, microscope, solder, flux, multimeter		
MCU0	Solder on components for MCU0. Check for shortcircuit around the MCU. Connect to MCU0 with debugger.	Soldering Iron, microscope, solder, flux, multimeter, nRF debugger		
Program MCU0	Turn on one LED, set high from MCU0	computer, nRF debugger		
MCU1	Solder on components for MCU1. Check for shortcircuit around the MC1. Connect to MCU1 with debugger.	Soldering Iron, microscope, solder, flux, multimeter, nRF debugger		
Program MCU1	Turn on one LED, set high from MCU1	computer, nRF debugger		

Figure 46: Test plan

4.6 Bill of material

A bill of materials has been created to specify the components that will be used on the printed circuit board. This list includes the quantities and specifications for each component, as well as the vendor where it can be purchased. The BOM is divided into five appendices, identified as E, F, G, H and I. The rows in the BOM have been color-coded to indicate the vendor: yellow for Mouser, blue for JLCPCB, orange for RS Components, green for Kjell & Company, red for components related to antenna design, and white (uncolored) for components that could not be found.

4.7 Production and assembly

JLCPCB was selected as the manufacturer for the PCB. Gerber files, a type of file that contains the necessary information to produce a PCB, were created using Altium Designer. In addition to producing PCBs, JLCPCB offers assembly services if they have the necessary components in stock. The nRF52840 chosen for this PCB requires assembly at a production site due to its package type, which cannot be hand-soldered.

JLCPCB maintains a library of assembly parts on their website, which includes all of the components they offer and assigns each component its own JLCPCB part number. When ordering a PCB and requesting assembly, a BOM list must be provided in a specific format and must include the JLCPCB part numbers for all components. Figure 47 shows the BOM in the format required by JLCPCB. Note that the full BOM from earlier, contained in appendices E through I, is not included because some of the components were not available from JLCPCB. This was checked in advance by searching the assembly part library.

Comment	Designator	Footprint	JLCPCB Part #
2K	R1,R2,R5,R6	R0603	
4K	R3,R4,R7,R8	R0603	
4.7uF X7R 10%	C23,C24	C0603	
4.7uF X7R 10%	C44,C45	C0603	
100nF X7R 10%	C27,C25	C0603	
100nF X7R 10%	C10,C14	C0603	
100nF X7R 10%	C49,C46	C0603	
100nF X7R 10%	C32,C35	C0603	
820pF NP0 +5%	C21,C43	C0603	
0.8pF NP0 +10%	C18,C40	C0603	
0.5pF NP0 +10%	C19,C41	C0603	
100pF X7R +10%	C15,C36	C0603	
1.0uF X7R 10%	C6,C11	C0603	
1.0uF X7R 10%	C29,C33	C0603	
TVS DIODE	D2,D3,D4,D5	SOD523	C552514
TVS DIODE	D6,D7,D8,D9	SOD523	C552514
2.2pF	C12,C22	C0603	
3,9nH	L2,L3,L5,L6	L0603	
10K	R16,R17	R0603	
nRF52840-QIAA	U3	AQFN-73	C190794
nRF52840-QIAA	U4	AQFN-73	C190794

Figure 47: Bill of material list for JLCPCB

To ensure that JLCPCB assembles the components correctly, a list of all the components that will be soldered on the PCB must be created, including their orientation and placement. This list can be seen in figure 48. The components that were not available from JLCPCB were ordered from vendors and were soldered manually.

Designator	Mid X	Mid Y	Layer	Rotation
U3	102.79mm	11.77mm	Top	0
U4	26.1mm	16.2mm	Top	180
R16	64mm	8.3mm	Top	270
R17	58.9mm	8.3mm	Top	270
L2	123.5mm	7.5mm	Top	270
L3	124.1mm	10mm	Top	270
L5	3.7mm	16.626mm	Top	270
L6	4.4mm	19.2mm	Top	0
C12	123.4mm	13.4mm	Top	270
C22	3.7mm	22.5mm	Top	270
D6	47.8mm	17.5mm	Top	90
D7	46.3mm	17.5mm	Top	90
D8	47.2mm	7.1mm	Top	0
D9	51.7mm	17.5mm	Top	90
D2	81.8mm	5.8mm	Top	0
D3	71.2mm	6.6mm	Top	270
D4	81.8mm	7.3mm	Top	0
D5	81.8mm	2mm	Top	0
C29	28.6mm	10.9mm	Top	0
C33	34.8mm	10.4mm	Top	180
C6	100.327mm	17.327mm	Top	180
C11	94.274mm	17.2mm	Top	0
C15	111.274mm	18.7mm	Top	0
C36	17.9mm	12.1mm	Top	0
C19	116.4mm	11.5mm	Top	270
C41	12.2mm	22.6mm	Top	270
C18	108.8mm	11.5mm	Top	90
C40	19.6mm	17mm	Top	270
C21	109.6mm	7.4mm	Top	180
C43	18.1mm	22mm	Top	0
C32	34.8mm	13.1mm	Top	180
C35	23.8mm	10.9mm	Top	180
C49	25.9mm	23.1mm	Top	270
C46	23.4mm	23.1mm	Top	270
C10	94.274mm	14.4mm	Top	0
C14	105.5mm	17.4mm	Top	0
C27	103mm	4.1mm	Top	90
C25	105.6mm	4.1mm	Top	90
C44	34.2mm	23.1mm	Top	270
C45	31.8mm	23.1mm	Top	270
C23	93.9mm	3.8mm	Top	90
C24	96.9mm	3.8mm	Top	90
R3	45mm	54.8mm	Top	180
R4	59.3mm	54.8mm	Top	180
R7	73.6mm	54.8mm	Top	180
R8	87.4mm	54.9mm	Top	180
R1	45mm	56.6mm	Top	0
R2	59.3mm	56.6mm	Top	0
R5	73.55mm	56.6mm	Top	0
R6	87.4mm	87.4mm	Top	0

Figure 48: Component orientation list for JLCPCB

The test plan, introduced in section 4.5, was used as a reference to determine the order in which the components should be soldered. Due to the size of certain components, it was necessary to solder some connectors or headers later, as it was difficult to maneuver the soldering iron between them. For example, there is less space between the headers that connect the PCB to the development kit and the circuits for voltage regulation, so these headers were soldered later. This also applied to other components. The inductor L1 and power connector, for example, were not soldered first because they are larger and taller than the other components in the circuits for the 5 V buck regulator.

The tools used for assembling the PCB where:

- Solder - metal to connect component to copper on the PCB.
- Soldering iron - tool to melt the solder.
- Microscope - to solder SMD components as they are small and difficult to solder.
- Tweezer - tool to hold components in place.
- Fume extractor - tool to extract the fumes that are dangerous and toxic while soldering.
- Flux - tool to prepare the metal surfaces for soldering by cleaning and removing any oxides and impurities.
- ESD-band - tool to disperse static electricity generated from a person safely to ground.
- Schematics and layout of PCB - for reference when soldering components .

Figure 49 shows the setup for assembling the PCB.

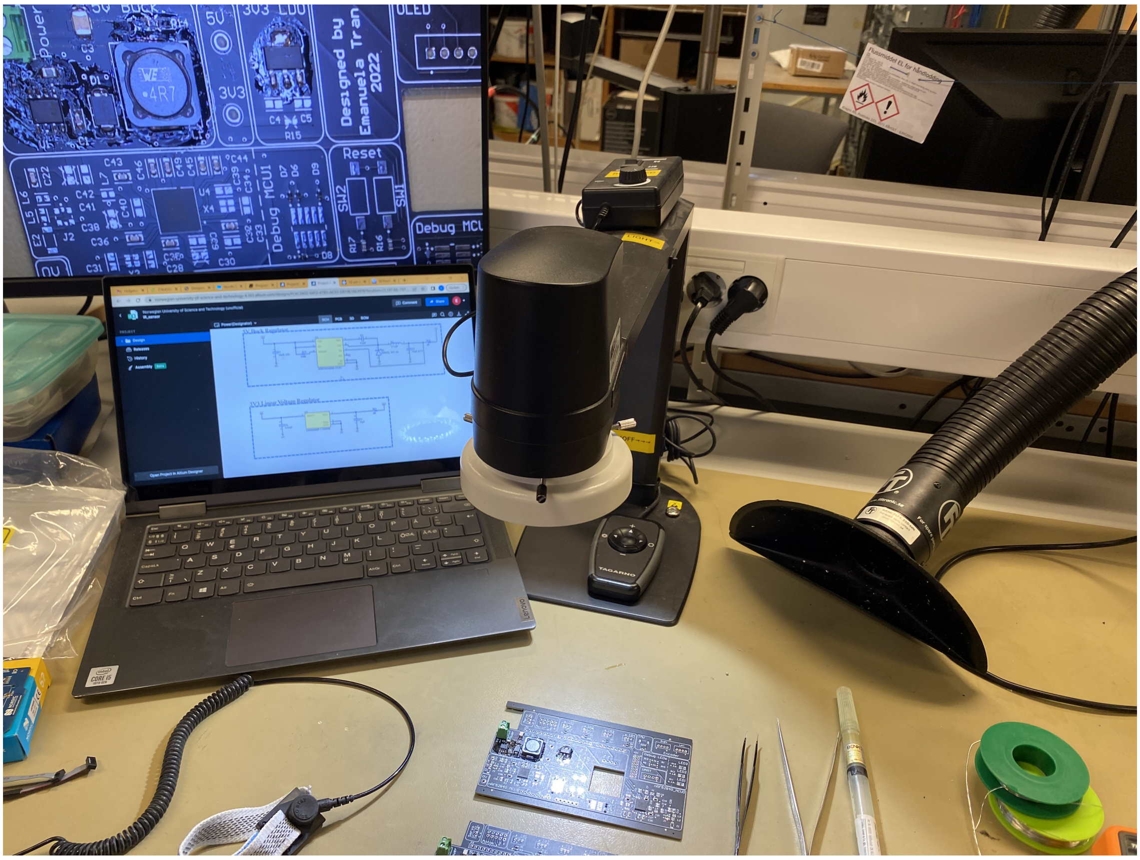


Figure 49: Set up for soldering the PCB

4.8 Test setup

For testing the tools used where:

