Armature Joint Module

Submitted by Flynn Stilwell (2180710)

November 2020

Supervised by Frank Beinersdorf

Submitted as the Final Report for the MG7101 Engineering Development Project
for the Bachelor of Engineering Technology Programme
Electrical Major and the Mechatronics Specialisation

1. Abstract

Prototyping large robots at a miniature scale is a great way to validate a robot's design. It allows the engineer to test a robot's kinematic structure design and its range of motion in the real world, at a low cost. MechAdept employs this method for their high-performance mechatronic systems, but they do not give useful joint position feedback without manual measurement.

The aim of this project was to develop a functional prototype of a single joint module to prove the concept of a puppeteering armature which would be manipulated by hand and stream out position information to a user terminal via Bluetooth. This was for MechAdept to prototype robotics and present to clients a virtual puppeteered armature. The design had to be compact, battery powered, modular and simple. Rapid iterative prototyping was used to design the mechanical structure of the joint and the electronics system was designed using Autodesk Eagle, manufactured by PCBZone and assembled with the help of MechAdept.

The files for this project will be made available at the following GitHub Link:

https://github.com/FStilwell/Armature_joint

TABLE OF CONTENTS

Page

2. Statement of Objectives

The main objective of this project was to develop a low-cost functional prototype for a smart armature revolute joint to be used by MechAdept.

To meet the main objective, the following sub-objectives needed to be addressed. These were:

a) Develop a modular mechanical design that considers manufacturability, ease of use and assembly, and available resources.
b) Develop an electronics system design making use of the nRF51 or nRF52 series Bluetooth System on Chip (SoC) and implement it on a PCB.
c) Design and implement a working program to interface the electrical system elements and provide joint position data to a user terminal.

3. Research Scope

Traveler Hauptman, the Principal and Director of MechAdept, and Frank Beinersdorf, this project's supervisor and a Mechatronics/Robotics Engineer also at MechAdept, needed a mechatronic 'Smart Armature.' This is an actuator-free robotic joint purposed to demonstrate joint positions of model robot arms, while streaming the data to a user terminal. It needed to be as precise as a human, simple, modular, make use of the Nordic nRF51 or nRF52 chipsets for data transmission, and needed to cost less than $200. Beyond this project, the purpose of the joint was to be one of many, able to be connected interchangeably with links to form a miniature robot arm.

It was be preferred that the smart armature joint would be battery powered, held together magnetically, the position would be measured using a Hall effect sensor and the size would be kept as small as practically possible, but there was no hard constraint on this in the prototype stage.

From these statements, the following list of requirements/targets were established. These were subject to minor changes during the prototyping stage.

General	
Low Cost	<$200
Simple	
Fast to assemble/disassemble	
Modular design	
Mechanical Structure	
2 link joint	
Full angular range of motion	± 360°
Dimensions	<= Ø60mm X 100mm
Angular position holding torque	Approximately 1.8Kg.cm
Electronics	
Use nRF52 Chipset	
Battery Powered	
System on PCB	
Simple PCB	<= 4 layers
Accessible debugger point	
Software	
Bluetooth capable	
Moderate Precision	<= ± 1°

Although the armature would eventually be constructed using multiple joints, this project had a focus on one joint module prototype. After speaking with the client, it was mutually agreed that the project could focus more on the mechanical and PCB design, with the software working well enough to prove the concept.

Constraints

The time allocation for this project was 300 hours and was expected to be completed by the 13th of November. A per unit budget of $200 was given by MechAdept. Manufacture time for each PCB was approximately no more than two weeks as a general estimation based on most PCB prototyping services. Mechanical prototyping was to be manufactured using 3D printing for its rapid prototyping capabilities and its availability.

4. Investigation of Existing Knowledge

3D Printing Considerations

GitHub user Arielle Hein (2016) posted a guide on 3D printed joints. A relevant joint mentioned was an annular snap-fit joint, see figure 2 below for an illustrated example (Troughton, 2008).

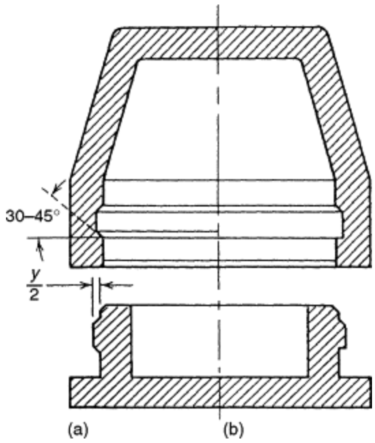


Figure 1: Annular snap-fit example

A research paper by Cauller, Smit, Zadpoor and Breedveld (2018) proposed ten design considerations and confirmed their validity by 3D printing a fully functional prosthetic hand. This paper was useful for manufacturability considerations during the prototype design. The design considerations relevant to this project were as follows:

1. Large Play – Additive manufacturing accuracies vary but this paper recommends a minimum tolerance of 0.5mm between parts to guarantee that they do not meld together while printing.
2. 3D printed surfaces – To avoid excess friction, it is recommended to print the sliding surfaces perpendicular to the print bed to avoid using support structure.
3. Support structure – It is important to provide space to allow for removing support structure material. Where support material is enclosed within a part, it recommended that the support should be minimally placed 1mm away from the part, and there should be minimally 4mm2 holes to fit common long-nose pliers in.
4. Compliant parts – This consideration refers to the elastic compliance of many extrusion materials. This is most useful to reduce play by causing the part to deform slightly to provide an elastic force back onto the part.
5. Cross-section of parts – It is important to consider the force direction of the parts to inform the cross-sectional area of the parts. Things to consider are presenting large surface areas in contact and avoid sharp corners to avoid stress concentrations.
6. Density of parts - Choose the material density (when 3D printing) according to the stress
7. Align the 3D printing plane with the dominant load.

Electronic Component Initial Research

<u>Micronas HAL 3725 2D Hall sensor IC</u>

This sensor was provided by MechAdept as a possible Hall sensor solution. It is an Integrated Circuit (IC) in an 8SOIC package supplied by 4.5-5.5V. The chip uses a ratiometric linear analogue signal output meaning it outputs an analogue voltage proportional to the magnetic field intensity. The magnetic field vectors are shown in figure 2 below, taken from the datasheet (Micronas, 2017). This variant is capable of measuring in all three directions.

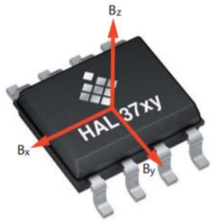


Figure 2: HAL 37 series magnetic field vectors

Infineon TLV 493DA 3D Magnetic Sensor with I2C Interface

The TLV 493DA is a 3D magnetic sensor was also provided by MechAdept as a possible sensor solution. It features a small TSOP6 package with an operating voltage range of 2.8-3.5V. It outputs A/D readings at up to 12bit data-resolution via an I2C interface. The magnetic field sensing directions can be seen in figure 3 below (Infineon, 2019). Hall sensors are commonly advantageous over other position sensors for their contactless nature with a magnet and repeatability of measurements.

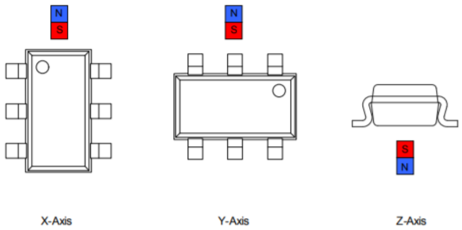


Figure 3: TLV 493DA 3D Magnetic Sensor field directions

MuRata SV03A103AEA01 Potentiometer

This potentiometer is a compact 10Kohm variable resistor housed in a SMD package with an effective angle range of 333.3°, however it has no mechanical limit. This sensor also outputs a ratiometric linear analogue voltage signal. It can be viewed in figure 4 below (MuRata, 2017). This potentiometer is being considered as there is one available to me from another project, it has a compact SMD package and is simple retrieve position data from.

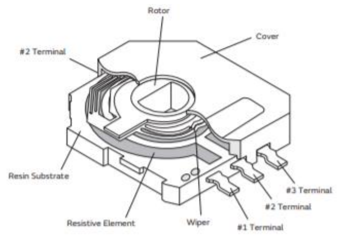


Figure 4: Construction view of the SV03A103AEA01 Potentiometer

Nordic nRF52840 Dongle

The nRF 52840 Dongle has been provided by MechAdept as a processing system solution that features Bluetooth and other radio frequency (RF) capabilities. It is a development dongle for Bluetooth 5, Mesh and other RF protocols. Relevant key features from the datasheet include Bluetooth Low Energy support, USB interface directly to the nRF52840 system on chip (SoC), 15 general purpose input/output pins, and 1.5V-5.5V operating voltage from USB. An image of the dongle can be seen in figure 6 below (Nordic Semiconductor, 2019).

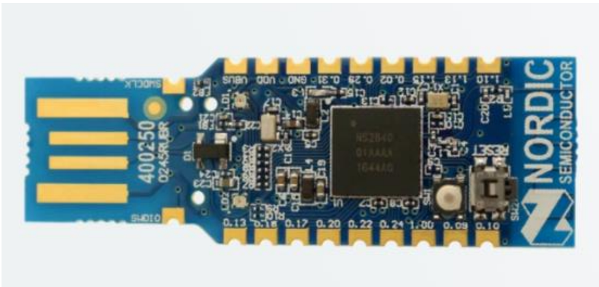


Figure 5: nRF52840 Dongle

I2C Protocol

A brief description of the I2C protocol is given in a previous assessment for another course (Stilwell, 2020).

> I2C, or Inter-integrated circuit, is a synchronous, half-duplex communication protocol. This means that it includes a dedicate clock wire to synchronise the communication between devices, and a single data signal wire which can carry information in both directions, one direction at a time. Each device on using the I2C protocol needs an address so that the master controller can select which device it wants to send or receive information from. A simplified diagram can be seen in figure [7] below (Circuit Basics, n.d.). Pullup resistors are often added to discharge the capacitance induced charge between the two wires.

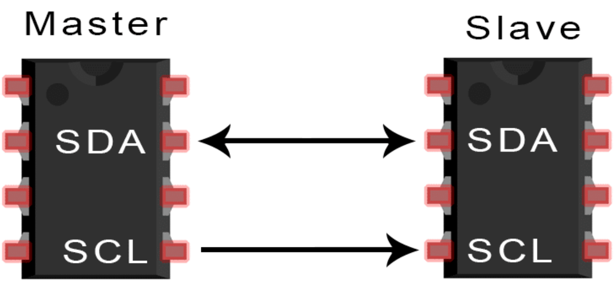


Figure 6: I2C connection simple diagram

<u>Bluetooth Low Energy</u>

Bluetooth Low Energy is a personal-area-network (PAN) technology which uses wireless communication and aims to offer Bluetooth capabilities at a lower cost and power consumption (Dastjerdi, 2016). It is commonly used in small devices because of this low power consumption. It uses a master and slave topology. Before two devices are connected, they are considered a central and peripheral device. After connection, the devices can be either client or server. The client and server can then be master or slave (Laura, 2014).

<u>Nordic Semiconductor's Software Development Kit (SDK)</u>

Nordic Semiconductor's SDK is a kit which provides developers with all the necessary examples and modules to build reliable and fully featured applications on their nRF52 or nRF51 series chips (Nordic Semiconductor, n.d.).

5. Project Design/Methodology

Final Prototype Overview

The final prototype can be seen in the images below:

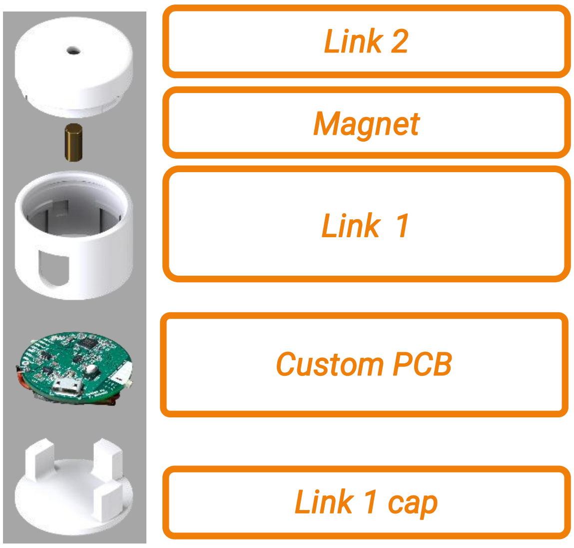


Figure 7: Exploded rendered view of joint module with image of PCB

Figure 8: Assembled joint module prototype

The prototype mechanical structure was fully 3D printed in three parts with PLA material and FDM 3D printing technology.

Link 2 holds the magnet with a press-fit and the male snap-fit tabs to interface with link 1.

The magnet is a diametrically magnetised N45 Neodymium magnet with a diameter of 6mm and a height of 12mm.

Link 1 provides the female slot for the snap-fit connection and an enclosure for the PCB and battery. The slot holds link 2 in place axially, allows relative rotation of both links and provides friction and hence a holding torque.

Powered by a Lithium-Polymer battery, the custom PCB features an nRF52810 Bluetooth-capable System on Chip (SoC), PCB trace antenna, 3D hall sensor for field position measurement, battery charging circuitry and a 3.3V voltage regulator. This will be discussed in-depth later in this report.

The joint is operated by rotating link 2 relative to link 1. Since link 2 holds a magnet, this causes a magnetic field to be rotated relative to the Hall effect sensor and thus the position can be measured between the two links. The PCB is then capable of sending this position information over Bluetooth Low Energy to a user terminal, which could be a mobile phone, PC, another joint module, etc.

Design method

The design philosophy consisted of designing and constructing a low fidelity prototype, labelled the 'V0.1 Prototype,' followed by the final prototype, labelled the 'V1.0 Prototype.' The reason for creating a V0.1 Prototype first was to test rapidly test design decisions using 3D printing and breadboarding for use in the V1.0 prototype. The design of the V0.1 protype aimed to implement the electronics on a breadboard using an nRF52840 dongle and the hall sensor was to be implemented in a mechanical structure separately, with connections to the breadboard.

The design of the V1.0 prototype aimed to implement the electronics on a PCB enclosed by the mechanical structure and thus the full system could be a standalone device.

Rather than designing these prototypes individually as planned, in practice they were designed interdependently.

V0.1 Prototype Design

Potentiometer Sensor Concept Idea

The first sensor concept idea was using the SV03A103AEA01 potentiometer. A simple test was performed to determine its suitability as a possible solution. A detailed overview of this test can be found in appendix A – Potentiometer test. It was found that a false assumption was made that the potentiometer output would reset to zero volts directly after approaching the limit of 333.3°. This meant that this potentiometer could not be used on its own, however there was the possibility of having two of them implemented with a phase shift to eliminate its dead range of approximately 36°. More details on this idea can also be found in appendix A, and the results can be found in the results section.

After analysing the potentiometer concept, the following conclusions were made:

- Simplicity – For electronics interfacing, the potentiometers were relatively simple to interface to a microcontroller with just one signal wire each with a power supply and ground. The signal could easily be interpreted with an analogue-to-digital converter onboard the microcontroller and some simple math to scale the A to D reading to the angle range, however having two sensors complicates it mildly. For mechanical interfacing, the potentiometers require many more parts such as bearings to protect their integrated bearings, mating the potentiometers to the main PCB, and mating the shaft to the opposite joint link in such a way that it can be easily assembled/disassembled. In conclusion, the electronics interfacing would have been simple, but a reliable mechanical interface would have been too complex.

- Precision – The sensor's datasheet specifies a linearity of ±2% of its full range (333.3°) which comes to ≈ ±6.7°. This does not meet the precision target for the project.

Hall Effect Sensor and Magnetic Mechanical Interface Concept Idea

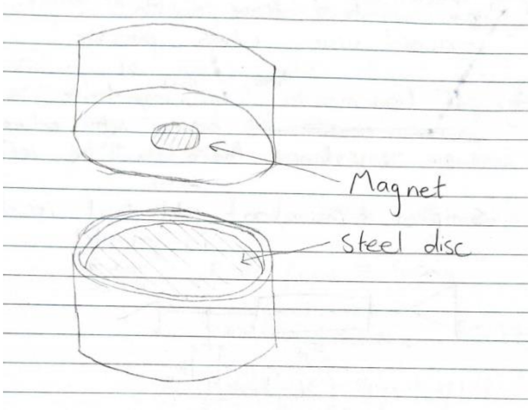


Figure 10: Magnetic mechanical interface idea drawing A

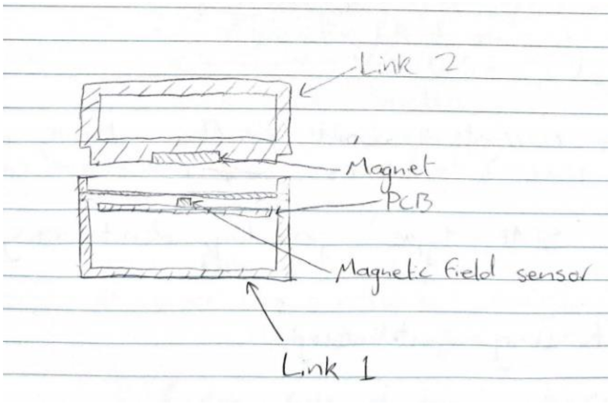


Figure 9: Magnetic mechanical interface idea drawing B

The second concept idea was using a magnet on a steel plate as a mechanical interface between two links while also providing joint position data using a hall effect sensor underneath the plate. The features of this concept were the following:

- The magnetic field was to be sensed through the steel plate. This had to be tested to validate the operation. More details on this test can be found in appendix B – Magnetic Mechanical Interface Concept.
- The method of coupling the joints axially was having the magnet directly pulling against the steel plate which was a simple solution. The rim profiles of link 1 and 2 would be such that the circular surfaces would keep the links concentric and aid with handling forces perpendicular to the axis of rotation. These forces, however, would increase the friction between the links.
- The holding torque would depend on the normal attraction force between the magnet and plate, the friction coefficient of the two surfaces, and the friction fit of the links' rims.
- This concept would allow for pulling the links apart with ease as only the magnetic force is holding them together axially. The magnet could easily be attached to link 2 with an adhesive.

In conclusion, this concept was worth taking further due to its simplicity and ease of assembly/disassembly.

Magnetic Interface Concept Taken Further

A Solidworks model was designed, modelled and exported for Frank to 3D print. The mock-up model included two links, wedges for temporarily fixing steel lids to the model, and a seat to easily hold the 3D magnetic sensor.

Link 1:

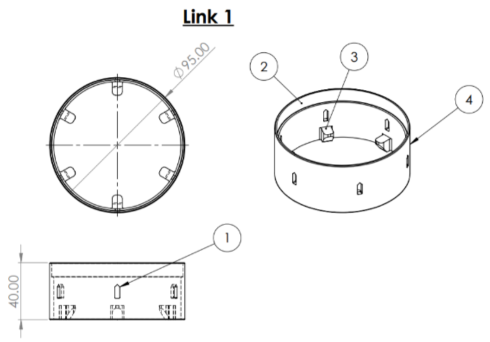


Figure 11: Magnetic interface concept implementation: Link 1 mechanical drawings

The first link had six small platforms (fig. above balloon 3) to hold the steel lids level. The link was oversized to fit the lids without any modification. This was to save time in assembly of the mock-up as it was a test. The part was designed with 3D printing in mind, so the overhanging platforms had a 45° slope to save on material, and the wedge holes (balloon 1) had to have 45° slopes to bridge the tops and a void needing support structures during printing. This was overlooked at first when the wedge-fixing method was implemented, so the wedges had to incorporate the same profile as the holes. A groove was added to the inside of the top rim for link 2 (balloon 2) to slot into for annular stability.

Wedges:

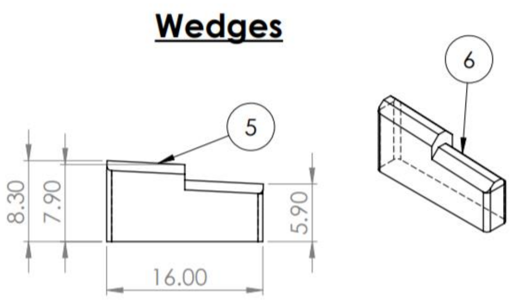


Figure 12: Magnetic interface concept implementation: Wedge mechanical drawings

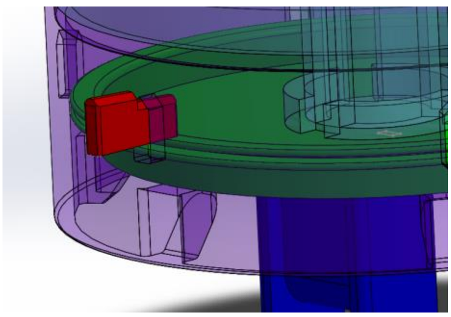


Figure 13: Wedge implementation in Solidworks model

The wedges (fig. above balloon 6) slot into the wedge holes in link 1 with the flat side pressing against the steel lid(s), and the trapezoidal profile pressing against the top triangular side of the hole. They had small slopes (balloon 5) to allow for printing inaccuracies and to ensure a snug fit when wedging in. There were two levels to allow for having 1 or 2 lids stacked up as seen in the figure above.

Link 2:

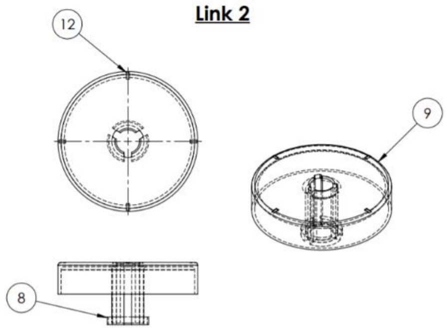


Figure 14: Magnetic interface concept implementation: Link 2 mechanical drawings

Link 2 was the top link which met the steel lids and slotted into the said groove in link 1's rim. There were difficulties with designing this part for 3D printing and thus a cap (fig. above, balloon 8) had to be printed separately and needed to be glued on. This cap came in contact with the steel lid and sat underneath the magnets. It had a thickness of 1mm which gave an uncertainty whether this would be a small enough gap for a good mate between the magnets and the steel lids. Magnets were held in place annularly by the cylindrical chute which the cap connected to. The chute and cap had three slots cut out of them to allow the magnets to be taken out. In this configuration, the chute was a weak point due to the 3D print layers being parallel to the torque direction and the lightweight connection to the link. This was constrained by the wedges taking up space in the joint. Four shallow slots were cut out of the top of the link (balloon 12) to easily find the centre of the part. This was necessary for attaching a lever arm to test the torque required to rotate the link.

3D magnetic test sensor seat:

Magnetic Sensor Seat

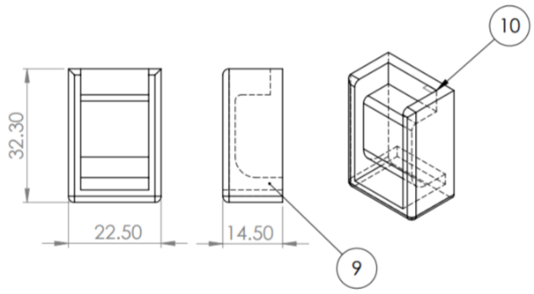


Figure 15: Magnetic Test Sensor Seat mechanical drawings

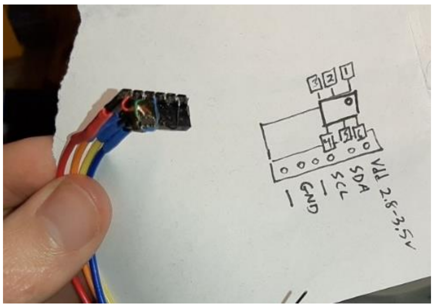


Figure 16: Magnetic test sensor

The magnetic sensor seat holds the given TLV493D test sensor. The small rectangle (fig. above left, balloon 10) corresponds to the position of the sensor package which can be seen in the figure above to the right. The sensor assembly was temporarily fixed into the seat with the cables facing downwards. The seat provided a channel for the wires to route through (balloon 9) for convenient cable management. The seat is used to secure the sensor and was attached to the bottom of the steel lids with the sensor approximately in the centre. The sensor's position is directly below where the magnet is mated to the lids.

The concept was 3D printed, assembled and tested. More details on the assembly and test process can be found in appendix C – Magnetic interface concept implementation and testing process. The sensor was attached to the bottom of the steel plate beneath the magnets and was connected to a teensy 3.2 microcontroller as was done previously in the initial concept idea. Initially, there were unexpected results. This was due to the south pole the magnet facing downwards in the z-direction. The Infineon TLV493D sensor library used was relying on the south pole of the magnet rotating about the z-axis with the south pole facing normal to the axis of rotation. The issue was resolved by placing the magnets on their side to orient the south pole correctly and this yielded the results seen in the results section 6. The C++ source code for this test can be also be found in appendix C. In conclusion, the magnet on plate mechanical interface provided an ease of assembly/disassembly, and the magnet placement and sensing method was proven to be worth using, however the interface lacked a simple method of providing a holding torque.

Snap-fit Concept

Mock-up mechanical interface concept 2

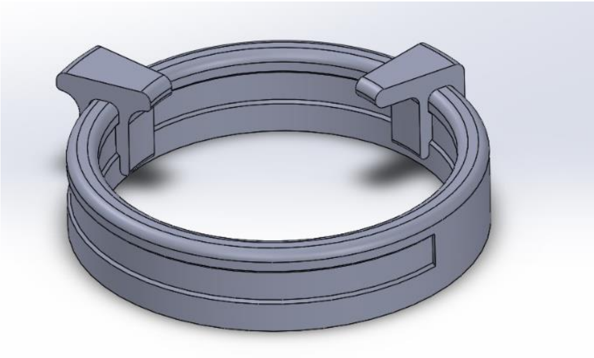


Figure 17: Snap fit mechanical interface mock-up Solidworks model

Figure 18: Snap fit mechanical interface mock-up 3D printed by Frank

A second mechanical interface concept was designed which made use of 3D printed cantilever snap-fits. The ring had an outer diameter of 40mm and a thickness of 3mm. This mock-up was 3D printed to test the suitability of the design. The ring was intended to be part of link 1 in the joint module design and the clips were for attaching to link 2 in the joint module concepts. Two female clips were printed; one with a two-sided clip and the other with one. See the images below showing the cross-sections of the clips and the male ring part. The clips had small platforms on the tops which were included to consider how they would connect to the rest of the link 2.

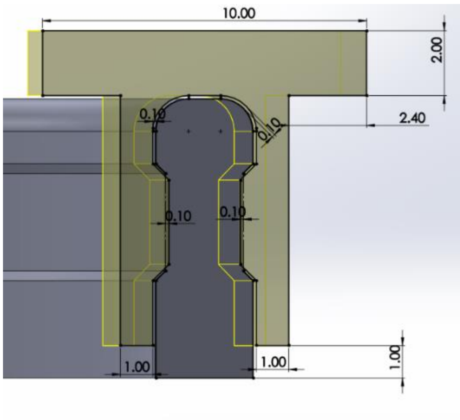


Figure 20: Snap-fit mock-up - two sided

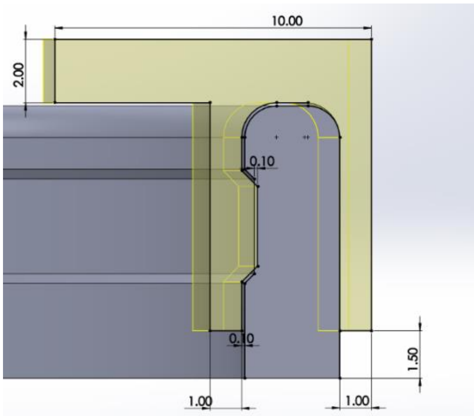


Figure 19: Snap-fit mock-up - single sided

The snap-fit was given by the profile of the annular groove. This mock-up used an angle of 45° on the upper and lower sides of the groove, allowing a two-way snap fit. This was important as the client requires easy assembly and disassembly of the joint. The walls of the clips were oversized by 0.1mm to apply a constant elastic force onto the ring to provide extra friction and to give a snug fit of the joint. The reason for producing clips with one- and two-sided variants was to test their effect on friction and the ease of assembly/disassembly. It was found that both clips were easy to press on and the single sided variant was easier to remove than the two sided one. The two-sided variant had a looser fit and was more difficult to remove than the other.

The choice of using cantilever snap-fits over an annular snap-fit was regarding the ease of disassembly of the cantilever fit. Having a smaller annular width of the clip allowed for more flex of the part and aided with FDM 3D printing constraints. It is difficult to bend an already curved surface, so keeping the annular width thin was needed for easy assembling/disassembling. The layers of the FDM 3D print need to be in-line with the direction of the applied force for a higher strength, otherwise the layers would be prone to delaminating and breaking the part. Having a small annular width meant that the print layers could mostly be in the correct direction.

Considerations:

- The snap fit worked as intended for the 3D printed implementation, however it was hoped that there would be more friction. This could have been added by increasing the oversizing of the clip walls to provide a greater normal force using the elasticity of the print.
- More clips could have been produced with different sizing for further testing.

In conclusion, the snap fit concept was taken further over the magnetic interface for its inherent resistive torque capability, rigid joint, and ease of assembly/disassembly. Although it wasn’t as easily disassembled as the magnetic concept, the simplicity of the resistive torque capability was more desirable.

<u>Snap-Fit Concept Taken Further as Prototype V0.1</u>

The previous snap-fit concept was taken further using the single-sided cantilever snap-fit clips. The finished V0.1 prototype can be seen in the image below.

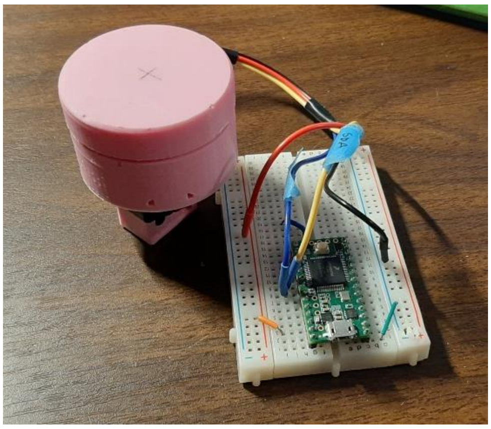


Figure 21: Prototype V0.1 assembled

Mechanical Interface

A new version of the snap-fit clips was designed and implemented. Three of them were 3D printed separately, which would then need to be glued together then glued to link 2. They can be seen in the Solidworks assembly in the image below:

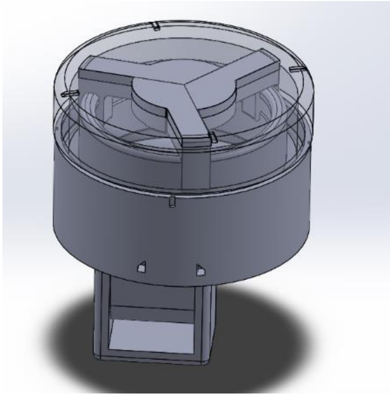


Figure 22: Prototype V0.1 3D model

The reason for 3D printing three separate clips was to maintain the strength of the 3D print by having the layers aligned with the direction of force exerted by clipping the clip onto the male part.

The centre of the three clips is where the neodymium disc magnets were to go, before the previous issue was found with the magnet orientation. An extension had to be printed to hold the cylindrical, diametrically magnetised neodymium magnet at a close distance from the magnetic sensor. This part also had to be glued onto the clip assembly. This can be seen in the image below.