- Power supply
- Test lead set
- Wires
- Multimeter
- ESD-band

The power supply was set to 12 V, imitating the 12 V battery on the robot. The black and red test leads where connected to the power supply and clipped onto the blue and red wire. Both wires were connected to the power connector. A multimeter was used to test the voltage across the board and check for short circuits. An ESD-band was used during testing whenever it was needed to physically touch the PCB or directly testing with a mutlimeter. Figure 50 shows the test setup for testing the PCB.

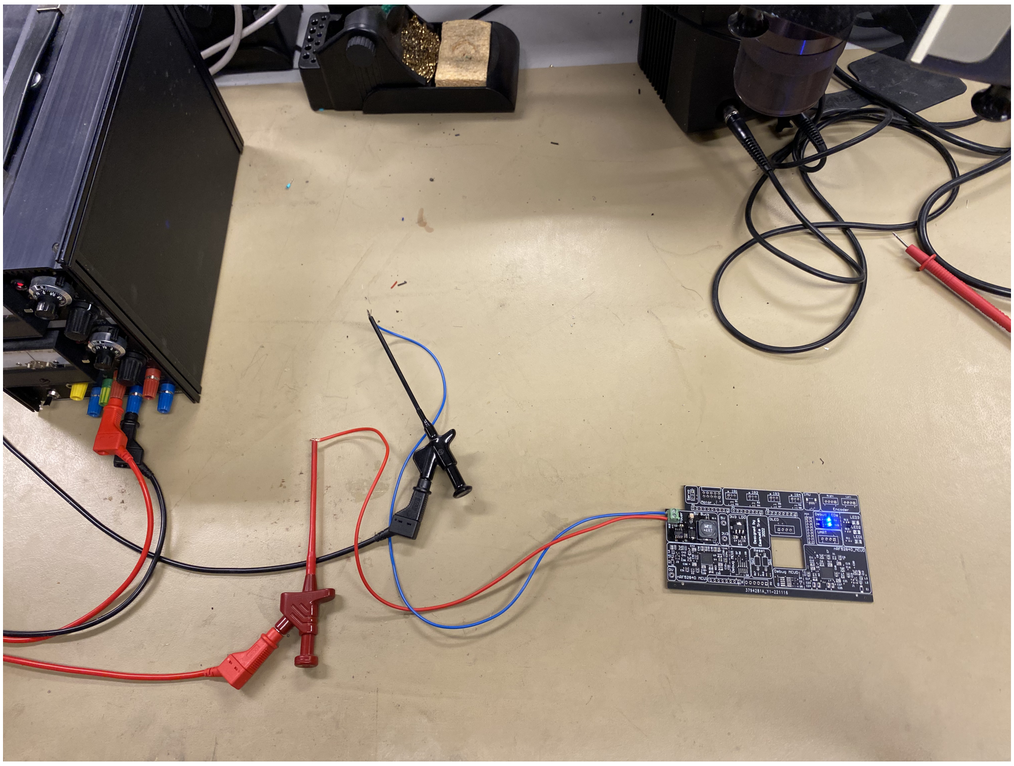


Figure 50: Test setup

5 Results

This chapter will present the results from designing the PCB and the results from testing the designed hardware.

5.1 Design results

The final design results for the PCB are:

- Board size: 130mm x 67 mm
- Four layers: top, ground, power and bottom
- 1 Oz copper thickness
- PCB can deliver both 5 V and 3.3 V
- Two nRF52840 SoCs with debug connector
- Connectors supporting connection with nRF52840-DK
- Connectors for power supply
- Total of 11 connectors for peripherals such as servo, motor driver, IR-sensors, IMU, encoders, OLED and UART
- Test point for 0 V, 5 V and 3.3 V
- Debug LEDs for 5 V and 3.3 V
- Indication LEDs for the robot
- Board cutout for accessing debug connectors from the nRF52840-DK

Figure 51 shows the front of the PCB in the software tool Altium designer. It is designed such that all modules are visually separated with boxes. Connectors are placed for easy access and similar connectors are grouped together. Each box have describing text about the circuit or connector inside. Crosses over components pads as visible, to indicate that these components should not be soldered on immediately or at all depending on the module.

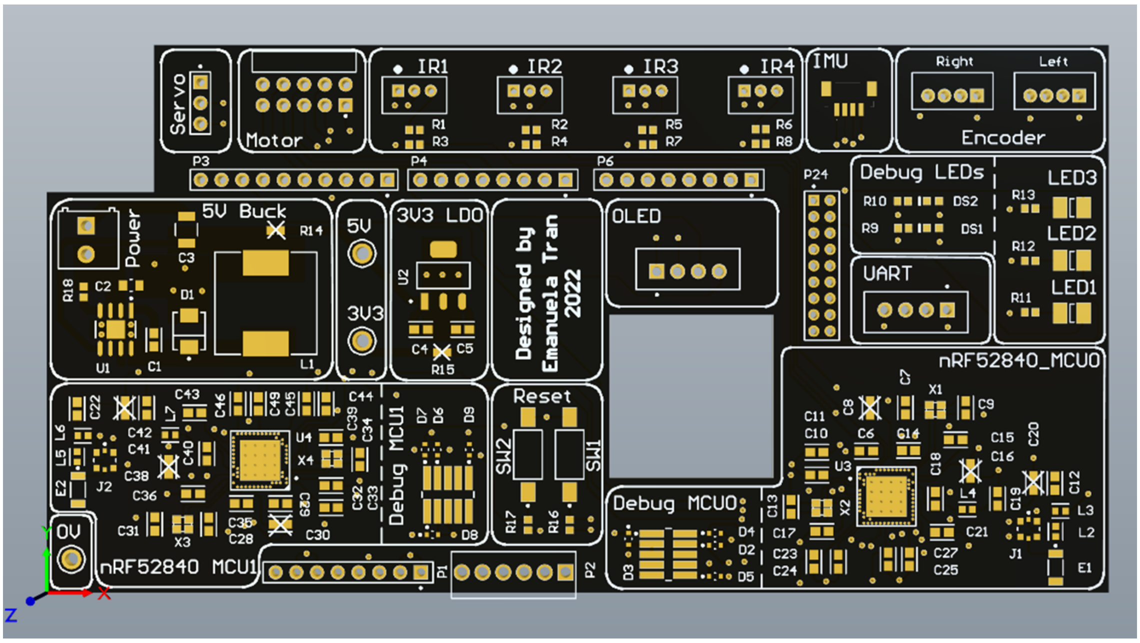


Figure 51: PCB front

Figure 52 shows a 3D example version of the PCB with components. Not all components are included or visually correct. Altium Designer gives an example view of the components they have i 3D view.

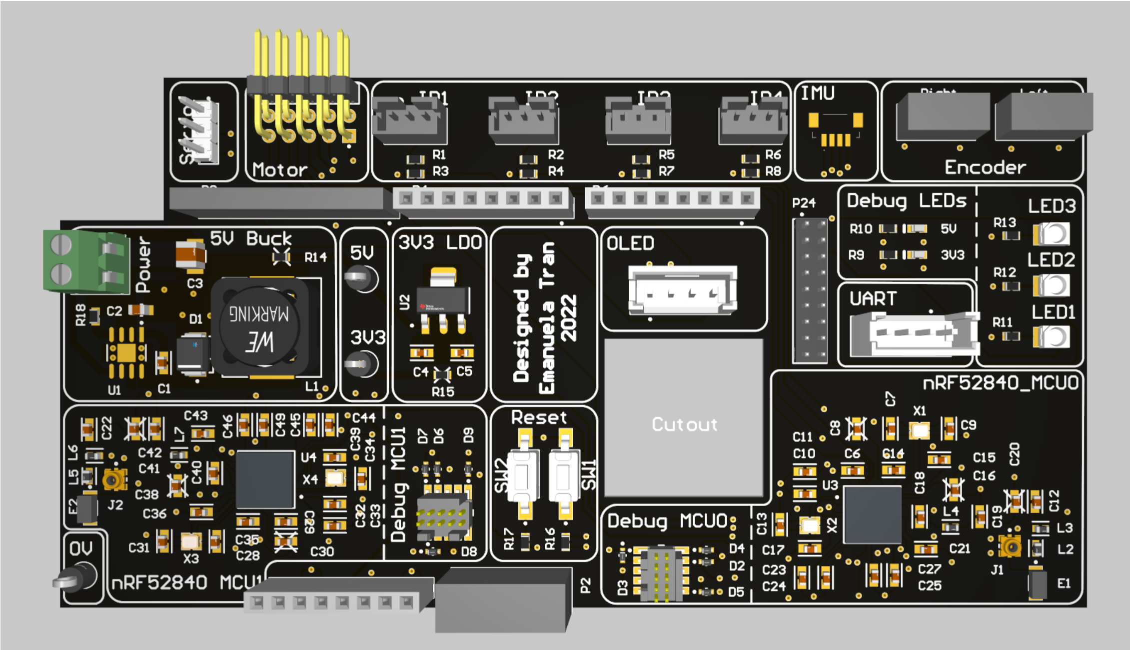


Figure 52: PCB 3D version with components

The PCB fits on top of the nRF52840-DK and on the robot as seen in figure 53 and 54.