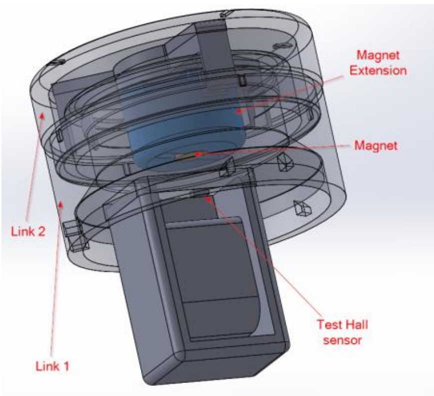


Figure 23: V0.1 Prototype Model Showing Magnet extension

The holding torque for this mechanical interface was tested and the method can be found later in this section.

Sensor Interface

The same method as in the hall effect sensor and magnetic mechanical interface concept idea mentioned previously was used for this prototype. However, this one includes the diametrically magnetised magnet.

Embedded Software

The software used for this was also the same as in testing. It was implemented using the Teensy, which differed from the plan of using the nRF52840 Dongle. The dongle was not used because the time was better spent learning the nRF52 platform straight on the V1.0 prototype's custom PCB.

Prototype Conclusion

This prototype was useful for applying and testing a form of snap-fit concept. The links were easy to assemble/disassemble, and it almost provided a sufficient holding torque because of the three-cantilever snap-fit design. However, this design was complex with a large number of parts which needed to be cleaned up and glued together. The size of the clips was also at the limit of the 3D printer's capabilities and strength of the PLA. This means that the prototype could not be miniaturised much further with this design and construction.

V1.0 Prototype Design

The V1.0 Prototype mechanical structure was based heavily off the work done for the V0.1 Prototype. The diameter of the joint module was limited by the diameter of the PCB.

Mechanical Structure and Interface

Multiple iterations of both link 1 and 2 were designed and 3D printed. A test was printed for

Mechanical Structure V1.0

The first test model of the V1.0 mechanical structure was 3D printed and can be seen in the images below. This design trialled the idea of using small 3D printed pegs. They were hexagonal and designed to fit into a circular hole located at the bottom of link 1. This hole was limited to the thickness of link 1's wall.

Figure 25: V1.0 Test print

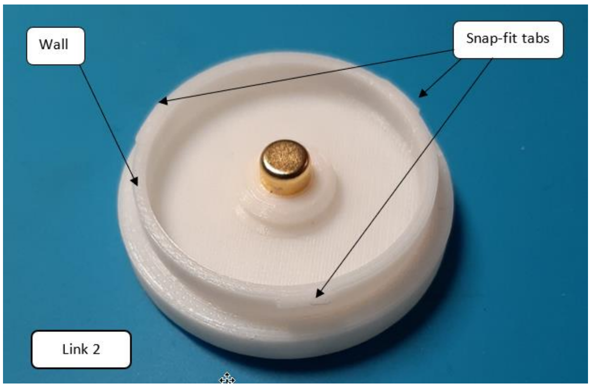


Figure 24: Link 2 V1.0 with magnet installed

Design notes:

- The 3D printed pegs pushed to the limits of the 3D printer due to how small they were. They also did not fit into the intended slots in link 1. Frank recommended designing for non-3D printed pegs or rods.
- The links snapped together but were difficult to take apart. The stiffness of the wall on which the snap fit tabs are connected to was greater than expected.
- The magnet pressed into link 2 perfectly and was held in place tightly.
- The snap fit was invisible with the links assembled so it was difficult to analyse the fit.
- The friction was tested along with prototype V0.1 and the results can be seen in table 1 in the results section 6.

<u>Mechanical Structure V1.1</u>

This iteration took the findings from the previous, and the PCB and battery were able to be installed. See the image below for this.

Figure 26: Link 1 V1.1 test fit with PCB and Battery

Design notes:

- The wedge design was updated to be wider, while conforming to the diameter of the wall for more strength and to rely on the sides for the friction fit rather than the previous small circular hole. This design worked successfully in holding the PCB in place with a friction fit as seen in the image above.
- The friction was increased by increasing the width of the snap-fit tab relative to link 1's slot to provide extra radial pressure between the links see the image below for a cross section.

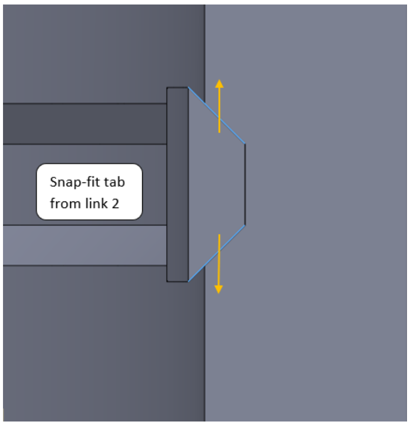


Figure 27: Cross-section view of link 2's increased tab width relative to link 1's slot

- Notches were cut out of link 2 where the joints meet on the outside rim to easily fit a flathead screwdriver in to aid with disassembly. See the image below for this.

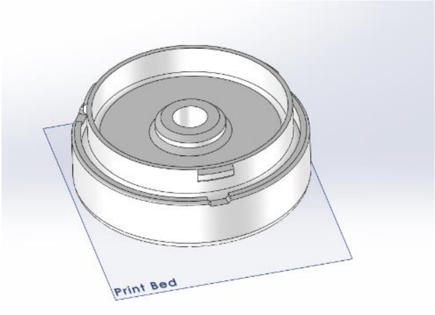


Figure 28: Link 2 V1.1

- The wall thickness of link 2 was decreased and the distance of the tab from base of the wall was increased as seen in the image below. However, this decreased the strength of the link considerably and resulted in layer delamination when disassembling the links.

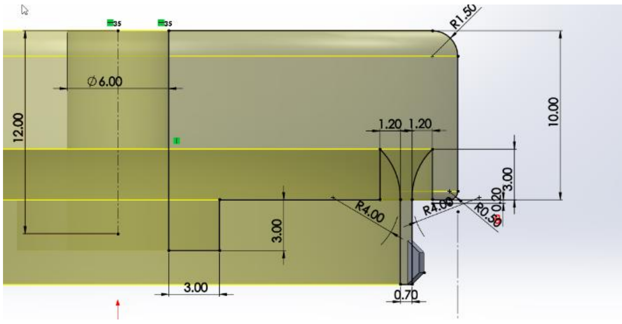


Figure 29: Link 2 V1.1 Cross-section

- The holding torque for this iteration was not tested due to the broken link 2.

Mechanical Structure V1.2

This iteration involved leaving the Link 1 unchanged and link 2 was altered slightly to increase the wall thickness for greater strength. In practice, however, link 2's wall was still too thin, and the layers delaminated again with disassembly.

A new part was designed and printed which was the link 1 cap. It used the previous working peg design, but they were implemented as a cap rather than pegs. This worked successfully in holding the PCB and battery inside link 1.

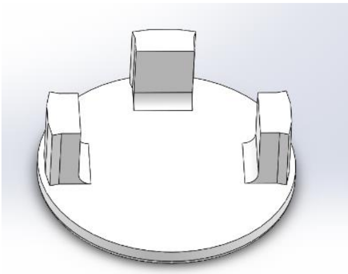


Figure 30: Link 1 cap

Mechanical Structure V1.3

This iteration was intended to be one which would be used to test the joint's position readout angle against the actual angle. However, the software functionality was not completed for this. The joint was modified by adding steps for a custom laser-cut protractor to slip onto. The implementation can be seen in the image below. The protractor was laser-cut at WelTec with a ULS V4.6 laser cutter.

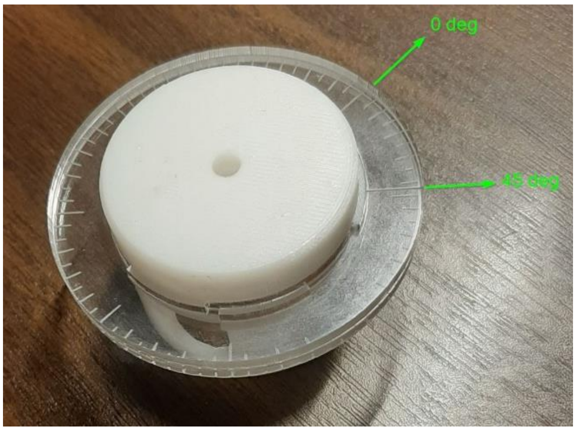


Figure 31: Iteration V1.3 with custom protractor

Along with the modification for holding the protractor the wall thickness was also increased again. This iteration was one of those tested for its holding torque and performed the best. The results can be found in table 1 in results section 6.

V1.0 Prototype's Mechanical Structure Conclusion

In conclusion, this prototype was a major improvement over the V0.1 prototype. It was compact and held the PCB well. It was harder to assemble/disassemble than V0.1 in the sense that a tool was needed to pry it open, but this design could be miniaturised more effectively.

PCB Design

The first step of designing the PCB was component selection. The features needed included the following:

- The position sensor was selected already which was the TLV493D 3D hall effect sensor.
- The nRF52 series SoC had to be selected with all its required passive components.
- A power supply for converting a battery voltage into a stable voltage for the SoC and other components
- A system status indicator, which was in the form of an RGB LED to indicate different states that the system is in using different colours. This was for a distinctive visual status and it was also a strong recommendation from Frank.
- A battery was needed to eliminate power supply wires for the joint module. A rechargeable battery was desired for sustainability considerations and easy maintenance in the form of charging the battery when needed, rather than installing a new one.
- An onboard battery charger was needed for ease of maintenance – a single rechargeable battery would be left in the joint module.

- A power point was needed to charge the battery.
- An antenna was needed for the SoC's Bluetooth Low Energy capability.
- A programming point was needed for programming the SoC.
- A reset button was needed for fast resetting of the SoC.

nRF52810 SoC

The only sensor required was the TLV493D 3D Hall effect sensor which communicated using an I2C interface. Therefore, the system

Rough Estimate of Maximum Current Consumption

With an estimated absolute maximum current of 10mA from the TLV493D sensor, 25mA from the SoC and 5mA from the RGB LED, the total estimation came to 40mA. These values came from the associated datasheets.

Battery

A nominal 3.7V Li-Po (Lithium-Polymer), 600mAh battery from AliExpress seller EHAO was used for its small physical size of 22mm X 25mm X 11mm and large capacity. At a worst-case scenario, using the previously mention max current of 40mA, the longest operation time of the board would be 15 hours using the full battery capacity (600mAh/40mA). However, it is unlikely that the current consumption would get this high. This battery is rechargeable, and the seller was recommended by Traveler.

Battery Charger Integrated Circuit (IC)

The MCP73831T Li-Po Charge Management controller by Microchip Technology was used to safely charge and monitor the battery. The SOT-23-5 package was chosen for its small overall footprint size of 3mm X 3mm and its programmable charge current. The chip series was also recommended by Traveler.

Power Point for Charger

The 10118192-0002LF Micro B USB female USB port by Amphenol ICC (FCI) was used with the Micro B SMD package as this is a common connector type and it is convenient to charge via a USB cable. The SMD type was chosen to avoid risk with incorrect footprints using the through-hole type. There was still risk with this package but the SMD type could be reworked more easily compared to the thru-hole type. It was also used so that the PCB could sit flat on a surface, as the Li-Po battery was to be seated directly beneath it.

Voltage Regulator

Research was needed to determine how the battery power would be supplied to the system. A tutorial by Maxim Integrated (2001) outlined six regulator topologies for battery powered systems; Linear regulator, Charge Pump, Step-down (Buck), Step-up (Boost), Inverter and Flyback. The main takeaway from this tutorial was that voltage supply must be higher than the output voltage for most of the regulators. An external voltage regulator with a fixed output voltage of 3.3V was chosen as they are small, cost effective, and simple. Switching regulators were unsuitable because of their larger size, price, noisiness and complexity.

The TC1015 fixed 3.3V Low Dropout Voltage regulator by Microchip Technology was chosen with a SOT-23 component package. Its 100mV dropout voltage was desirable to get the most out of the battery. The SOT-23-5 package option was used for its small overall footprint size of 3mm X 3mm.

RGB LED

The same RGB LED was used as was on the nRF52840 Dongle board which was the APHF1608LSEEQBDZGKC by Kingbright using the 0402 package. This was chosen for its small package overall footprint size of 1mm X 0.5mm

RF Antenna

A PCB trace antenna was used from Nordic's 'nRF Misc' eagle library for its low cost, simple tunability, and ease of manufacturing. The resistive impedance of a PCB trace antenna can be tuned by incrementally removing the end of the antenna, rather than needing to constantly swap resistor components for a chip antenna.

Debug Point

The Serial Wire Debug (SWD) interface was used for its simplicity (only needs data signal and clock signal wires). It could also be implemented as a small size because of this. Six pin holes were used with a 1.27 mm pitch in a two by three configuration was used with 0.65mm holes as recommended by Traveler. The Segger Embedded Studio IDE also supports SWD natively with J-link debuggers.

Eagle Schematic

Once the components and packages were selected, the schematic was produce using Autodesk eagle. This schematic was spread over three pages; one for SoC related components, one for power related components and one for sensor related components. The first sheet was largely given as reference circuitry from Nordic Semiconductor's GitHub repository and can be seen in the image below. The SoC's datasheet strongly recommended using their layout for the high frequency parts for the best performance. The only changes to this circuitry were the connections to the added components on the other sheets.

One of the most important aspects of this circuit for the Radio Frequency (RF) driver is the impedance-matching circuit in the bottom right of the image, consisting of the 0.8pF capacitor and 3.9nH inductor. This circuit matches the driver's impedance to 50Ω to reduce signal reflections in the transmission line and antenna. 50Ω is a common impedance for 2.45GHz antennae.

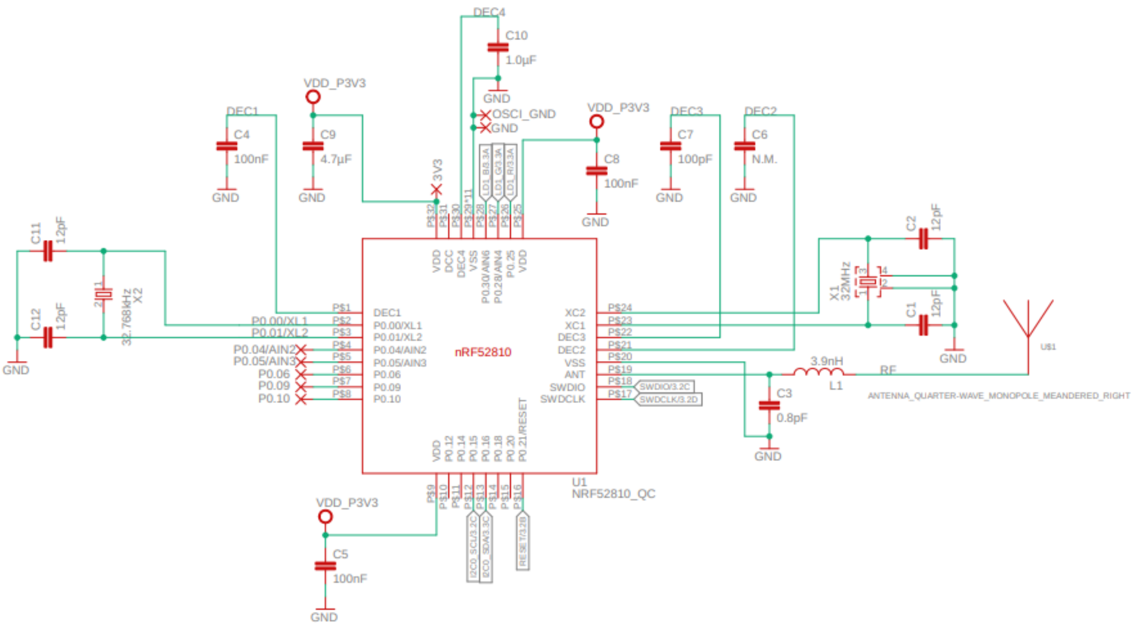


Figure 32: SoC section of Schematic

The second sheet can be seen in the image below. The voltage regulator circuit was implemented as recommended from its datasheet. The micro USB has only its power pins connected as this SoC does not have USB data lines. Ferrite beads were used to snub high frequency noise which is common in USB power supplies from PCs and Laptops for example. The Li-Po charger IC was also implemented as recommended in its schematic, only a 10kΩ programming resistor was calculated using the following equation from the MCP73831 charger IC's datasheet:

$$I_{REG} = \frac{1000V}{R_{PROG}}$$

Where:

R_{PROG} = kOhms

I_{REG} = milliampere

Figure 33: MCP73831 programming resistor equation for charge current

The charge current was chosen to be 100mA as more than this would require additional efforts for heat dissipation.

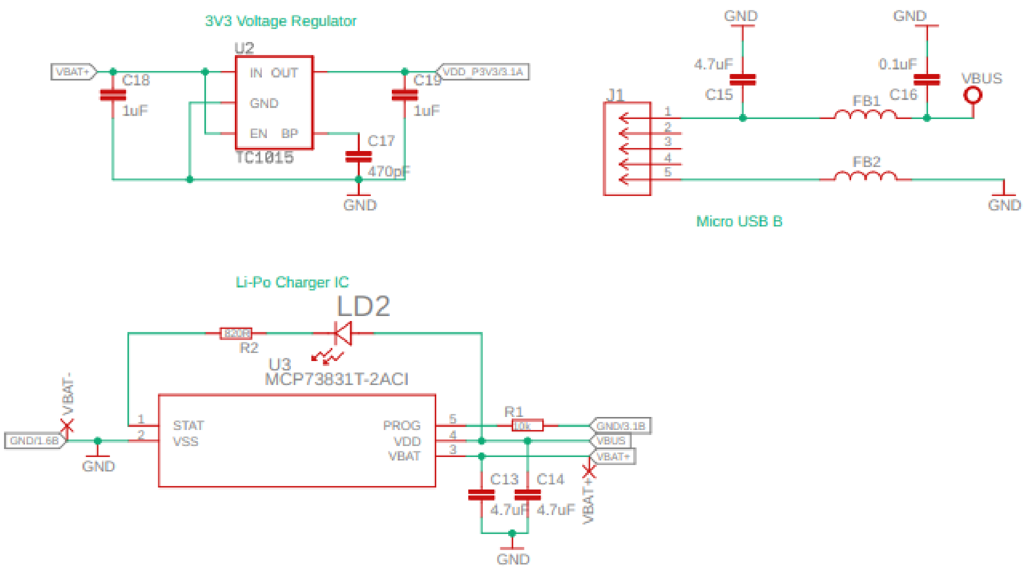


Figure 34: Power-related components section of schematic

The third and final sheet can be found in the image below. Current-limiting resistors were implemented for a limit of 2mA per pin. These vary as the forward voltage of each colour varies. It is a common anode LED, so the anode is connected to the 3V3 supply and the cathodes connect to the SoC pins; 0.25, 0.28 and 0.30 via the resistors. They are driven by pulling the pins low. The pushbutton was connected in a pullup configuration, so the SoC reset pin; 0.21 registers this as 'active low' meaning it registers a button press as a low voltage signal. It should be noted that the implementation of the button in this schematic was incorrect as its pins; 1 and 3 should not be connected do GND.

The TLV493D sensor was implemented as recommended in its datasheet with two bypass capacitors in parallel and using 1.2kΩ pullup resistors on its I2C data and clock lines.

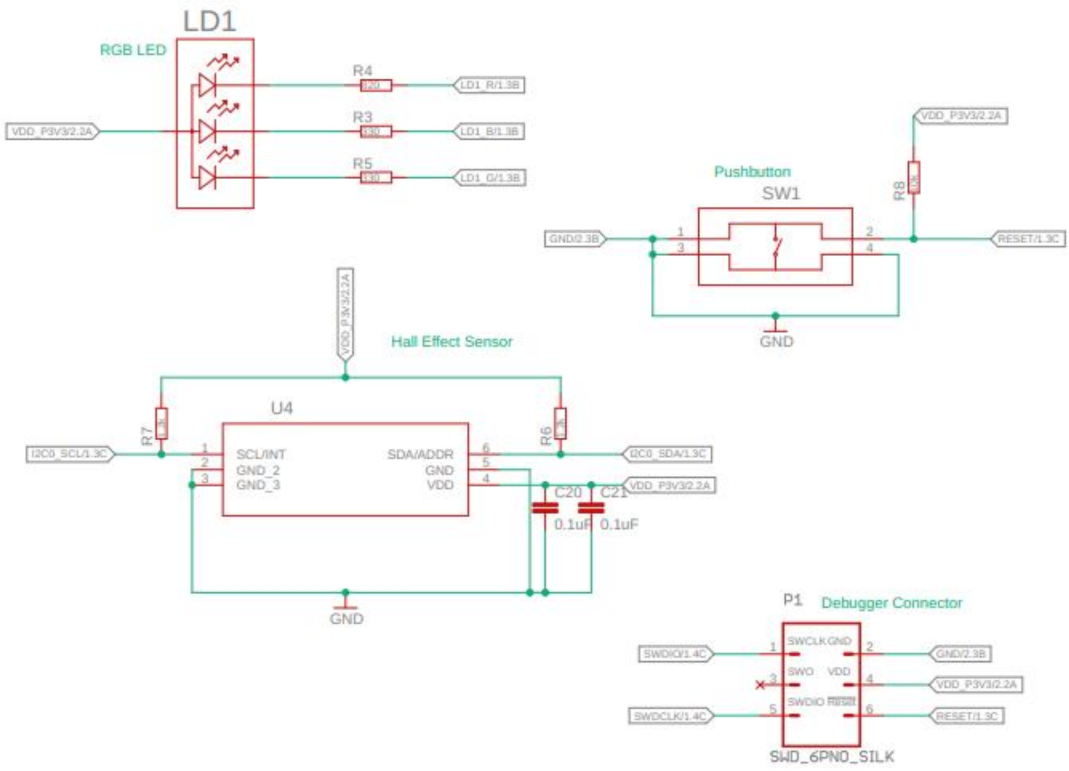


Figure 35: Sensor-related components section of schematic

Eagle Layout

The final layout of the PCB can be seen in figure 37 below. Blue boxes have been placed on the image to highlight the different sections. The board was 35mm in diameter and features a two-layer stack-up. The top (red) layer was for signals, power, and components and the bottom (blue) layer was a full ground plane. No through hole components were used and no components were placed on the bottom of the board for a simpler assembly process and to have a flat surface at the bottom, where the battery would be placed.

PCBZone's manufacturing constraints had to be respected during the layout of the PCB. These can be found in the image below (PCBZone, n.d.). These rules were implemented using AutoDesk Eagle's design rule checker functionality, where it would be configured using these rules.

Standard PCB Specifictaions

1.6mm FR4 Tg 140
1oz Copper
White Component Ref.
PTH Slots
Routed profiles.
Flying Probe Test

Basic Design Rules

Minimum Track Size	0.15mm (0.006")
Minimum Gap Size	0.15mm (0.006")
Minimum Drill / PAD Size	0.25mm / 0.45mm
Minimum Silk Screen Line	0.10mm (0.004")

Figure 36: PCBZone manufacturing considerations

The power-related components; micro USB port, battery charger and terminals, voltage regulator and all their passive components were grouped up in one area to minimise the power trace lengths. The power traces' widths were kept wide at 0.5mm to keep impedances low. Every ground SMD pad had a via placed next to it which barrowed down to the ground plane. This was to provide an easy return current path to avoid disturbances to other components. The hall effect sensor was placed first as it had to be in the centre of the board to line up with the magnet to make the maths simpler for position measurement.

The components found in the nRF52810 SoC group were all placed as given in the reference circuitry for the reasons mentioned earlier.

The battery charger was laid out as recommended in its datasheet. Large areas of copper were placed at each pin to aid with power dissipation by spreading the thermal load over a greater area. Many vias were also used to conduct heat to the bottom layer.

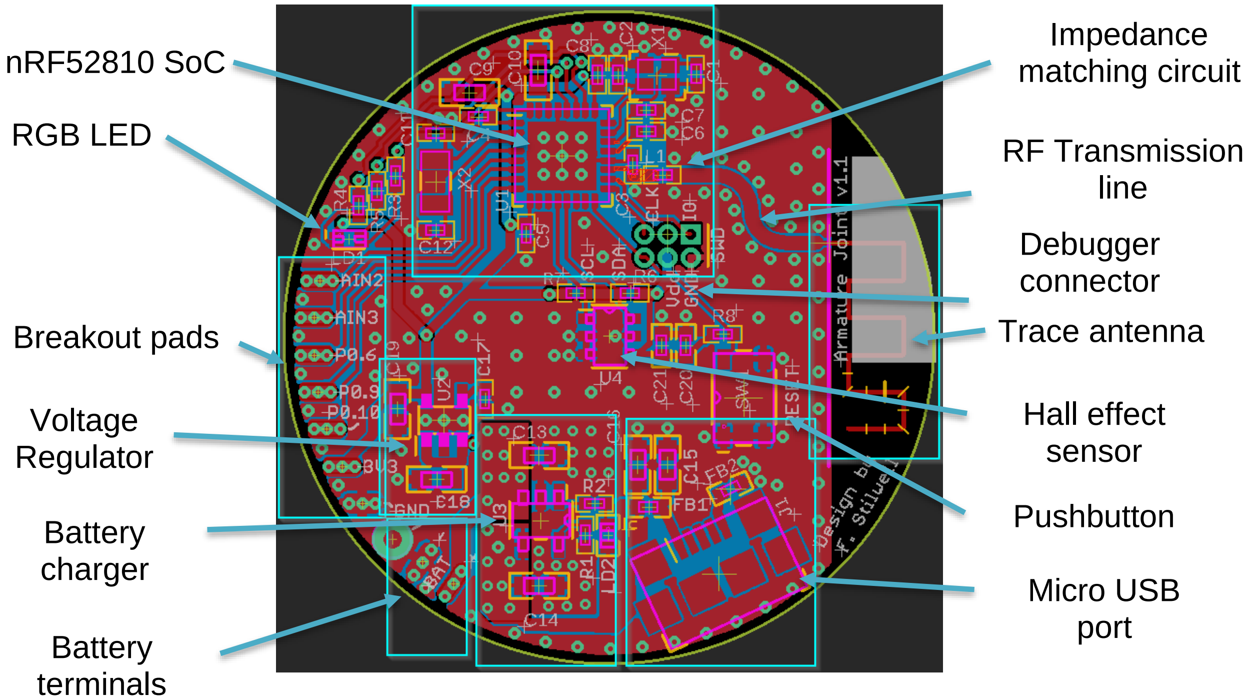


Figure 37: Armature joint PCB layout in Autodesk Eagle

The RF parts of the PCB had their own considerations. The transmission line had to have an impedance of 50Ω, so its trace width had to calculated. The image below shows the calculation done using AppCAD. AppCAD is a free tool by Avago Technologies that is dedicated to assisting with RF, Microwave and wireless design applications.

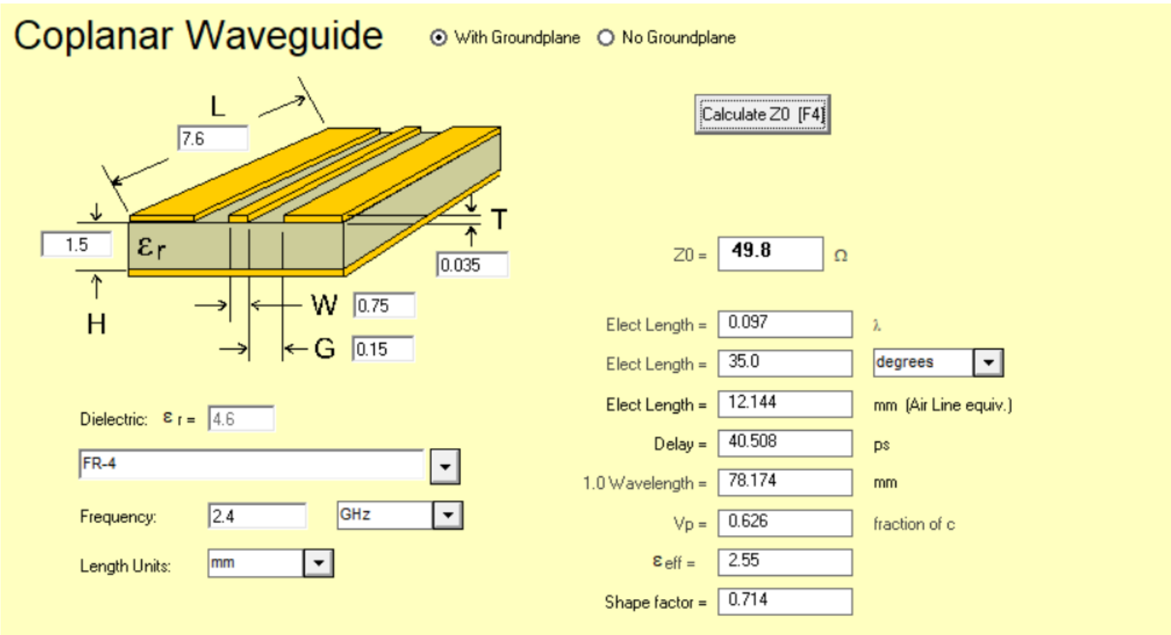


Figure 38: Trace impedance calculation with AppCAD

FR-4 material was used for the PCB which is a standard low-cost material. The height (H), trace thickness (T), and gap distance (G) was given by PCBZone, who was to manufacture the board. The impedance (Z_0) was calculated to be 49.8Ω through trial and error by altering the width until 0.75mm was found.

A 'fence' of vias was placed on the outsides of the transmission line and board to short induced currents to ground (Bailey, 2011). Vias were also placed liberally across the full area of the board for this reason.

Manufacturing

Once the board was finished and errors were checked, the hardware files were exported and sent to PCBZone by Traveler at MechAdept after resolving some minor issues with the solder mask configuration. The standard PCB manufacturing files are the 'gerber' type.

Assembly

Six of the boards were assembled at once at MechAdept's office with the help of Traveler. A confidentiality agreement had to be signed before entering their office to protect their clients' privacy and MechAdept's Intellectual Property (IP).

The 'reflow soldering process' was used to assemble the PCB's. This involved applying a solder paste to the boards' pads using a custom stencil ordered from PCBZone, placing components on all six boards under a microscope as the parts were so small, and then placing them in a reflow oven for the final soldering process.

All boards were checked and errors from the process were discovered and fixed. Some notable errors were 'tombstoning'. This is where only one pad of a component is grabbed by the solder and causes it to stand up like a tombstone. An example of this can be seen in the image below of board serial number two.

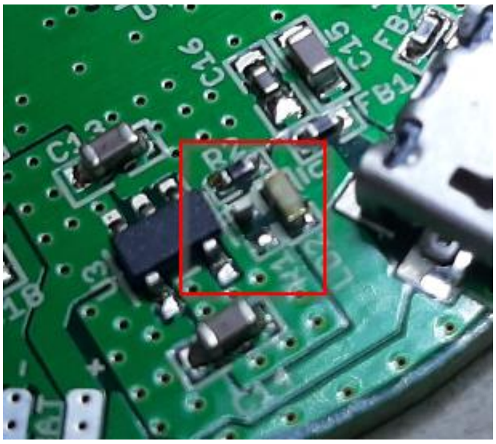


Figure 39: Tombstoned resistor assembly error

Embedded Software

Nordic Semiconductor's Software Development Kit was used to aid learning of the nRF52 platform through uploading and studying example programs. The plan for the software was to use an nRF52840 Dongle as a central Bluetooth device which connects to a PC via USB, and to the armature joints via Bluetooth. A modified version of a virtual com port example would be used to communicate with the PC via UART (Universal Asynchronous Receiver-Transmitter). The Bluetooth connectivity of this program would have to be modified to be a central Bluetooth Low Energy (BLE) device rather than a peripheral and the Nordic UART BLE service would be used.