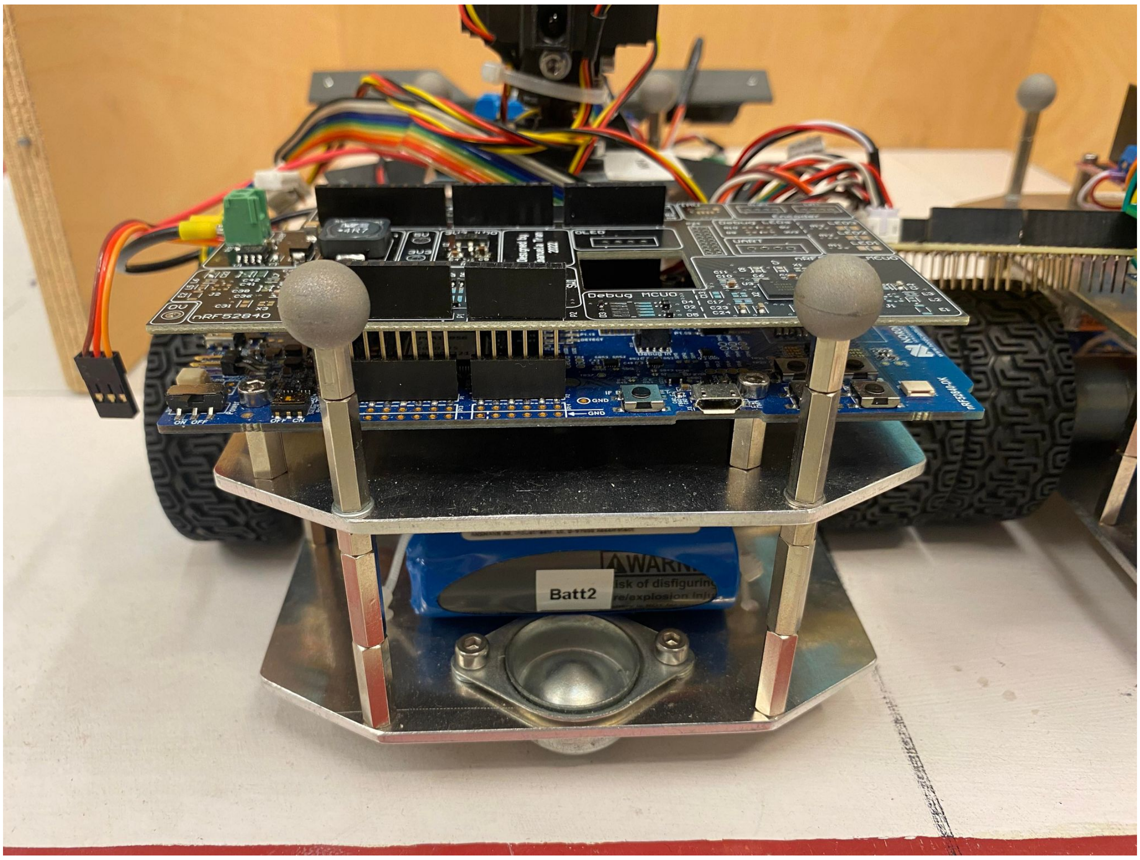


Figure 53: Side view of robot with new the PCB on top of nRF52840-DK

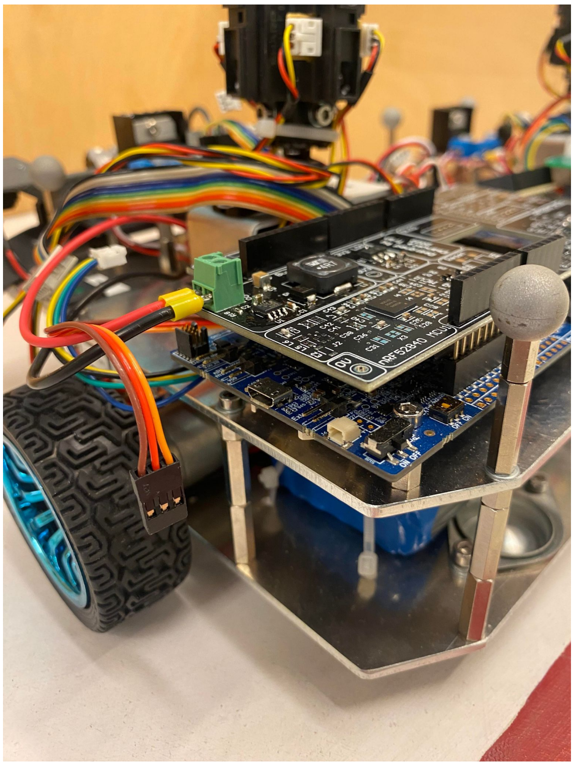


Figure 54: A close up on the robot with the new PCB on top of the nRF52840-DK

Figure 55 shows a side to side comparison of two robots where one has the new PCB and the other has the custom shield on top of the development kit.

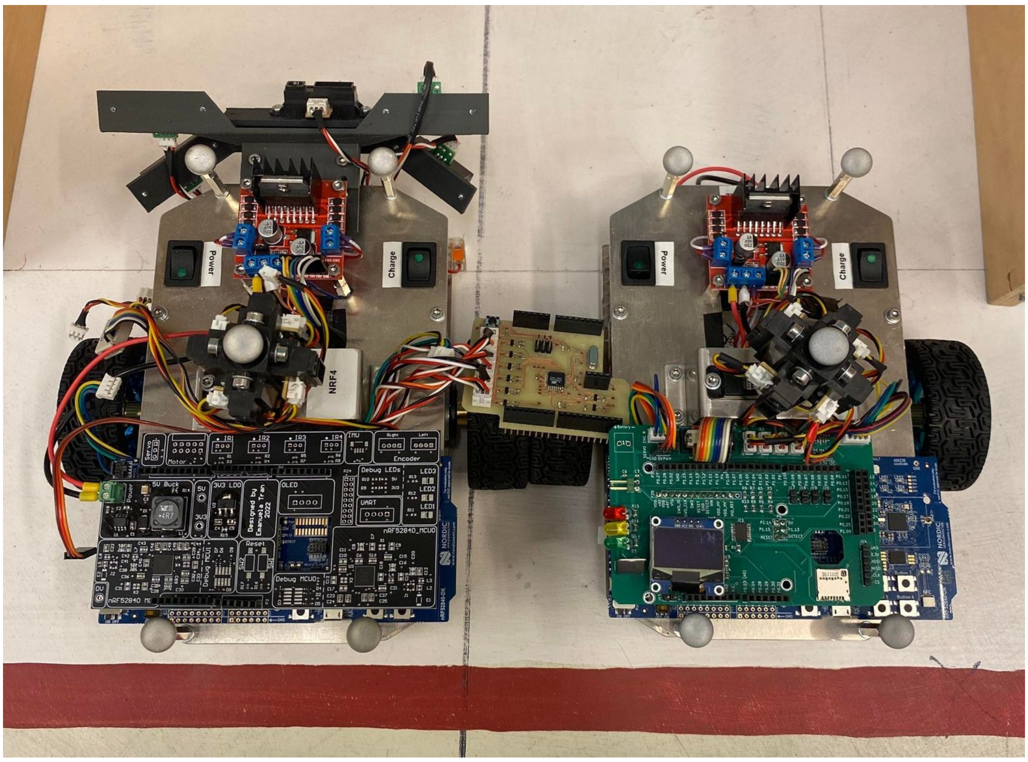


Figure 55: Overview of two robots: the left robot with the new PCB and the right with the custom shield

5.1.1 Faults in the design

- The footprint of connector P24 is wrong and does not fit with the connector below the nRF52840-DK. The pitch should be 2.54mm but the footprint of the used connector is 2.00mm.
- + and - signs are not included next to the power connector.

5.2 Test results

After soldering the 5V buck regulator, a multimeter was used to measure the voltage from the output. The output voltage was measured to be 5.012 V as seen in figure 56. The 0 V jumper resistor was then soldered, making the 5V available for the whole PCB. Same for the 3.3 V linear regulators, it was soldered and afterward tested with a multimeter. The output voltage was measured to be 3.254 V as seen in figure 57.