The armature joints would have custom programs set as BLE peripheral devices and connect to the dongle. These programs would use aspects of Infineon's TLV493D library to communicate with the position sensor. This library, however, is written in C++ so would need to be converted to C to communicate with the SoftDevice. The SoftDevice is a BLE central and peripheral protocol stack solution which all the nRF devices use for their BLE and other radio programs. The SoftDevice is used with its Application Programming Interface (API) which is implemented in the main program for interfacing with the SoftDevice's functionality.

The program can be uploaded to the armature board via a J-Link debug probe which is easiest done using the Segger Embedded Studio IDE (Integrated Development Environment). The debug probe needed a cable to be modified to fit the six-pin header used in the armature joint PCB. This was also used to debug programs by stepping through the programs step by step using Segger Embedded Studio and seeing exactly where errors occur.

Examples from the SDK had to be modified to be compatible with the armature joint PCBs. The memory sections had to be changed for the nRF52810 chip to accommodate the programs and various device setup files had to be swapped for nRF52810 ones.

BLE examples were successfully uploaded to two different armature joint PCB's and the range was tested using a simple BLE blinking led example on board serial number 6. The joint was connected to an android phone via the nRF connect app and the LED was commanded to change colour at increasing distances.

A BLE UART program was also modified to send continuous stings over BLE using a timer interrupt.

Testing

Holding Torque of the Mechanical Interface of the Armature Joint

The mechanical interface holding torque was tested for the both V0.1 and V1.0 prototypes. A simple test using a makeshift apparatus was setup using the diagram below.

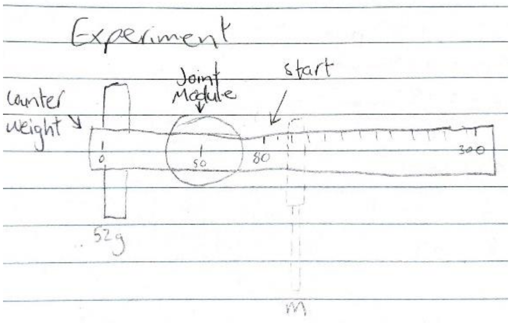


Figure 40: Drawing of torque test setup diagram

The mounting point was arbitrarily chosen to be the 50mm mark on a ruler, and a counterweight of 52g was placed a distance which balanced the torques about the mounting point. The test was done for three separate test articles; Link 2 V1.0 and V1.3 of prototype V1.0 and link 2 of V0.1 prototype. Link 1 of each was mounted to a table vice. Link 2 of each test article was mounted to the ruler with its centre aligned with the 50mm mounting point using hot glue. The setup can be seen in the image below.

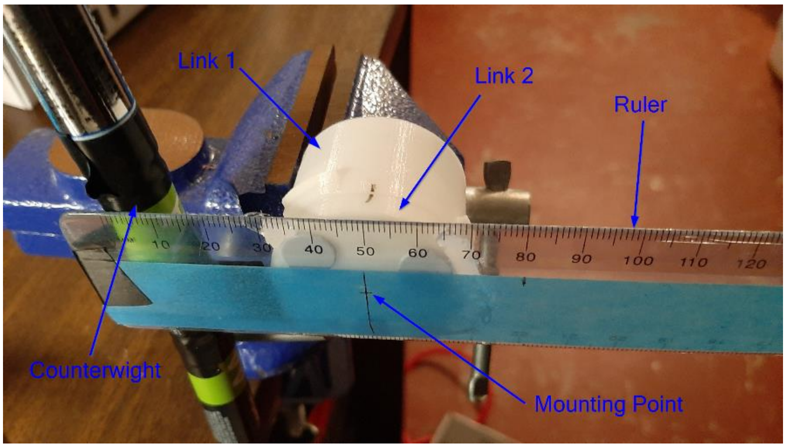


Figure 41: Torque experiment setup with Prototype V1.0 using Link 2 V1.4

The theory was that the holding torque would be calculated using the mass and distance along the ruler at which link 2 of the joints began to rotate relative to link 1 using the equation $Torque = mgd$ where m is the applied mass (kg), g is the gravitational acceleration (m/s^2) and d is the distance (m) of the mass on the ruler relative to the mounting point. This assumes a perfect balance of the ruler about the mounting point. The test results can be seen in the table 1 in the results section 6.

PCB Testing

Resistance tests have been done for all six manufactured boards and the results for which can be seen in appendix D – Armature Joint PCB Testing as at 11/11/2020. The schematic error was found for the button connections during testing and some of the LEDs were assembled with the wrong orientation. The most notable errors found were a battery charger IC not working on board serial number 2, and serial number 5 indicated

BLE advertisement but could not be found on the nRF Connect android app. However, the rest of the boards were found to be programmable and their Bluetooth functionality was working.

Problems and Difficulties Encountered

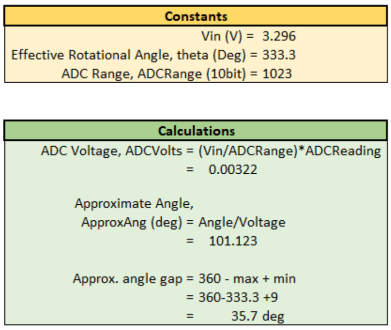
One of the major issues in the PCB design was that the nRF52840 chip was originally planned to be used, however it was discovered during the design phase that it was out of stock and had a wait time of approximately 26 weeks. After a discussion with Frank and Traveler, a different SoC had to be selected. The nRF52810 SoC was selected for its small number of GPIOs as the project did not require many. It was discovered later on that choosing this chip from the lower end of the nRF52 chip series meant that the memory size was too small for many of the example programs which made it difficult to construct programs at a beginner level. Another issue with the original nRF52840 SoC was that its package required manufacturing capabilities greater than PCBZone could produce. After a discussion with Traveler it was determined that PCBOne's service in China would be used as MechAdept had been happy with their services in the past. Hoever, with the change to the nRF52810 chip, the package was simpler and therefore PCBZone could manufacture it.

The level of knowledge required to write the embedded software was beyond the scope of the B. Eng. Tech program. Due to time constraints and the scale of the electronics and mechanical design that was needed in this project, there was not enough time to learn the skills required on top of the work required by the project.

A significant programming challenge appeared where example programs would compile but throw a 'unplaced memory section' error during the linking process. The solution to this was not able to be found online. To find the error, the Segger Embedded Studio project file was compared to a working project file using a program called 'Meld.' It was found that something within the project renamed the memory sections. After this was fixed, the first successful upload to the armature board occurred.

6. Results

Potentiometer Concept Idea Results



Constants
Vin (V) = 3.296
Effective Rotational Angle, theta (Deg) = 333.3
ADC Range, ADCRange (10bit) = 1023

Calculations
ADC Voltage, ADCVolts = (Vin/ADCRange)*ADCReading
= 0.00322
Approximate Angle,
ApproxAng (deg) = Angle/Voltage
= 101.123
Approx. angle gap = 360 - max + min
= 360-333.3 +9
= 35.7 deg

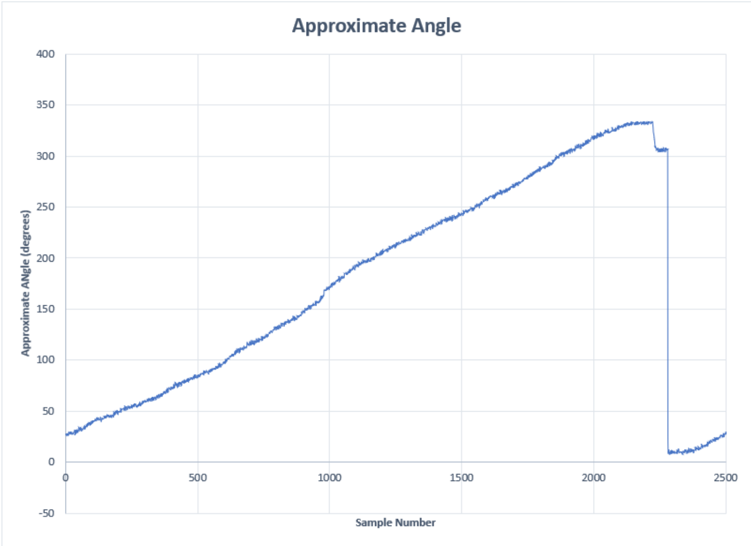


Figure 42: Potentiometer angle range test results

Magnetic Mechanical Interface Concept Test Results

The results of the magnetic mechanical interface concept test can be found below. Channel 1 (red) shows the over magnetic field intensity in milli-Tesla's (mT). Channel 2 (blue) shows the relative angle of the joint in degrees.

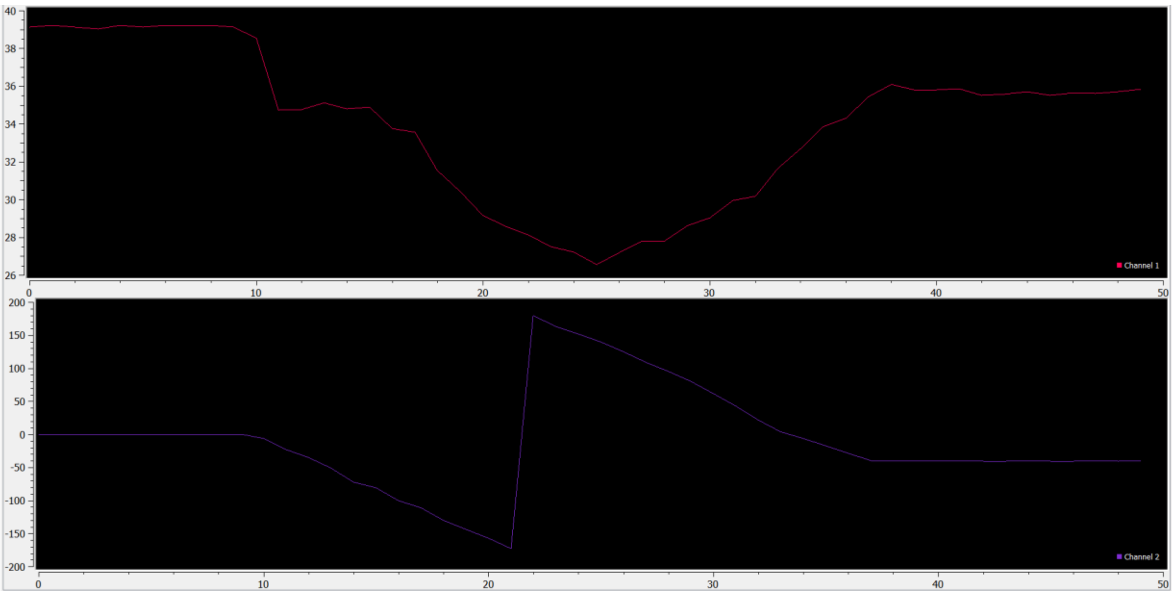


Figure 43: Magnetic Mechanical Interface Concept Test Results

Operating Torque Test Results

The following table shows the final results of the operating torque of the prototype V0.1 mechanical interface, and two Link 2 iterations of the prototype V1.0 mechanical interface.

Table 1: Operating Torque Test Results

Test Number	Link 2 version number	Mass, m (kg)	Average distance, d (m)	Torque, T (Nm)
1	0.1	0.085	0.182	0.152
2	1.0	0.085	0.1538	0.128
3	1.3	0.154	0.237	0.358
			MG-90S Servo Benchmark:	0.177

Assembly Drawing

A drawing of the mechanical assembly can be seen in the image below. A full-sized version of this can be found at appendix I. This drawing shows the overall dimensions of the V1.0 prototype; 40.2mm diameter and 37.2mm height.

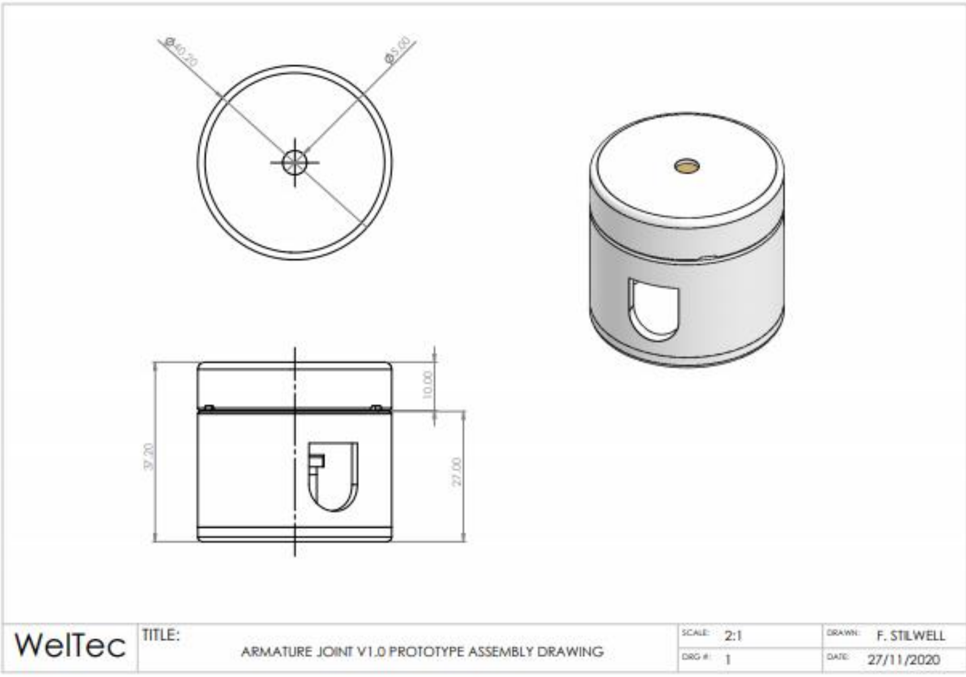


Figure 44: Prototype V1.0 Assembly Drawing

Project Expenses

Table 2: Project Expenses

Armature Joint Module Cost - Total Project					
Item	**Quantity**	**Order cost excl. shipping**	**Total order cost**	**Single item cost excl. shipping**	**Total single item cost**
Neodimium Magnets	3	$ 9.42	$ 21.49	$ 3.14	$ 7.16
Li-Po Batteries	2	$ 17.58	$ 28.66	$ 8.79	$ 14.33
Electronic components order*	6	$ 97.76	$ 97.76	$ 16.29	$ 16.29
PCBs manufactured +1 stencil	6	$191.76	$ 201.76	$ 31.96	$ 33.63
	Total:	$316.52	$ 349.67	$ 60.18	$ 71.41

*The electronic components order included all necessary components and spares for six PCB’s

V1.0 Prototype Notable Results

- Six boards were manufactured and assembled
- Bluetooth Low Energy (BLE) basic examples were uploaded and working on board serial numbers; 1, 3, 4 and 6
- Serial number 6 remained connected and operating with LED blinking BLE example program from over 10m away. The armature can be seen connected to the nRF Connect app in the image below.