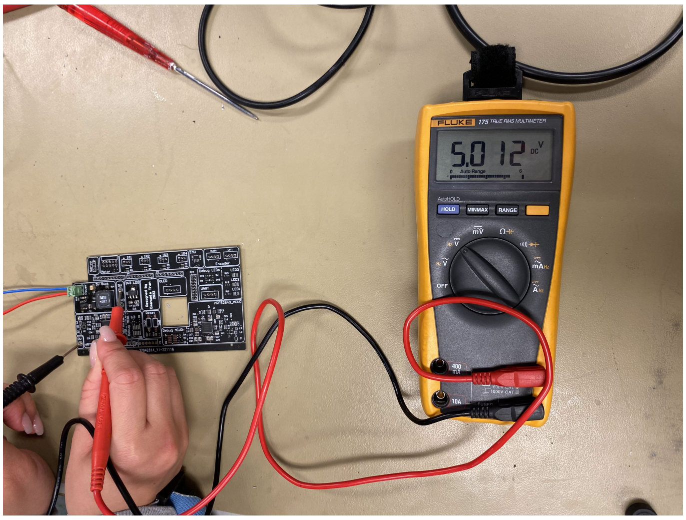


Figure 56: Measuring the output voltage for the buck regulator with multimeter

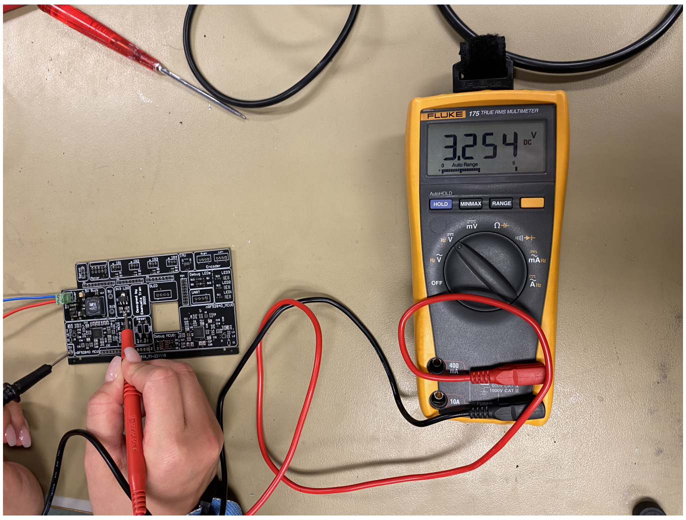


Figure 57: Measuring the output voltage for the linear voltage regulator with mulitmeter

Now that the power lines were measured and passed the test, they could be made available for the rest of the PCB. Debug LEDs where soldered on to isolate issues regarding power. Figure 58 shows two shining blue LEDs which indicates that 5 V and 3.3 V are available.

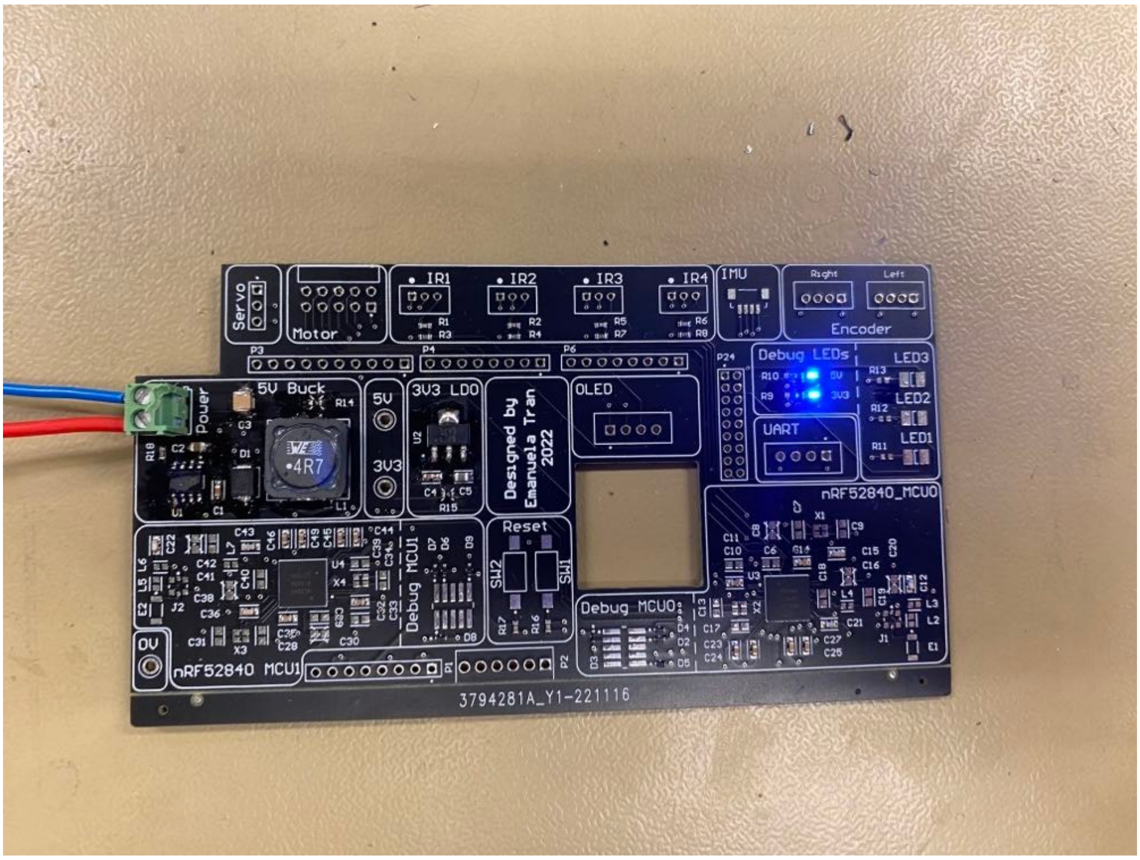


Figure 58: Debug LEDs shining for 5 V and 3.3 V

Unfortunately, due to time constraints, the soldering around the two nRF52840 circuits was not completed, so further testing could not be performed. However, upon visual inspection, no issues with the layout around the circuits were observed compared to the reference layout for the SoC.

6 Discussion

This chapter will discussed the design of the PCB as well as the testing of it.

6.1 PCB design

There are several aspects to consider when discussing the design of the PCB, which will be divided into three parts: the software tool used for design, the schematic, and the layout.

6.1.1 Software tool Altium Designer

As the complexity of the robot project has increased over time, I determined that it was necessary to upgrade the software tool used for PCB design. In the past, the KiCAD software was used to design a custom shield, which was suitable given the complexity of the shield. However, the PCB designed this year includes two SoCs, which makes the circuits on the PCB more complex. PCBs with SoCs tend to require more revisions, so it is important to have a software tool that allows for iteration on the design. That is why I chose to use Altium Designer, which includes features such as cloud integration and version control. These features enable me to make changes to the design while still being able to access earlier phases of the project if necessary.

For example, if there are changes to be made to the 5 V buck regulator, those changes can be made in the power sheet without affecting any other part of the design. When the changes are saved and uploaded to the cloud, anyone with access to the project can see what was modified and in which sheet the changes were made. If any mistakes were made, it is possible to track back to a previous version and identify any potential errors.

6.1.2 Schematic Design

The schematic was designed with the intention of facilitating future development and providing an easy-to-understand overview of the system to anyone working on the project. The usability and comprehensibility of the schematic may vary depending on the user's level of hardware knowledge and familiarity with the robot project. As the developer of the PCB, my perspective may be biased due to my prior understanding of the system. However, I have tried to approach the schematic from an objective standpoint and make it as clear and organized as possible, using it as a reference for layout design, soldering, and testing the PCB

The main sheet includes a large number of connectors and sheet symbols. Initially, I considered placing all of the connectors on the right side and the stackable headers that connect the PCB to the nRF52840-DK on the left side. However, I realized that grouping the remaining connectors together could be confusing and make it difficult to determine which connectors are connected to the development kit or one of the SoCs.

As a result, it was decided that the stackable headers should be placed with the four connectors related to the motor driver, OLED, and encoder. In addition to being labeled with their own boxes, these four connectors have the word 'bypass' included in their titles. The connectors that pass through one of the SoCs were placed on the right side of the sheet.

The center of the PCB sheet contains the core of the PCB. The titles of the boxes for both SoCs are misleading because the nRF52840 chips are actually SoCs and not microcontrollers. This also applies to the titles on each sheet symbol. It would be more accurate to label them as nRF_ServoTower and nRF_SLAM or SoC_ServoTower and SoC_SLAM.

Despite some minor errors in terms of text and placements, the rest of the schematic is well organized. The power sources are always shown at the top, while ground is shown at the bottom. The different sheets and circuits have a consistent structure and rarely intersect, which helps to minimize the time required to understand the various circuits. These design choices were made to minimize the time used to understand the different circuits.

The schematic was very useful during the soldering and testing phase for the PCB. The clear division of the different circuits made it easier to focus on soldering, as there were fewer symbols to consider. The schematic was also essential as a reference when designing the layout. While it is possible to create the layout without referring to the schematic, as the layout is synchronized with the schematic, this would not provide an accurate representation of the circuit flow and order. The layout only includes information about components that share the same net labels, and does not provide an understanding of the overall circuit structure.