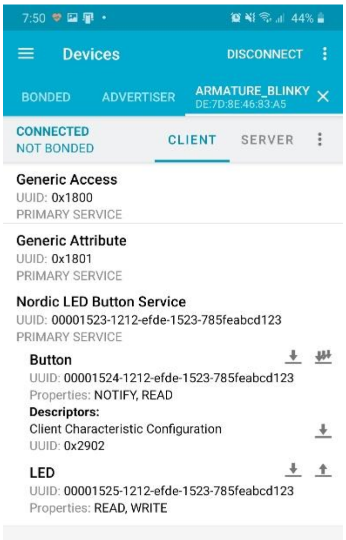


Figure 45: Android phone connected with Armature board with a Blinky program with nRF Connect

- An I2C scanner program was implemented on serial number 6 which successfully detected the address of the hall sensor (0x5E). This can be seen in the screenshot below from a debugging session in Segger Embedded Studio.

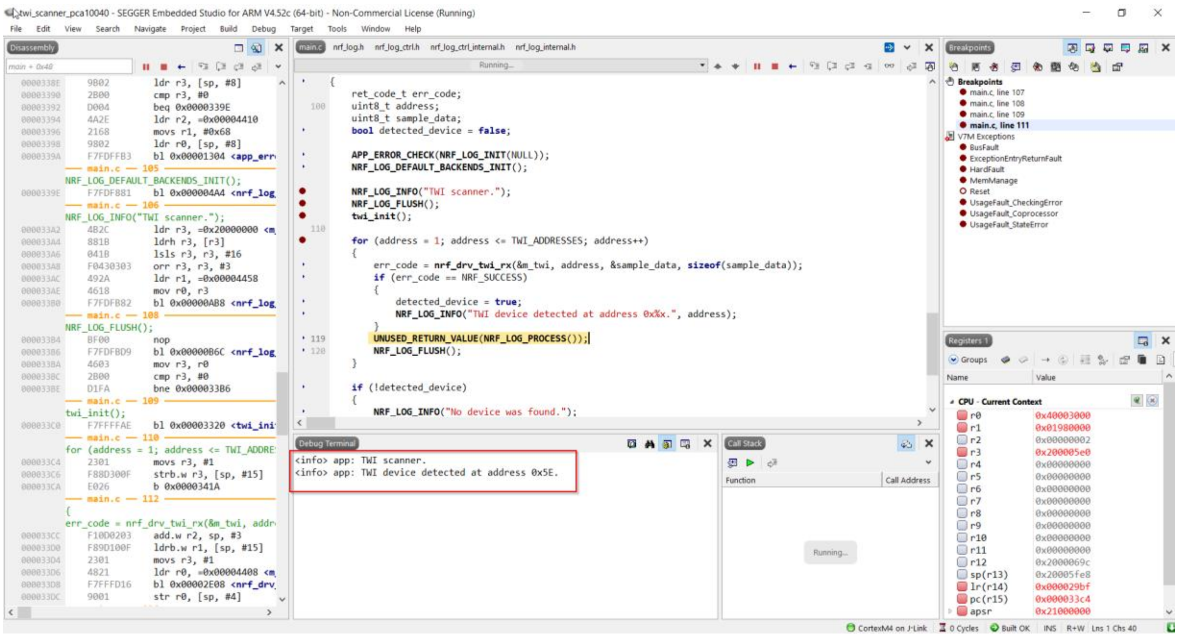


Figure 46: Debug screenshot of I2C scanner app result

7. Discussion

Objectives

Main Objective

> *The main objective of this project was to develop a low-cost functional prototype for a smart armature revolute joint to be used by MechAdept.*

Due to the time constraints, the functionality of the prototype was not obtained. However, the developed prototype is fully capable of the required functionality. Therefore, this project has given MechAdept a platform to develop the embedded software for beyond this project's timeline. As mentioned previously in the report, the required level of knowledge to use the nRF52 platform effectively was not taught in the program. Had there been more time however, this would have been learned.

Sub-Objective a)

> *Develop a modular mechanical design that considers manufacturability, ease of use and assembly, and available resources.*

This sub-objective was successfully met. A mechanical design was developed which included three main parts; Link 1, Link 2, and Link 1's cap. The links were modular as they could easily be swapped out and a new one could be 3D printed and implemented, as was done for the operating torque testing. The links could easily be assembled and disassembled with minimal tooling by pressing them together and prying them apart. 3D printing was fully utilised for rapidly iterating on parts as could be seen in the project methodologies section under the V1.0 prototype design.

Sub-Objective b)

Develop an electronics system design making use of the nRF51 or nRF52 series Bluetooth System on Chip (SoC) and implement it on a PCB.

An electronic system PCB was designed, featuring an nRF52810 SoC from the nRF52 series chipset. Six PCBs were manufactured and most of serial number 6's functionality was tested. The BLE Blinky program demonstrated that the high frequency crystal, low frequency crystal, antenna, and all the power circuitry worked sufficiently.

Sub-Objective c)

Design and implement a working program to interface the electrical system elements and provide joint position data to a user terminal.

Working programs were implemented on the joint, however the intended functionality was not obtained. The SoC was able to interface with the position sensor by reading its I2C address as seen in the results section and the RGB was successfully controlled on serial number 5. The RGB LEDs on most of the boards were oriented incorrectly, but they were able to be controlled with certain configurations of their output pins, which was demonstrated in the BLE Blinky program.

Final Prototype Analysis

Requirements Outcome

Table 3: Requirement outcomes

General	Conditions	Requirement met? Y/N	Comments
Low Cost	<$200	Y	The total expense = $316.52 however a single unit costed $71.41
Simple		Y	Uses only 3 manufacture mechanical parts, a PCB, battery, and magnet.
Fast to assemble/disassemble		Y	Can be pressed together and pried apart in under 20 seconds
Modular design		Y	Links were designed to be swappable parts
Mechanical Structure			
2 link joint		Y	

Full angular range of motion	± 360°	Y	
Dimensions	<= Ø60mm X 100mm	Y	Diameter is 40.2mm and height is 37.2mm
Angular position holding torque	Approximately 1.8kg.cm	Y	The V1.3 iteration of link 2 proved to hold 0.358Nm (3.65kg.cm)
Electronics			
Use nRF52 Chipset		Y	nRF52810 Used
Battery Powered		Y	Li-Po battery used
System on PCB		Y	PCB's designed, constructed and tested
Simple PCB	<= 4 layers	Y	Uses two layers with components on one side
Accessible debugger point		Y	Can be accessed by pulling apart links
Software			
Bluetooth capable		Y	Uses nRF52810
Moderate Precision	<= ± 1°	N	Functionality was not completed

One of the targets of the armature joint is to be as miniature as possible. This prototype has succeeded in this; however, it could be made smaller. The PCB could accommodate components on both sides of the board, allowing for further component density improvements. The layout of the PCB also had many gaps where components could have been placed. This was due to the circular shape of the board, having the hall sensor placed in the direct centre, strictly following Nordic Semiconductor's recommendations on layout, and having a trace antenna. The hall sensor could have been offset and accounted for in software using a lookup-table-type method of position measurement rather calculating it. The joint could also be made smaller by using a different technology than 3D printing. Moving beyond the constraints of 3D printing would help with miniaturisation but wouldn't be as simple to manufacture and replace.

Timing

The various aspects of this project took longer than expected. An updated Gantt chart can be found in the image below. An extension two weeks was granted from the project coordinator for the final report. The boxes in the dark red were approximately accurate depictions of the actual time spent.

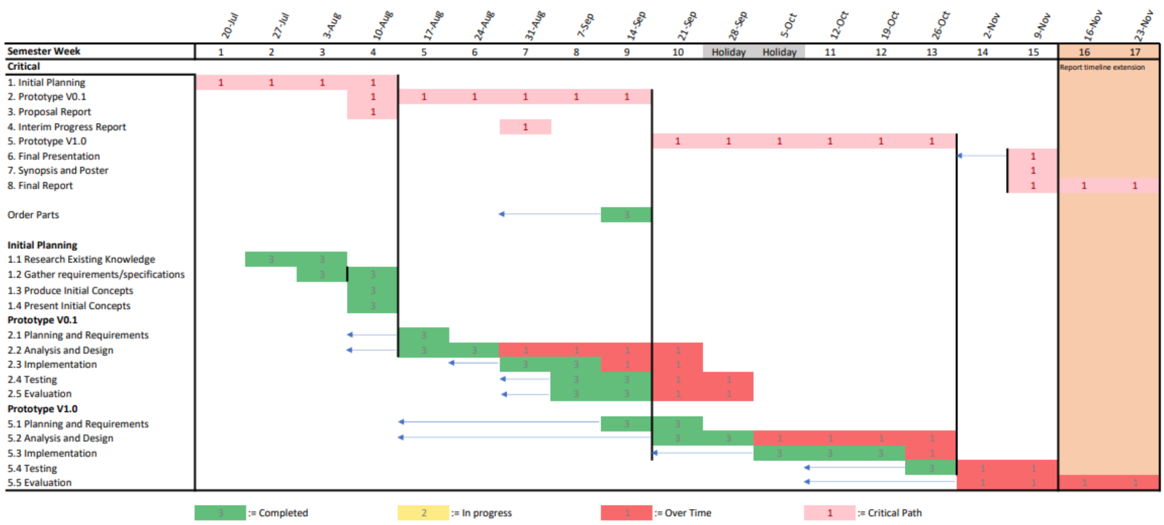


Figure 47: Updated Gantt Chart

Across the full project timeline, time was managed with the help of an online tool called 'Clockify.' This proved to be a useful was to track time and aided with improving time efficiency. The full project Clockify report can be found in Appendix F – Clockify Project Timeline Report. Over 415 hours were spent on the project which was over 138% of the expected 300 hours. This extra time was spent attempting to complete milestones and meet deadlines, however there was limited success with this.

Weekly Journal

A weekly journal from week one through to week 14 was submitted to the project coordinator each week to keep track of learnings and progress. It was also commonly used for the week's agenda. The 15th and following two weeks were not kept or submitted due to prioritising the resources on other aspects of the project and critical tasks from other courses. The full weekly diaries can be found in Appendix G – Weekly Diaries.

8. Conclusion

In conclusion, the main objective was not met fully. The level knowledge needed for the C programming knowledge, communication protocols, and Nordic's SoftDevice was not obtained in time to complete the software functionality. This provided an opportunity to find gaps in knowledge which can be learned in the future. However, the prototype met the mechanical and electronics objectives. The mechanical structure was designed through rapid iteration of concepts using 3D printing technology and the electronics system was designed using CAD with reference to Nordic Semiconductor's example layout for the RF functionality and the components' datasheets. The electronics were tested using direct measurements and using direct demonstration from example programs from the Nordic Semiconductor SDK. Although this prototype was not fully functional, it was fully capable of the required functionality. The project took more time than expected, and there is still more work to do in the future.

9. Recommendations for future work

The most important recommendation for future work is to complete the software functionality. Other than this, more documentation needs to be produced for building upon this project since this project was for MechAdept more functionality is needed for the vision of the armature. A guide was written for modifying SDK examples for use of the armature joint which can be found in Appendix H. The power consumption could be measure and hence battery-life could be calculated in the future when the system is running with full intended functionality.

10. References

Bailey M. (2011, September 14). *PCBs layout guidelines for RF & mixed-signal*. Maxim Integrated - Analog, Linear, and Mixed-Signal Devices. https://www.maximintegrated.com/en/design/technical-documents/tutorials/5/5100.html#:~:text=sets%20as%20applicable.-,RF%20Transmission%20Lines,buried%20in%20an%20internal%20layer

Circuitbasics. (n.d.). *Introduction-to-I2C-Single-Master-Single-Slave*. https://www.circuitbasics.com/wp-content/uploads/2016/01/Introduction-to-I2C-Single-Master-Single-Slave.png

Cueller, J. S., Smit, G., Zadpoor, A. A., & Breedveld, P. (2018, August 16). *Ten guidelines for the design of non-assembly mechanisms: The case of 3D-printed prosthetic hands - Juan Sebastian Cuellar, Gerwin Smit, Amir a Zadpoor, Paul Breedveld, 2018*. SAGE Journals. https://journals.sagepub.com/doi/full/10.1177/0954411918794734

Dastjerdi, A. V. (2016). Chapter 15 - Applied Internet of Things. In *Internet of things: Principles and paradigms*.

Hein, A. (2016, October 18). *3D printing joints*. GiHubIO.

https://coloringchaos.github.io/form-fall-16/joints

Infineon. (2019, April 9). *TLV493D-A1B6 3D Magnetic Sensor v1.1*.

https://www.infineon.com/cms/en/product/sensor/magnetic-sensors/magnetic-position-sensors/3d-magnetics/tlv493d-a1b6/

K&J Magnetics. (n.d.). *All about magnetization direction*.

https://www.kjmagnetics.com/blog.asp?p=magnetization-direction

Laura. (2014, December 4). *Bluetooth low energy basics*. Wayback Machine.

https://web.archive.org/web/20160930015609/projects.mbientlab.com:80/bluetooth-low-energy-basics/

Maxim Integrated. (2001, January 31). *Regulator Topologies for battery-powered*.

https://www.maximintegrated.com/en/design/technical-documents/tutorials/6/660.html

Micronas. (2017, October 27). *HAL® 371x, HAL 372x,HAL 373x*.

https://www.micronas.tdk.com/en/system/files/downloads/files/HAL371x_HAL372x_HAL%20373x_Programmable_2D_Position_Sensors_with_Arbitrary_Output_Function_1DSH_final.pdf?ref_disty=digikey

MuRata. (2017, November 11). *Rotary Position Sensors*. Retrieved August 14, 2020, from

https://www.murata.com/en-global/products/productdetail?partno=SV01A103AEA01R00

Nordic Semiconductor. (2019, January 24). *nRF52840 Dongle - PCA10059 v1.0.0*.

https://www.nordicsemi.com/Software-and-tools/Development-Kits/nRF52840-Dongle

Nordic Semiconductor. (n.d.). *NRF52 DK - Downloads*. https://www.nordicsemi.com/Software-and-tools/Development-Kits/nRF52-DK/Download#infotabs

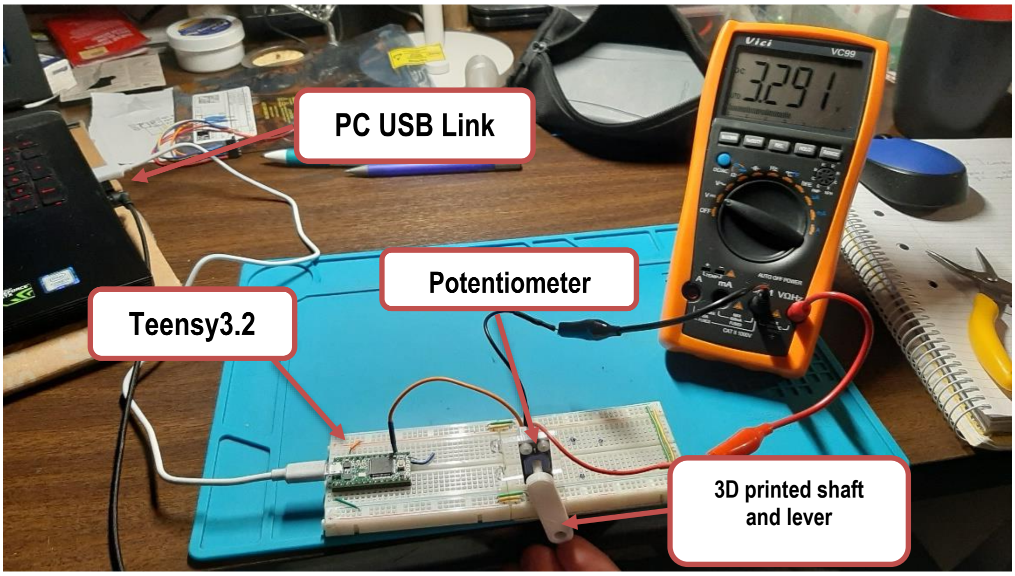
PCBZone. (n.d.). *FAQs and Guides*. https://www.pcbzone.net/user/faq.aspx

Stilwell, F. (2020, July 8). *1_Logic_Analyser-Minimal_Oscilloscope-Signal_Generator*. GitHub. Retrieved August 14, 2020, from https://github.com/FStilwell/1_Logic_Analyser-Minimal_Oscilloscope-Signal_Generator

Troughton, M. J. (2008). *Handbook of plastics joining: A practical guide* (2nd ed.). William Andrew.