6.1.3 Layout Design

Designing the layout can be time-consuming, and it is important to consider the amount of time that will be invested in a project like this. If the goal is simply to create a PCB that functions, it may be possible to do so in a few days. However, this approach would neglect the fact that such a PCB would be difficult to debug, test, and understand for future development. In this project, I have prioritized the layout for future development and the efficient testing and debugging of the PCB. It can be overwhelming to be presented with a PCB that lacks labels and information, particularly in the context of the robot project where some students may not have any background in hardware. Therefore, the layout includes descriptions of each circuit on the PCB to help users understand its functionality.

The most challenging aspect of the layout design was around the nRF52840. I chose a larger component package than the one recommended in the datasheet, which resulted in a layout that differs from the reference design. The layout extends further outward, but the positions of the components around the nRF52840 remain relatively similar. Nordic Semiconductor advises on their website that "To ensure good RF performance when designing PCBs, it is highly recommended to use the PCB layouts and component values provided by Nordic Semiconductor." This suggests that my choice may only impact the radio frequency performance, which is not relevant for this project.

Matching the lengths of the communication lines was not a primary focus. The shortest route was chosen, but in retrospect it is evident that the SPI lines are not evenly matched as the pinouts are spread across the board and not selected to match the communication lines.

Compared to the custom shield, the connectors on the new PCB are more widely spaced and positioned in a way that allows for easy disconnection and reconnection without having to pull the wires. The silkscreen on the PCB has been useful during the soldering and testing process, as it has made it easy to solder components one at a time and ensure that all components for a particular circuit are properly soldered in place.

6.2 Testing

From the test results it can be observed that the PCB delivers 5.012 V and 3.254 V. The output voltage from both regulators are acceptable as multimeters typically has an accuracy of 1% or less. As the output voltage of the buck regulators will be regulated ones more before delivering power to the nRF52840 on the development kit, the accuracy is not as important as the power delivered for the two SoC on the PCB. Figure 59 shows the recommended operation condition for the SoC, which means that the results from testing the linear voltage regulator are satisfied.

Symbol	Parameter	Notes	Min.	Nom.	Max.	Units
VDD	VDD supply voltage, independent of DCDC enable		1.7	3.0	3.6	V

Figure 59: Recommended operation condition for nRF52840

The multimeter used does not measure or indicate power spikes, which means that the stability of the output voltage from the regulators cannot be determined. This could potentially affect the performance of the different actuators on the robot, as unstable voltage input can cause servos to vibrate or shake.

7 Future work

This chapter will present the future work for this project. First the future work specifically on the PCB that has been developed will be discussed, and last suggestions for future work of the system where suggested changes can effect other systems on the robot-project.

7.1 Future work on the PCB

Chapter 5 ends with test results of the PCB where only the regulators were tested. As discussed in chapter 6, measuring voltage spikes over both regulators should be considered before soldering the rest of the board. After soldering the rest of the board, testing of each nRF52840 can be done.

Most of the hardware on the system is done and software development can be further developed. Additional hardware testing is still needed, but can now be tested simultaneously with new software.

It is important to divide the testing into different phases. First the main goals is to have running software on the one of the SoC on the board.

The next phase would be testing the PCB with the development kit and test for communication. Here, testing the SPI communication between both SoC on the board might be better and easier as there are almost little to nothing of software to handle. After figuring out the SPI setup for both SoCs, then testing with the development kit might be easier.

The last phase is testing the board with the robot and see how the software on the board cooperates with the rest of the robot.

During hardware testing, if more faults are found, a new revision of the PCB should be considered. Here are some fixes that should be considered if a new revision is necessary:

- Change descriptive titles in the schematic for both SoC.
- Add + - signs in silkscreen for the power connector.
- The connector P24 must be changed to be the same pitch as the connector below on the development kit. Also possible to physically modify the connector such that it connects the used pinouts to the development kit, but these are short term modifications.
- Consider using 0402 package around the nRF52840 if it is known that these components can be soldered at a manufacturer. This is to make the use of RF technology available for future development.
- Consider using a solder oven than hand soldering. A stencil might be necessary to be ordered.
- More describing text around connectors such as around the stackable headers.

7.2 Future work of the system

As the project keeps evolving and more people are involved, it is challenging to develop a robot where all systems must be tightly integrated. Because the project is carried out through project thesis' and masters thesis', collaboration between students is some times limited, and developing a system that must be integrated on the robot is challenging.

A key reason for this, in my opinion, is that the robot is not divided into separate systems that have a defined communication interface between each others. Without this clear separation, it is difficult to make changes to a system, without risking breaking the functionality of other systems.

This is a development-area that is key for the success of the project. A proposal to improve modularity, is therefore to incorporate a CAN communication network on the robot. As of now, SPI and I2C is used on this PCB to communicate with various systems on the robot, such as the nRF52840-DK, the IMU and motor driver. Some systems, such as the motor driver, only needs to communicate with one device, however because SPI and I2C is used, the master-device (which might not be the end receiver) needs to facilitate the data transfer and serve as a middle-man. This makes the robot overly dependent on each system, and makes it prone to software bugs.

If CAN, which is a message-based protocol is used however, every device could in practice communicate with each others, and a predefined set of messages on the network could be defined. This would make it easier to develop a specific system, as the interface with the robot is clearly defined. In addition, new sensors or devices would be much easier to integrate, as they would only need to be connected to the CAN-bus in order to communicate with the other systems.

Implementing CAN is however no small task. As mentioned in section 3.5.4, CAN requires two hardware components: a CAN controller and a CAN transceiver. Some microcontrollers or SOCs integrate the CAN controller, however all would need a transceiver. In other words, every system on the robot would need a new revision. In addition, some systems or sensors, such as the motor driver and the IR sensors, does not incorporate a microcontroller. These would therefore also need a microcontroller to process messages.

Because of this, implementing CAN should not be rushed, but first thoroughly developed and tested on a subset of the systems, for instance the sensor data acquisition board and nRF52840DK. Once these two are upgraded to use CAN, the other systems can be upgraded.

Bibliography

[1] M.Lindefjeld. *Obstacle Detection and Avoidance in a 3D Enviroment*. NTNU Trondheim, 2022.

[2] E.Jølsgard. *Shield for embedded nrf52840-DK robot*. NTNU Trondheim, 2020.

[3] Nordic Semiconductor. *nRF52840DK*. URL: https://www.nordicsemi.com/Products/Development-hardware/nrf52840-dk (visited on 10/12/2022).

[4] Machifit. *Machifit 25GA370 DC 12V Micro Gear Reduction Encoder Motor*. URL: https://www.banggood.com/Machifit-25GA370-DC-12V-Micro-Gear-Reduction-Encoder-Motor-with-Mounting-Bracket-and-Wheel-p-1532242.html?cur_warehouse=CN&ID=6157423 (visited on 10/12/2022).

[5] Handson Technology. *L29N Dual H-Bridge Motor Driver*. URL: http://www.handsontec.com/dataspecs/L298N%5C%20Motor%5C%20Driver.pdf (visited on 10/12/2022).

[6] InvenSense. *ICM-20948*. URL: https://www.elfadistrelec.no/Web/Downloads/_m/an/SEN-15335_eng_man.pdf (visited on 10/12/2022).

[7] Sharp Electronics. *Sharp GP2YA21YK0F datasheet*. URL: https://global.sharp/products/device/lineup/data/pdf/datasheet/gp2y0a21yk_e.pdf (visited on 10/12/2022).

[8] DGServo. *Servo - Generic Metal Gear (Micro Size)*. URL: https://www.sparkfun.com/products/14760 (visited on 10/12/2022).

[9] KiCad Development Team. *KiCad*. URL: https://www.kicad.org/ (visited on 15/12/2022).

[10] John Catsoulis. *Designing Embedded Hardware*. O'REILLY, 2020.