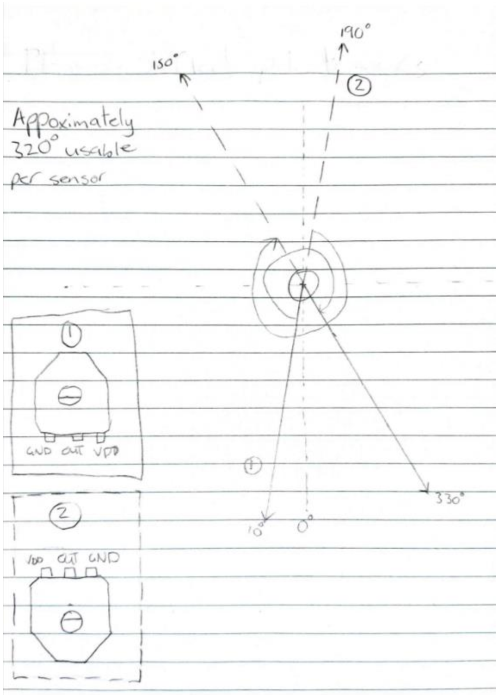
Appendix A – Potentiometer Test

A simple test was performed to determine the usable angle range of the SV03A103AEA01 potentiometer. See the image below for the test setup.

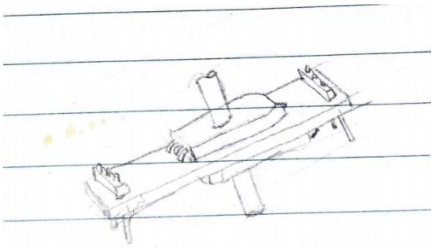


The test involved implementing the potentiometer on a breadboard with a Teensy3.2 microcontroller. Since the potentiometer outputs a 'linear' voltage from 0V to the supply voltage depending on the position of the potentiometer's wiper, the voltage was read by an analogue-to-digital-converter-capable pin and this result was sent to a PC via USB UART with a 115200 baud rate. A 3D printed shaft and lever was steadily rotated for a complete revolution of the arm. A free programme called Serial Plot was used to plot the readings. From the A/D readings, the voltage could be calculated and therefore an approximate angle reading. The resulting plot can be found in figure in the results section. It was determined that the potentiometer had a dead range of about 36 degrees. A false assumption was made that the potentiometer output would reset to zero volts directly after approaching the limit of 333.3°. This was a showstopper for using a single one of these sensors, but there was the possibility of using two that are phase shifted.

Since one potentiometer is insufficient with its useful angular measurement range, piggy-backing another one on top of it with a 180° phase shift would complete the full range of motion. The image below shows the ranges of motion of two of the SV03A103AEA01 potentiometers in said configuration.



The potentiometers could be interfaced on a single PCB with one on each side as shown in the sketch below. This differs slightly from the shown ranges of motion as they are showing the potentiometers in the same orientation (with a 180° shift about the wiper shaft axis) whereas the sketch below has potentiometers phase shifted 180° both about the axis of the wiper shaft and 180° about the vertical axis. This would result in sensing the opposite direction and could be accounted for in software. Male pin headers are shown positioned at either end of the PCB to carry the signals down to a host main PCB.



Considerations:

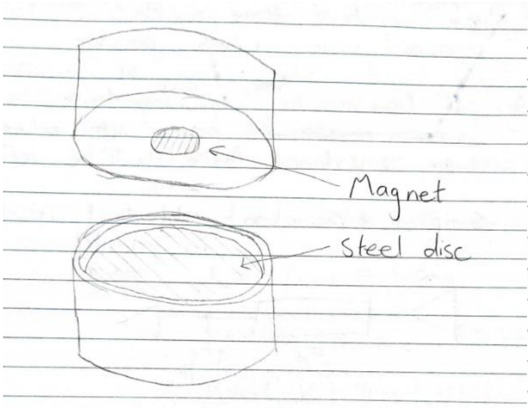
- Simplicity – For electronics interfacing, the potentiometers a relatively simple to interface to a micro controller with just one signal wire each with a power supply and ground. The signal can be easily interpreted with an analogue-to-digital converter onboard the microcontroller and some simple math to scale the A to D reading to the angle range, however having two sensors complicates it mildly. For mechanical interfacing, the potentiometers require many more parts such as bearings to protect their integrated bearings, mating the potentiometers to the main PCB, and mating the shaft to the opposite joint link in such a way that it can be easily assembled/disassembled.

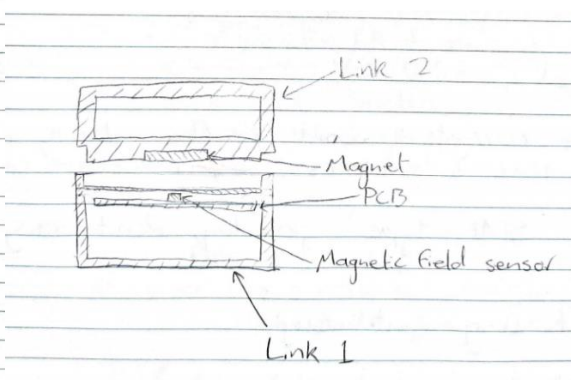
- Precision – The sensor's datasheet specifies a linearity of ±2% of its full range (333.3°) which comes to ≈ ±6.7°.
- Construction – The PCB prototype could be engraved using WelTec's LPKF CNC engraver and future boards can be ordered cheap from China. The shaft and future enclosure assembly could be 3D printed, though the shaft could be machined out of metal.

Appendix B – Magnetic Mechanical Interface Concept

Axial mechanical connection idea:

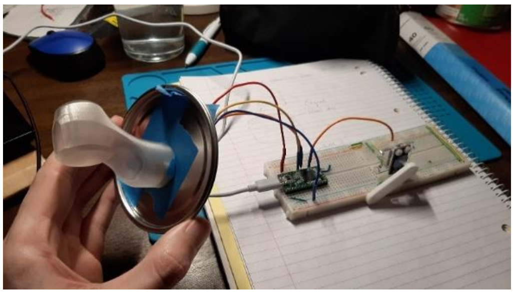
Use a strong magnet on one link and a metal disc on the other.

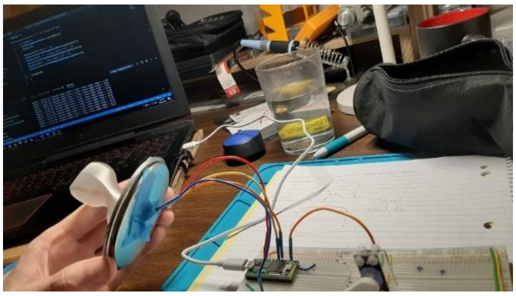




Testing the Sensor Concept

Using a Teensy3.2 microcontroller it was confirmed by observation that the magnetic field can be sensed almost as if there were no obstructions using the TVL493D 3D magnetic sensor. The test configuration is shown in the images below and note that the potentiometer on the breadboard was not part of the test. Two steel/tin lids were used as a mock-up steel plate and the magnetic sensor was taped to the opposite side from which the magnets were interacting with. A small 3D printed joint mock-up given by the supervisor held the magnets. A stack of 4 and 5 Ø15mm X 1mm N45 Neodymium magnets were used in separate tests. A limitation of this test was that there was nothing to hold the joint in place axially, so the position of the magnets was not consistently over top of the sensor.





An issue came up where the data readings were lost after a while but was resolved with a restart of the microcontroller.

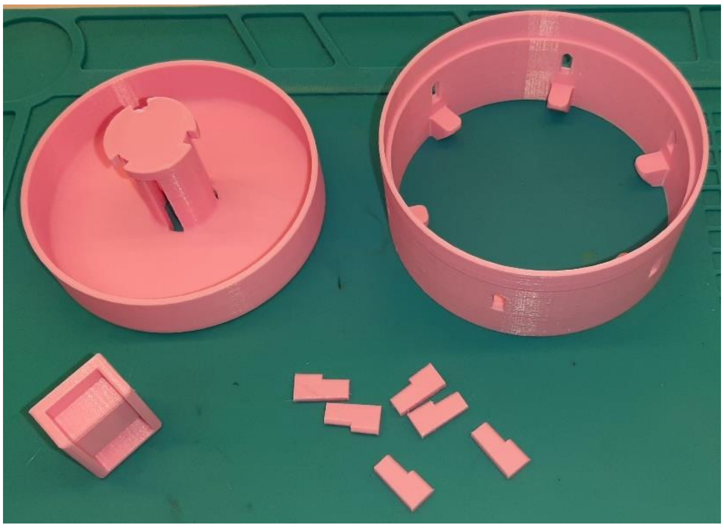
- Feel how much force would be required manipulate the position of the link against a steel plate.

Using the same steel/tin lids, 3D printed mock-up joint and magnets as above, it was found that the stack of 6 magnets against the two lids together provided the greatest normal force. Masking tape was applied to the contact surfaces for some extra friction. This concept seems promising with the amount of torque that can be required to rotate the disc just with the magnet on the plate surface. This test also had a limitation of having nothing to hold the joint in place axially.

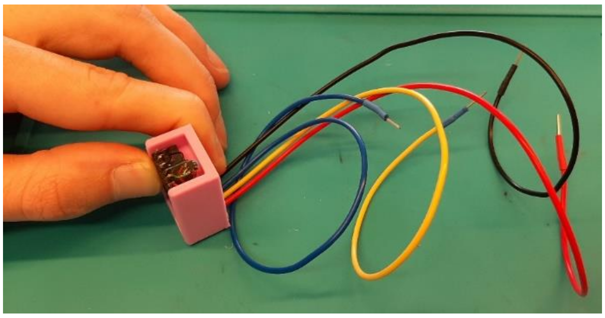
Further work:

Based off these simple tests, the concept seems to be worth taking further. 3D printed mock-up links could be used to provide a fixed axial coupling, a means to test the torque required to actuate the joint using a lever arm, and a platform for testing more friction-increasing methods. A small seat could be 3D printed to hold the magnetic sensor in place more rigidly.

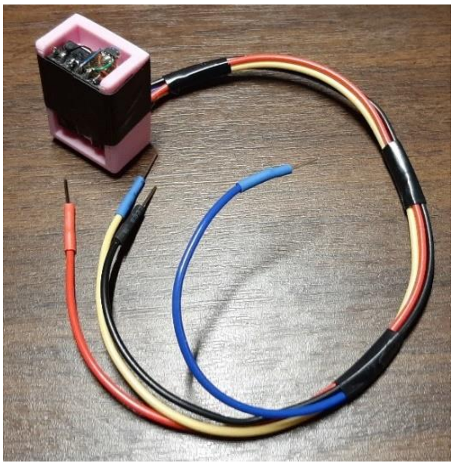
Appendix C – Magnetic Interface Concept Implementation and Testing Process



The parts were 3D printed successfully after a failed attempt on the 31st of August due to an axis skipping stepper motor steps. The 3D printer had been fixed and the parts turned out well. The cap on link 2 had warped, making it difficult to assemble when the magnets were in place. The chute on link 2 which holds the cap was flimsy as expected.



Longer leads for the TLV493D magnetic sensor were constructed using multicore wire with pin headers soldered and heat shrunk to each end. The longer leads were needed for convenient implementation with a breadboard for testing. The magnetic sensor was in a different position than expected due to overlooking a measurement in the design of the test sensor holder. However, the main sensor was slightly above flush with the top of the holder which was manageable.



The image above shows the test sensor fully assembled with its holder and leads. These were taped together with electrical tape.

Only one lid was able to fit into link 1 due to another overlooked measurement. Therefore, this mock-up will be tested with just 1 lid as the link was not able to be modified to accommodate the second lid, and it was not worth printing another part as this would be a waste of PLA material and time. Link 1 was designed with the intention of fitting the other lid in, so the height was not as expected. This was overcome by placing 3 of the 6 wedges on the platforms underneath the lid, allowing the wedges to work properly on the link's wall. The image above shows a view of this from underneath the link. The wedges had to be used upside-down to what was expected, however they functioned in the same way. Since this was a test mock-up, it just needed to work well enough to test the sensor and mechanical interface concept.

The cap was warped while 3D printing due to how thin the bottom layer was. It was 1mm thick to have a minimal gap between the magnet and the steel plate (lid). This can be seen in the image above.

An issue resulting from this was not being able to fit the cap on the chute with the magnet in place, so a rotary tool with a sander attachment was used to wear down the inner diameter of the cap's walls until it fit on the chute. This allowed a press-fit to be used which helped clamp onto the magnets, alleviating the need use the adhesive that came on the magnet. The image above shows the cap and magnets assembled onto link 2's chute.

The other issue caused by the warpage was having an uneven surface which caused a low friction coefficient with the steel plate. The surface was sanded with 120 grit sandpaper to even help even the surface.

Masking tape was added to the outside rim of the link 2 to provide more friction between the links. However, this would be removed before testing the resistive torque.

Testing the Concept

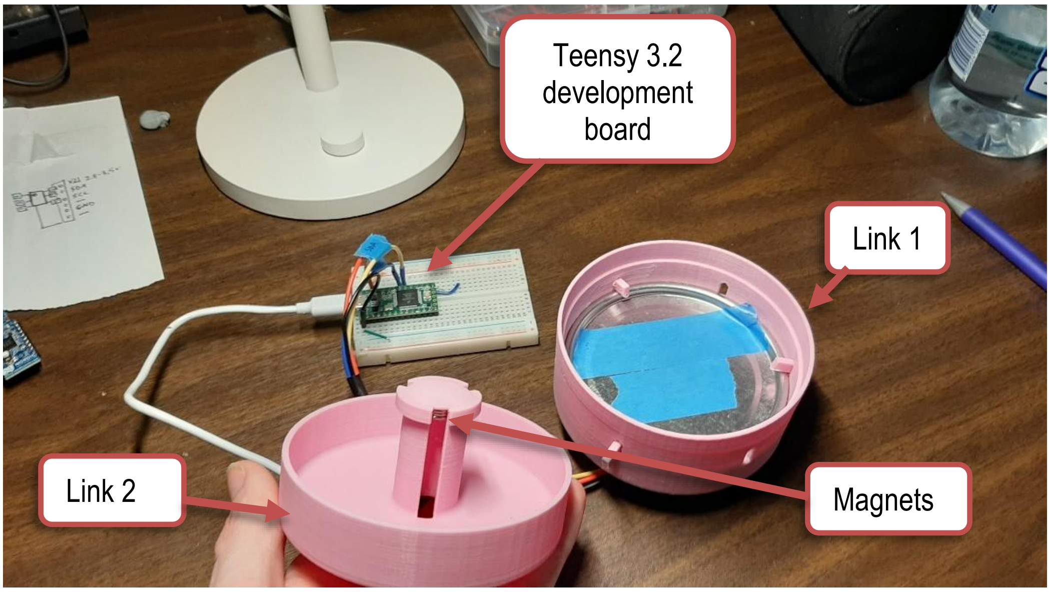
The mock-up 3D print of the magnet on steel mechanical interface was tested using a teensy3.2 development board and the test 3D magnetic sensor.