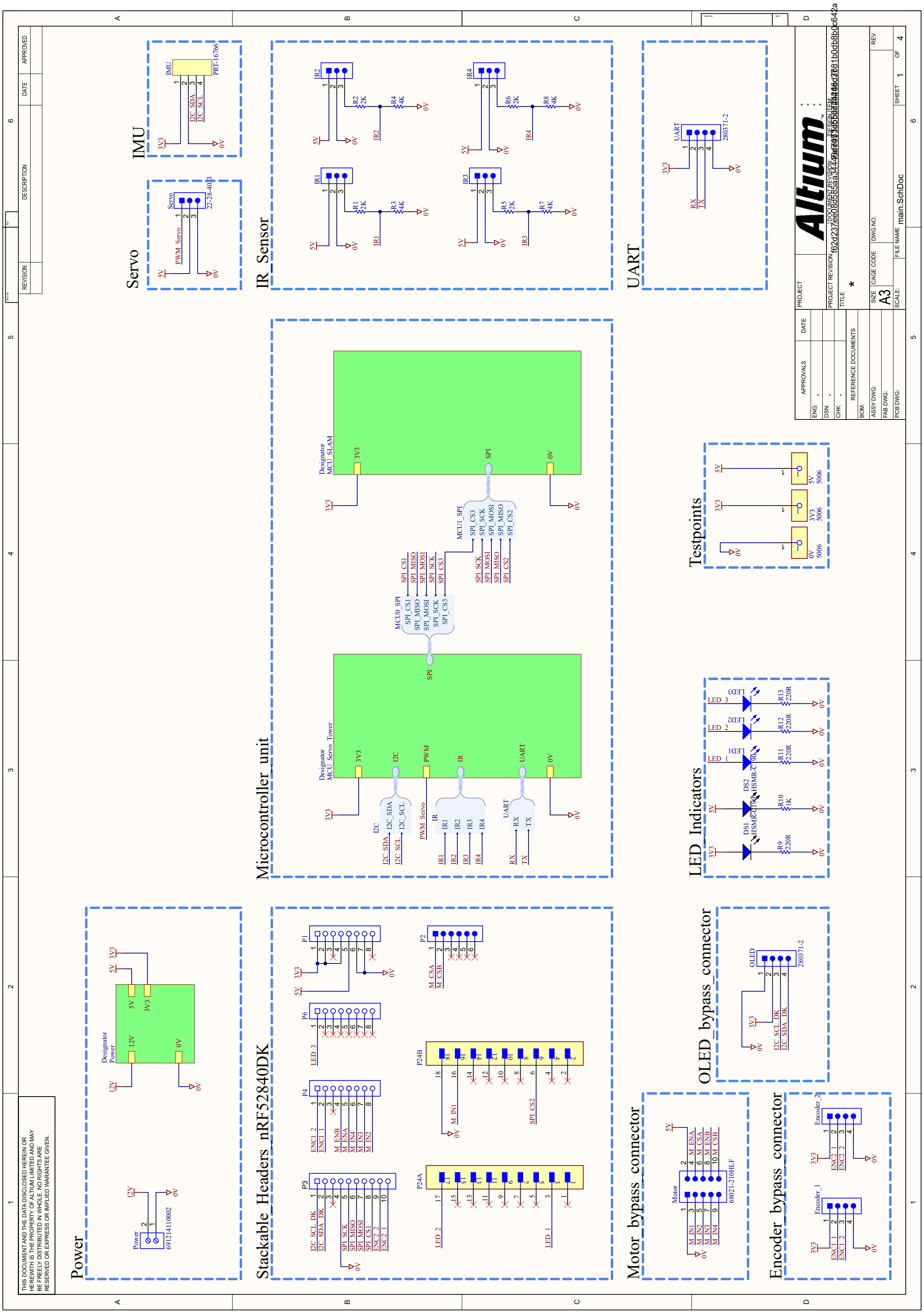
[11] Nordic Semiconductor. *nRF52840*. URL: https://infocenter.nordicsemi.com/pdf/nRF52840_PS_v1.1.pdf (visited on 18/12/2022).

[12] Analog Devices. *ADP2303 datasheet*. URL: https://www.analog.com/media/en/technical-documentation/data-sheets/adp2302_2303.pdf (visited on 18/12/2022).

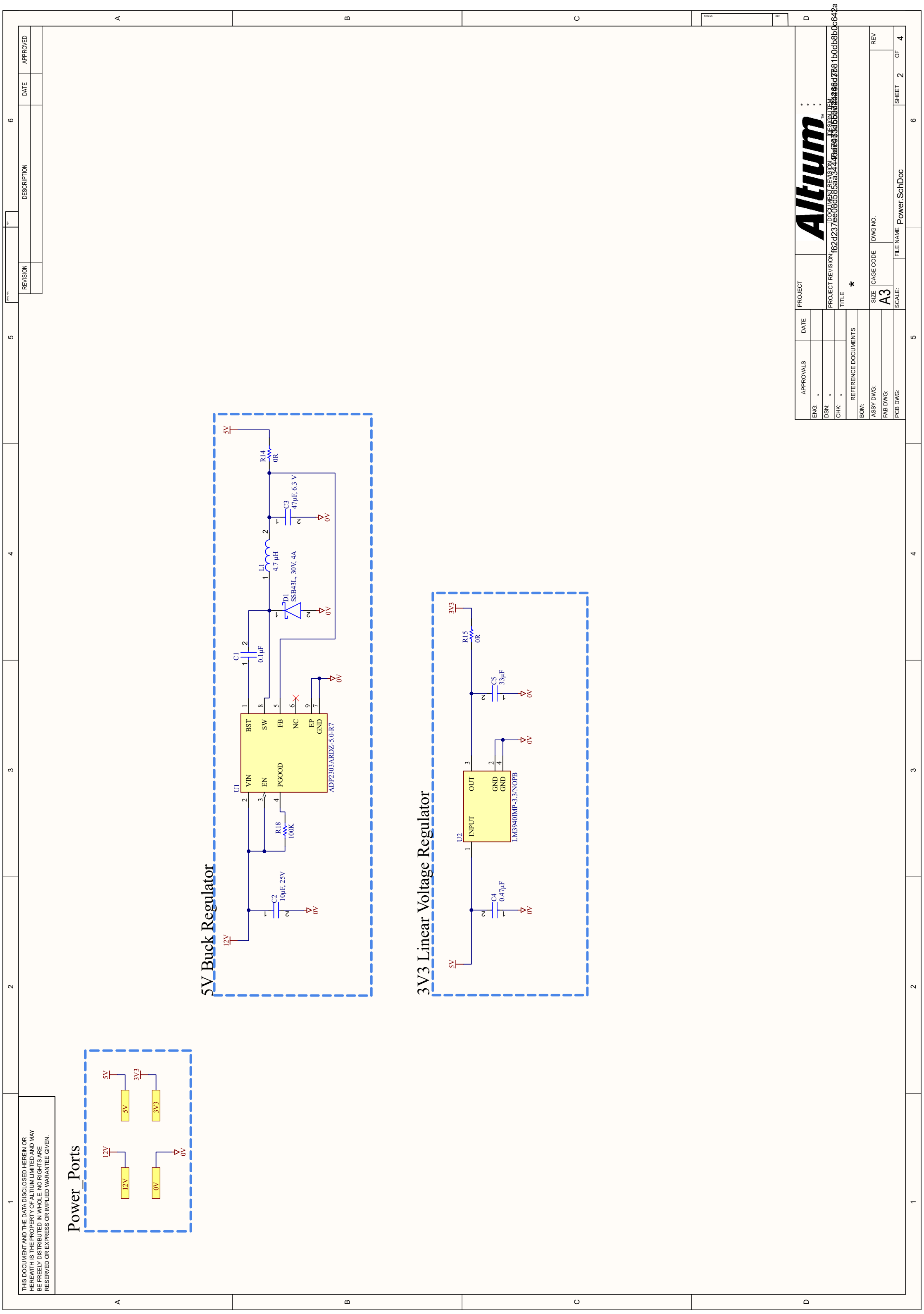
[13] Texas Instruments. *LM3940 datasheet*. URL: https://www.ti.com/lit/ds/symlink/lm3940.pdf (visited on 18/12/2022).

Appendix

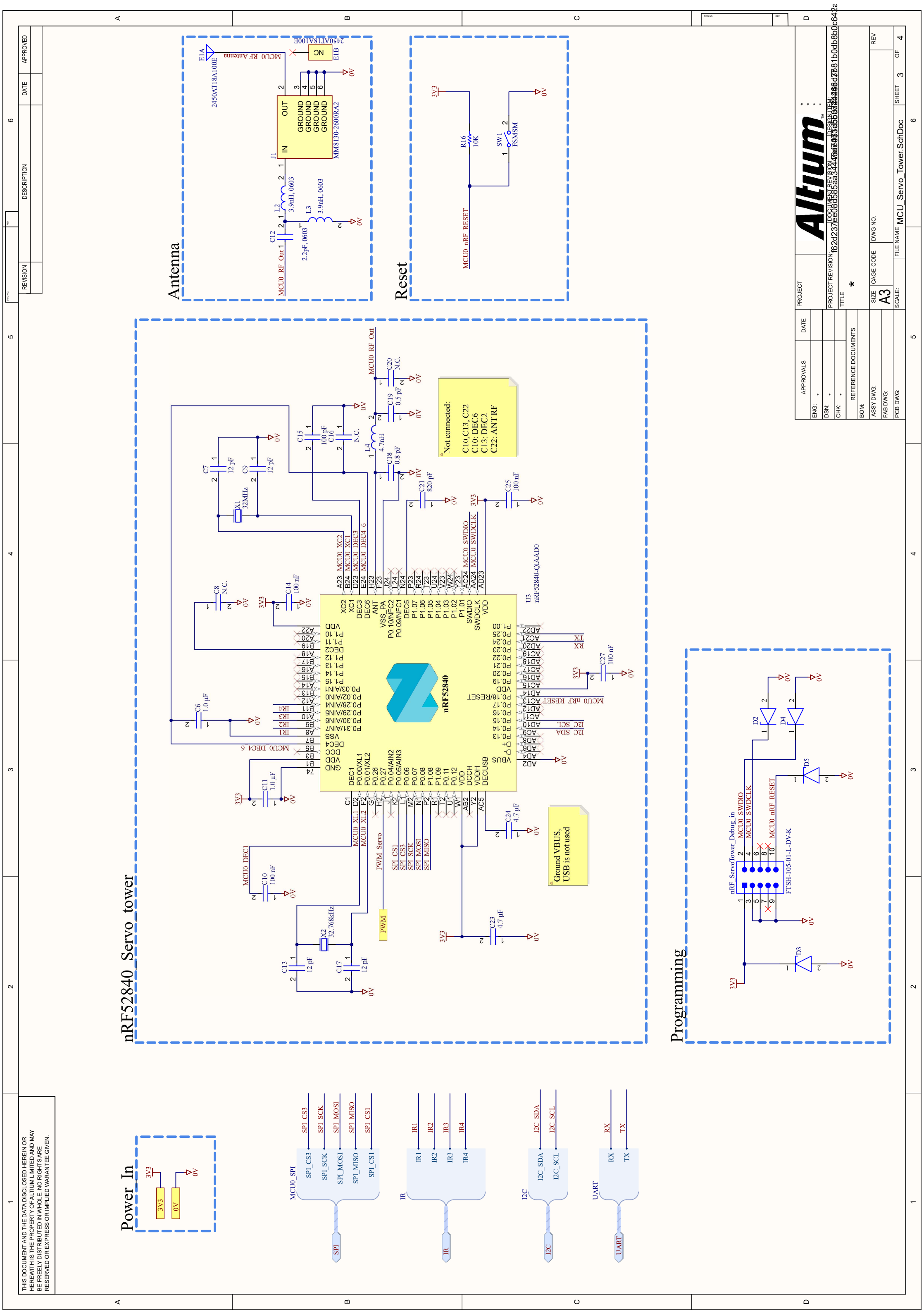
A Schematic main



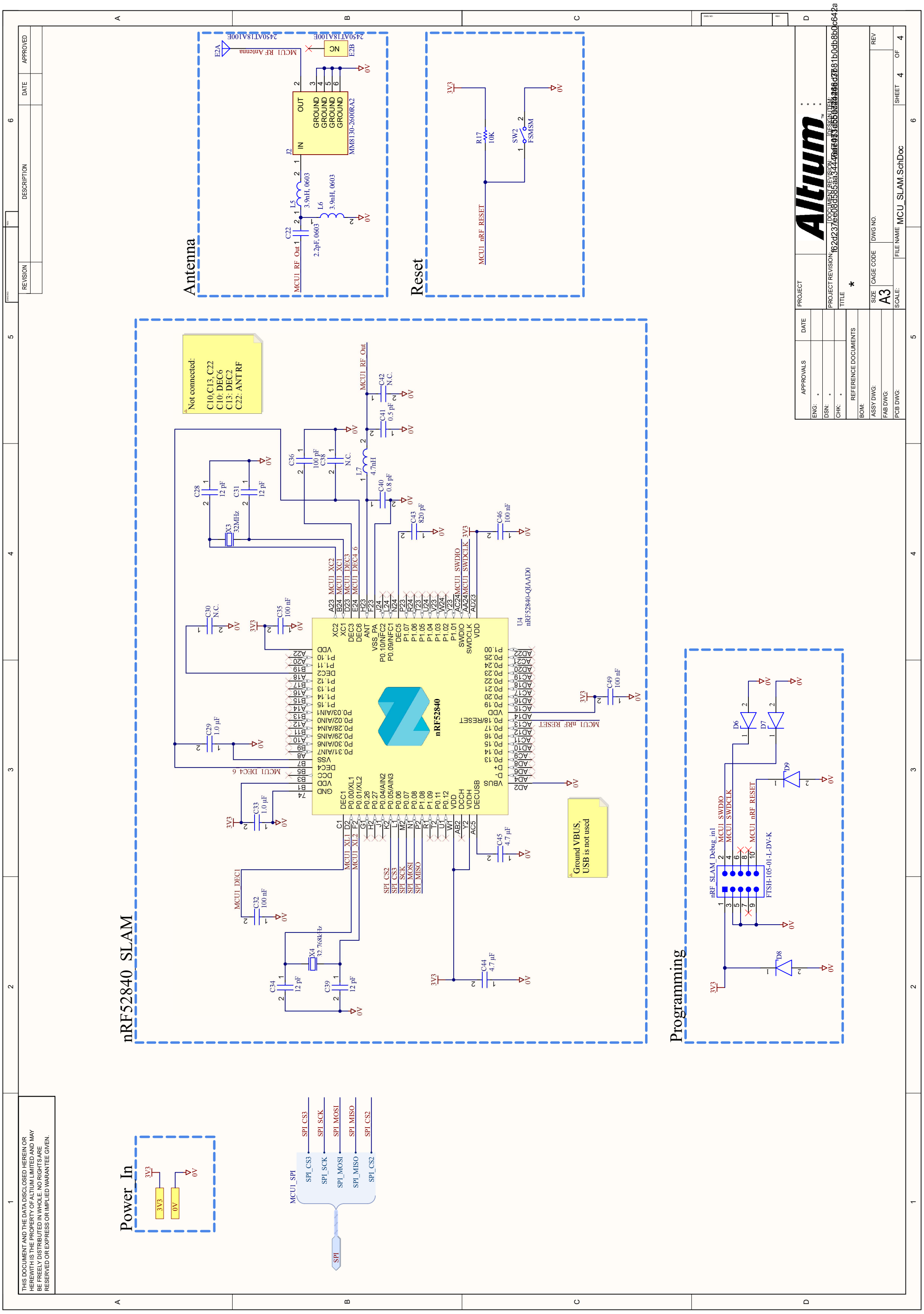
B Schematic power



C Schematic servo tower



D Schematic SLAM



E Bill of Material 1/5

Name	Description	Designtor	Manufacturer	MPN	Where?	Amount
5006	PC TEST POINT COMPACT BLACK	0V	Keystone Electronics	5006	Mouser	5
5006	PC TEST POINT COMPACT BLACK	3V3	Keystone Electronics	5006	Mouser	5
5006	PC TEST POINT COMPACT BLACK	5V	Keystone Electronics	5006	Mouser	5
0.1uF	Multilayer Ceramic Capacitors MLCC - SMD/SMT 0201 16VDC 0.1uF 10% X5R	C1	Kyocera AVX	EMK063BJ104KP-F	Mouser	8
TMK212BBJ106KG-T	Multilayer Ceramic Capacitor 10uF 25V X5R 10% SMD 0805 T/R	C2	Taiyo Yuden	TMK212BBJ106KG-T	Mouser	8
GRM32ER60J476ME20L	CAP CER 47UF 6.3V X5R 1210	C3	Murata	GRM32ER60J476ME20L	Mouser	6
CL03A474KP3NNNC	0.47uF	C4	Samsung electro mechanics	187-CL03A474KP3NNNC	Mouser	8
C2012X5R1A336M	33uF	C5	TDK	810-C2012X5R1A336M	Mouser	6
CS603KPX7R7BB105	1uF	C6,C11,C29,C33	YAGEO	603-CS603KPX7R7BB105	Mouser	24
02015A120GAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C7,C9,C13,C17, C28,C31,C34,C39	Kyocera AVX	02015A120GAT2A	Mouser	45
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C10	Kyocera AVX	06035C102KAT2A	JLCPCB	
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C12	Kyocera AVX	06035C102KAT2A	JLCPCB	
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C14	Kyocera AVX	06035C102KAT2A	JLCPCB	
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C15	Kyocera AVX	06035C102KAT2A	JLCPCB	
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C18	Kyocera AVX	06035C102KAT2A		
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C19	Kyocera AVX	06035C102KAT2A		

F Bill of Material 2/5

06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C20	Kyocera AVX	06035C102KAT2A		
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C40	Kyocera AVX	06035C102KAT2A		
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C41	Kyocera AVX	06035C102KAT2A		
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C42	Kyocera AVX	06035C102KAT2A		
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C43	Kyocera AVX	06035C102KAT2A	JLCPCB	
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C44	Kyocera AVX	06035C102KAT2A	JLCPCB	
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C45	Kyocera AVX	06035C102KAT2A	JLCPCB	
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C46	Kyocera AVX	06035C102KAT2A	JLCPCB	
06035C102KAT2A	General Purpose Ceramic Capacitor, 0603, 1nF, 10%, X7R, 15%, 50V	C49	Kyocera AVX	06035C102KAT2A	JLCPCB	
VS-10BQ015-M3/5BT	DIODE SCHOTTKY 15V 1A DO214AA	D1	Vishay	VS-10BQ015-M3/5BT	Mouser	6
PESD3V3L1UB,115	TVS DIODE 3.3VWM 11VC SOD523	D2	Nexperia	PESD3V3L1UB,115	JLCPCB	
PESD3V3L1UB,115	TVS DIODE 3.3VWM 11VC SOD523	D3	Nexperia	PESD3V3L1UB,115	JLCPCB	
PESD3V3L1UB,115	TVS DIODE 3.3VWM 11VC SOD523	D4	Nexperia	PESD3V3L1UB,115	JLCPCB	
PESD3V3L1UB,115	TVS DIODE 3.3VWM 11VC SOD523	D5	Nexperia	PESD3V3L1UB,115	JLCPCB	
PESD3V3L1UB,115	TVS DIODE 3.3VWM 11VC SOD523	D6	Nexperia	PESD3V3L1UB,115	JLCPCB	
PESD3V3L1UB,115	TVS DIODE 3.3VWM 11VC SOD523	D7	Nexperia	PESD3V3L1UB,115	JLCPCB	
PESD3V3L1UB,115	TVS DIODE 3.3VWM 11VC SOD523	D8	Nexperia	PESD3V3L1UB,115	JLCPCB	
PESD3V3L1UB,115	TVS DIODE 3.3VWM 11VC SOD523	D9	Nexperia	PESD3V3L1UB,115	JLCPCB	
150060BS75003	LED blue 0603	DS1,DS2	Wurth Electronics	150060BS75003	Mouser	12
2450AT18A100E	Ceramic Antenna, 2.45 GHZ, 2 W, -40 to 125 degC, 2-Pin SMD, RoHS, Tape and Reel	E1	Johanson	2450AT18A100E		

G Bill of Material 3/5

2450AT18A100E	Ceramic Antenna, 2.45 GHZ, 2 W, -40 to 125 degC, 2-Pin SMD, RoHS, Tape and Reel	E2	Johanson	2450AT18A100E		
B4B-PH-K-S(LF)(SN)	Male Header, Pitch 2 mm, 1 x 4 Position, Height 6 mm, Tail Length 3.4 mm, -25 to 85 degC, RoHS, Bulk	Encoder_1,Encoder_2	JST	B4B-PH-K-S(LF)(SN)	RS components	
PRT-16766	Qwiic JST Connector SMD 4pin	IMU			Mouser	5
B3B-PH-K-S(LF)(SN)	Male Header, Pitch 2 mm, 1 x 3 Position, Height 6 mm, Tail Length 3.4 mm, -25 to 85 degC, RoHS, Bulk	IR1,IR2,IR3,IR4	JST	B3B-PH-K-S(LF)(SN)	RS components	
MM8130-2600RA2	CONN SWF RCPT STR 50 OHM SMD	J1,J2	Murata	MM8130-2600RA2		
7447715004	SMD-Shielded Power Inductor WE-PD, L = 4.70 µH	L1	Wurth Electronics	7447715004	Mouser	5
	3.9nH 0603	L2,L3	Murata			
LQM18NNR10K00D	Multilayer Type Inductor for General Circuits, 0603 (1608), 100nH, 10%, 0.5Ω, 50mA	L4	Murata	LQM18NNR10K00D		
LQM18NNR10K00D	Multilayer Type Inductor for General Circuits, 0603 (1608), 100nH, 10%, 0.5Ω, 50mA	L5	Murata	LQM18NNR10K00D		
LQM18NNR10K00D	Multilayer Type Inductor for General Circuits, 0603 (1608), 100nH, 10%, 0.5Ω, 50mA	L6	Murata	LQM18NNR10K00D		
LQM18NNR10K00D	Multilayer Type Inductor for General Circuits, 0603 (1608), 100nH, 10%, 0.5Ω, 50mA	L7	Murata	LQM18NNR10K00D		
150141GS73100	SMD mono-color Top LED, WL-SMTW, Green	LED1	Wurth Electronics	150141GS73100	Mouser	5
150141YS73100	SMD mono-color Top LED, WL-SMTW, Green	LED2	Wurth Electronics	150141YS73100	Mouser	5
150141RS73100	SMD mono-color Top LED, WL-SMTW, Green	LED3	Wurth Electronics	150141RS73100	Mouser	5

H Bill of Material 4/5

PRT-08506	Male Header, Pitch 2.54 mm, 2 x 5 Position, Height 4.83 mm, Tail Length 3.05 mm, Bulk	Motor			Mouser	5
FTSH-105-01-L-DV-K	Male Header, Pitch 1.27 mm, 2 x 5 Position, Height 6.25 mm	nRF_ServoTower_Debug_in1,2	Samtec	FTSH-105-01-L-DV-K	RS components	
280371-2	Male Header, Pitch 2.54 mm, 1 x 4 Position, Height 12.8 mm, Tail Length 3.5 mm, -55 to 105 degC, RoHS, Bulk	OLED	TE Connectivity	280371-2		
PPPC081LFBN-RC	Female Header, Pitch 2.54 mm, 1 x 8 Position, Height 8.5 mm, Tail Length 3.2 mm, -40 to 105 degC, RoHS, Bulk	P1	Sullins	PPPC081LFBN-RC	Kjell og company	
640456-6	Male Header, Pitch 2.54 mm, 1 x 6 Position, Height 10.03 mm, Tail Length 3.56 mm, -55 to 105 degC, Bulk	P2	TE Connectivity	6404566	Kjell og company	
929850-01-10-RA	Female Header, Pitch 2.54 mm, 1 x 10 Position, Height 8.26 mm, Tail Length 3.18 mm, -40 to 105 degC, RoHS, Bulk	P3	3M	929850-01-10-RA	Kjell og company	
PPPC081LFBN-RC	Female Header, Pitch 2.54 mm, 1 x 8 Position, Height 8.5 mm, Tail Length 3.2 mm, -40 to 105 degC, RoHS, Bulk	P4	Sullins	PPPC081LFBN-RC	Kjell og company	
PPPC081LFBN-RC	Female Header, Pitch 2.54 mm, 1 x 8 Position, Height 8.5 mm, Tail Length 3.2 mm, -40 to 105 degC, RoHS, Bulk	P6	Sullins	PPPC081LFBN-RC	Kjell og company	
TMM-109-01-T-D	CONN HEADER VERT 18POS 2MM	P24	Samtec	TMM-109-01-T-D	Kjell og company	
6.91214E+11	Serie 2141 - 3.50 mm Horizontal Entry Modular with Rising Cage Clamp WR-TBL, 2 pin	Power	Wurth Electronics	6.91214E+11	RS components	
ERJ3EKF1002V	55 to 155 degC, 0603 (1608 Metric),	R1	Bourns	CR0603-FX-1002ELF	JLCPCB	
ERJ3EKF1002V	55 to 155 degC, 0603 (1608 Metric),	R2	Bourns	CR0603-FX-1002ELF	JLCPCB	
ERJ3EKF1002V	55 to 155 degC, 0603 (1608 Metric), RoHS, Tape and Reel	R3	Bourns	CR0603-FX-1002ELF	JLCPCB	
ERJ3EKF1002V	Chip Resistor, 10 KOhm, +/- 1%, 0.1 W, - 55 to 155 degC, 0603 (1608 Metric),	R4	Bourns	CR0603-FX-1002ELF	JLCPCB	

I Bill of Material 5/5

ERJ3EKF1002V	Chip Resistor, 10 KOhm, +/- 1%, 0.1 W, - 55 to 155 degC, 0603 (1608 Metric),	R5	Bourns	CR0603-FX-1002ELF	JLCPCB	
ERJ3EKF1002V	55 to 155 degC, 0603 (1608 Metric), RoHS, Tape and Reel	R6	Bourns	CR0603-FX-1002ELF	JLCPCB	
ERJ3EKF1002V	55 to 155 degC, 0603 (1608 Metric),	R7	Bourns	CR0603-FX-1002ELF	JLCPCB	
ERJ3EKF1002V	Chip Resistor, 10 KOhm, +/- 1%, 0.1 W, -	R8	Bourns	CR0603-FX-1002ELF	JLCPCB	
ERJ-PA3F2200V	220R	R9,R11,R12,R13	Panasonic	ERJ-PA3F2200V	Mouser	23
CR0201AFW-1001GLF	1K	R10	Bourns	CR0201AFW-1001GLF	Mouser	6
CR0603AJ/-000EAS	0 Ohm	R14,15	Bourns	CR0603AJ/-000EAS	Mouser	20
ERJ3EKF1002V	Chip Resistor, 10 KOhm, +/- 1%, 0.1 W, -	R16	Bourns	CR0603-FX-1002ELF	JLCPCB	
ERJ3EKF1002V	55 to 155 degC, 0603 (1608 Metric),	R17	Bourns	CR0603-FX-1002ELF	JLCPCB	
CRCW0201100KFNED	100K	R18	Vishay	CRCW0201100KFNED	Mouser	8
22-28-4033	Male Header, Pitch 2.54 mm, 1 x 3 Position, Height 8.38 mm, Tail Length 3.18 mm, RoHS, Bulk	Servo	Molex	22-28-4033		
FSMSM	FSMSM Push Button Switch, 50 mA, -35 to 85 degC, 2-Pin SMD, RoHS, Bulk	SW1,SW2	TE Connectivity Alcoswitch	FSMSM	Mouser	10
ADP2303ARDZ-5.0-R7	IC REG BUCK 5V 3A 8SOIC	U1	Analog Devices	ADP2303ARDZ-5.0-R7	Mouser	6
LM3940IMP-3.3/NOPB	IC REG LINEAR 3.3V 1A SOT223-4	U2	TI National Semiconductor	LM3940IMP-3.3/NOPB	RS components	
280371-2	Male Header, Pitch 2.54 mm, 1 x 4 Position, Height 12.8 mm, Tail Length 3.5 mm, -55 to 105 degC, RoHS, Bulk	UART	TE Connectivity	280371-2		
CX2016DB32000D0WZRC1	XTAL SMD 2016, 32MHz, Cl=8pF, Tol: ±10ppm, Stab: ±15ppm	X1,3	CX2016DB32000D0WZRC1	MP06003		
	XTAL SMD 2016, 32MHz, Cl=8pF, Tol: ±10ppm, Stab: ±15ppm	X2,4		MP06003		