Test 1:

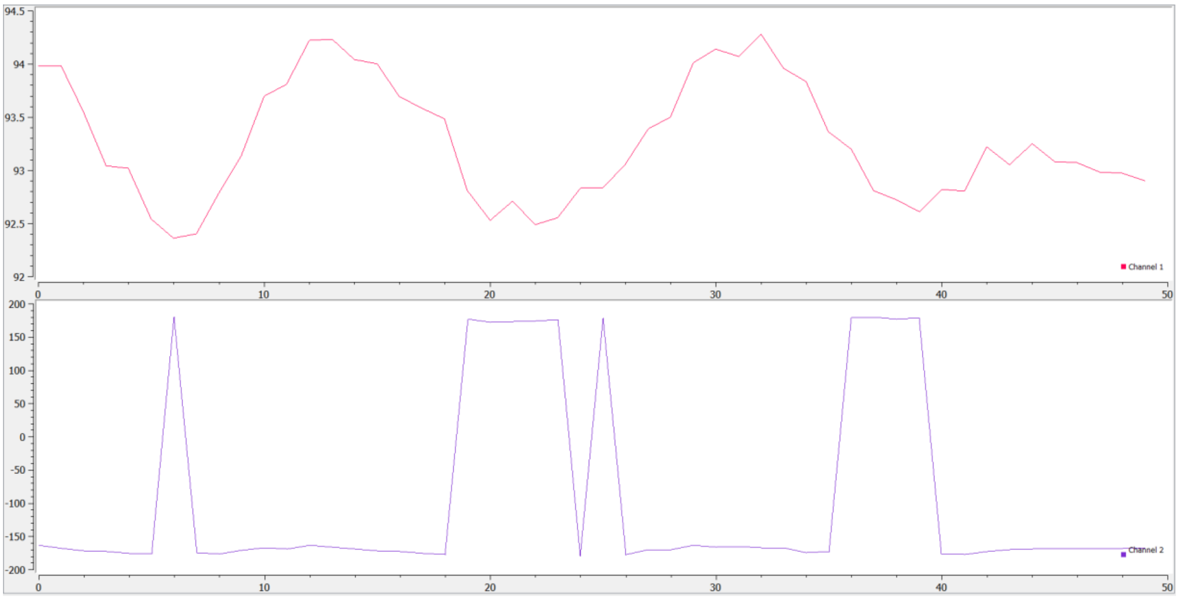
The aim of this test was to validate the operation of sensing the magnetic field through the steel disc with the magnet rotating about the fixed axis of the joint. The example code provided by Infineon for the magnetic sensor was modified to produce a usable serial output for testing. The output from the sensor was converted to radians and formatted in a way that the Serial Plot application would understand the two channels. The main code for which can be found below. Channel 1 outputted the magnitude of the sensed flux density from all axes in milli-teslas (mT) and channel 2 outputted the azimuth angle of the magnetic south pole in degrees.

```
#include <Arduino.h>
#include <Tlv493d.h>

// Tlv493d Object
Tlv493d Tlv493dMagnetic3DSensor = Tlv493d();

void setup() {
  Serial.begin(9600);
  while(!Serial);
  Tlv493dMagnetic3DSensor.begin();
}

void loop() {
  Tlv493dMagnetic3DSensor.updateData();
  delay(100);

  //Ch1 - Sensed flux density (mT), sqrt(x^2 + y^2 + z^2)
  Serial.print(Tlv493dMagnetic3DSensor.getAmount());
  Serial.print(",");

  //Ch2 - Azimuth angle (degrees)
  float degrees = Tlv493dMagnetic3DSensor.getAzimuth() * 180/3.14; //Azimuth a
ngle in degrees
  Serial.println(degrees);
  Serial.flush();
```

The test configuration can be found in the image below. The magnetic sensor was placed in the centre of the steel disc on the underside while housed in its holder. A stack of 4 N45, 15mm diameter and 1mm thickness magnets were placed in link 1.

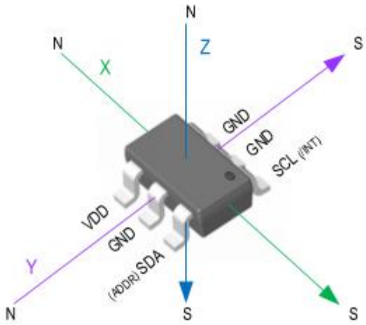


The links were put together and the Serial Plot application was setup. The waveform was captured for approximately two revolutions of link 2. The results of which can be found below.



The results were not what was expected as the concept seemed to work from the test without the 3D printed mock-up. The output was mostly consistent around -175 degrees until dropped down slightly past -180 and jumped up to positive 180 degrees. This jump was as expected due to the operation of the magnetic sensor. However, the conclusion of this test was that the operation of the sensor with the magnetic sensor was not understood fully. It was assumed that the example program measured the azimuth angle about the south pole of the magnet.

The example code for the TLV493D sensor calculated the azimuth angle by taking the inverse-tan or arctan of the Y reading over the X reading. After looking through the datasheet, it was found that each axis measures the flux density with the positive reading meaning it was a south pole. The directions can be found below, taken from the sensor's datasheet provided by Infineon.



The test configuration featured the magnetic south pole of the magnet facing the direction of the z-axis. This meant that there were no measurements taken in the X and Y directions as the magnetic field lines were parallel to the sensor.

To get a proper reading of the angle, the magnet could be placed on its side so that the south pole of the magnet points parallel to the X/Y plane and hence the magnetic field lines would cut the X/Y sensor plates at right angles.

Another solution could be to acquire new magnets which are diametrically magnetised as shown in the image below.

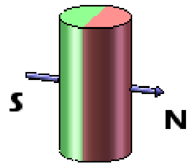
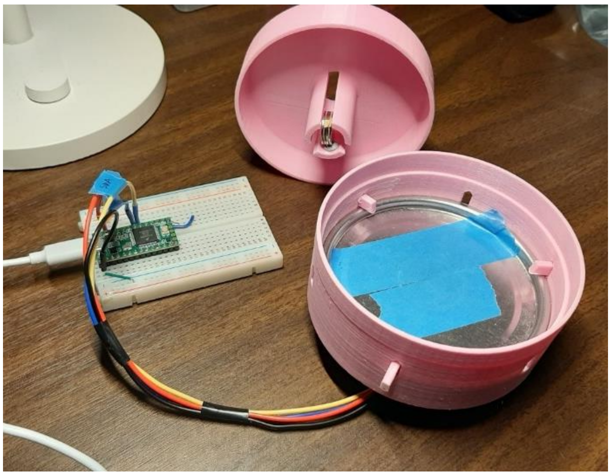
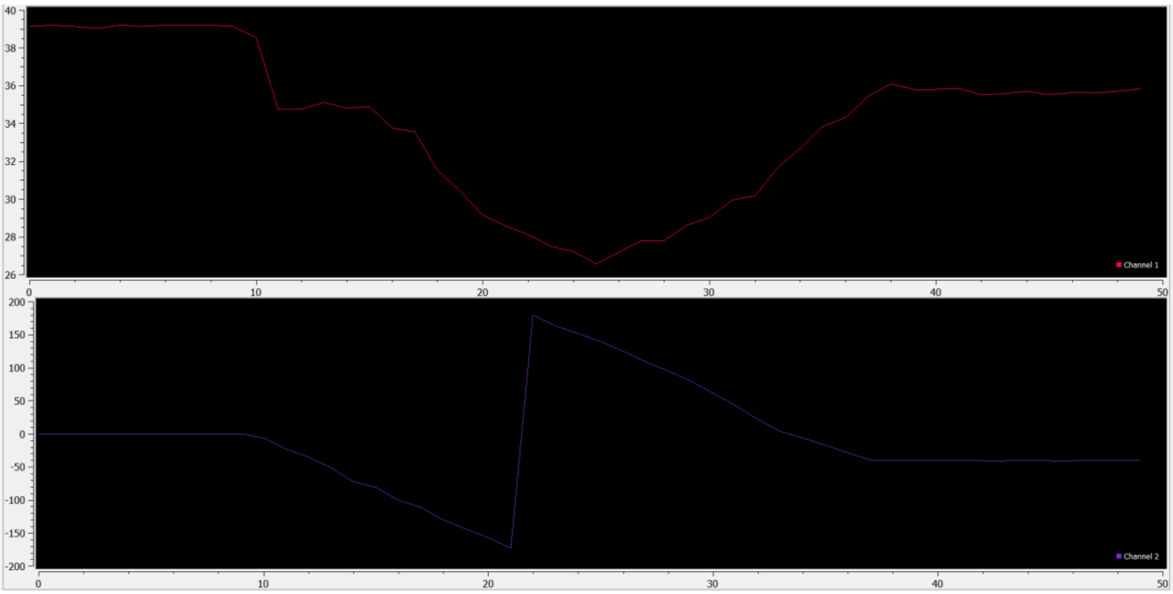


Figure 48:Diametrically magnetised cylindrical magnet (K&J Magnetics, n.d.)

Test 2:
A second test was implemented with the same configuration as the previous, but with the magnets standing up on their side in link 2's magnet housing. Blue-tack was used to keep the magnets poking through the groove cut-out in the chute to keep the magnets rotating with the link. See the image below. The cap was placed back over the chute after this image was taken to keep the distance between the magnets and the sensor consistent.



Link 2 was rotated approximately one revolution to test the full angle range. See the image below for the results. Unfortunately, only the plot with a dark background was captured in this test. From both tests, the total amount of flux density on channel 1 changes with the rotation of the joint. This could be from a misalignment of the magnet to the sensor. The range of variation in channel 1 of test 2 (approx. 10mT) was much greater than that of test 1 (approx. 4mT). There was an obvious misalignment of the magnet as it was standing on its side and offset in test 2.

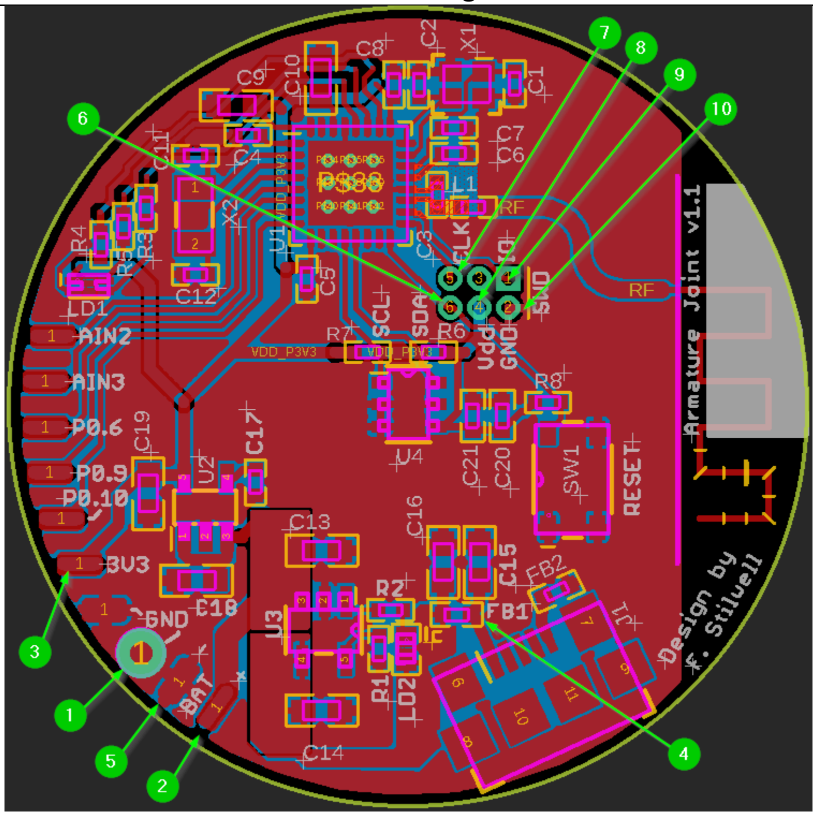


The result of test 2 was much more as expected. Although the actual angle was not measured to compare, the plot of the angle in channel 2 was very stable and responsive. It can also be seen that the was little to no variation in the signal when there was no movement, proving that the sensor gives a stable reading.

Appendix D – Armature Joint PCB Testing as at 11/11/2020

PCB Name	Armature Joint V1.1
Serial Number	3
Tester	Flynn Stilwell

Test Point Diagram



Top View

Test Points	Test Parameters / Conditions	Criterion, Units	Value	Comments
Multimeter Probes	No power connected	Resistance, Ω	0.1	
1-2			OL	
1-3			9.78k	
1-4			OL	
1-10			0.0	
1-6			0.0	Check button, all pins connected to GND. Button pin 4 connected to GND
1-7			6.12M	
1-8			OL	
1-9			6.15M	
1-2	USB Power Only	Voltage, V		
1-3				
1-4				
1-2	Battery Power Only	Voltage, V		
1-3				
1-4				

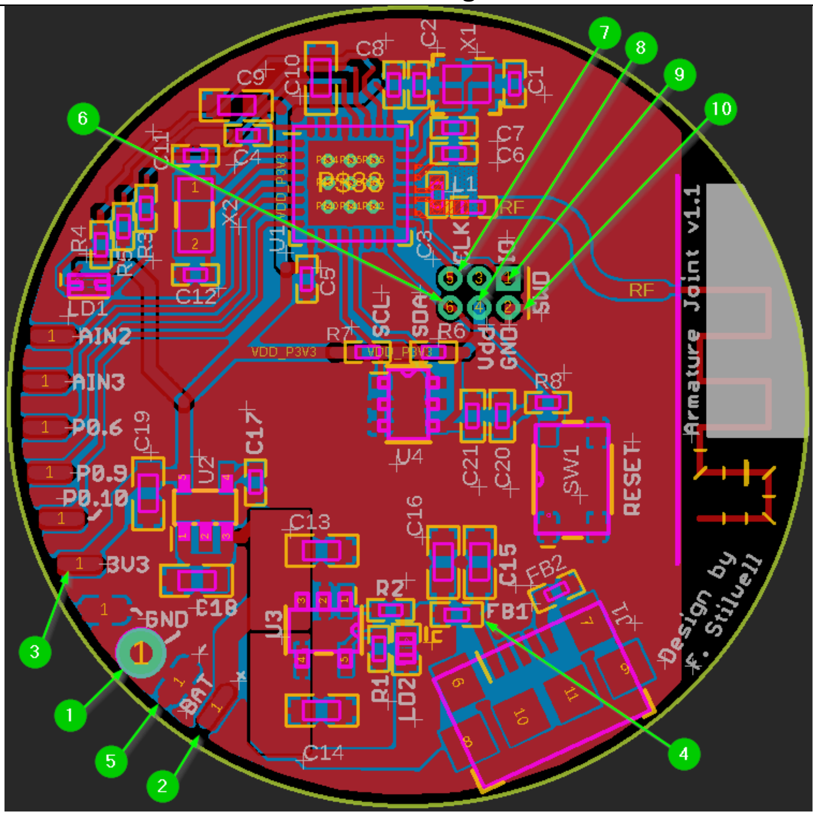
Notes:

11.11.20

- RGB LED orientation is wrong
- Button is not functioning due to schematic error, should be removed

PCB Name	Armature Joint V1.1
Serial Number	4
Tester	Flynn Stilwell

Test Point Diagram



Top View

Test Points	**Test Parameters / Conditions**	**Criterion, Units**	**Value**	**Comments**
Multimeter Probes	No power connected	Resistance, Ω	0.3	
1-2			OL	
1-3			9.82k	Value from pullup resistor R8, related to GND'd switch
1-4			OL	
1-10			0.1	
1-6			0.1	Check button, all pins connected to GND. Button pin 4 connected to GND
1-7			47.2k	
1-8			OL	
1-9			6.52M	
1-2	USB Power Only	Voltage, V	3.987	
1-3			3.343	
1-4			5.074	
USB Tester		Current, A		
1-2	Battery Power Only	Voltage, V		
1-3				
USB Tester		Power, W		

	Battery + USB, Charging battery	Current, A		
		Voltage, V		

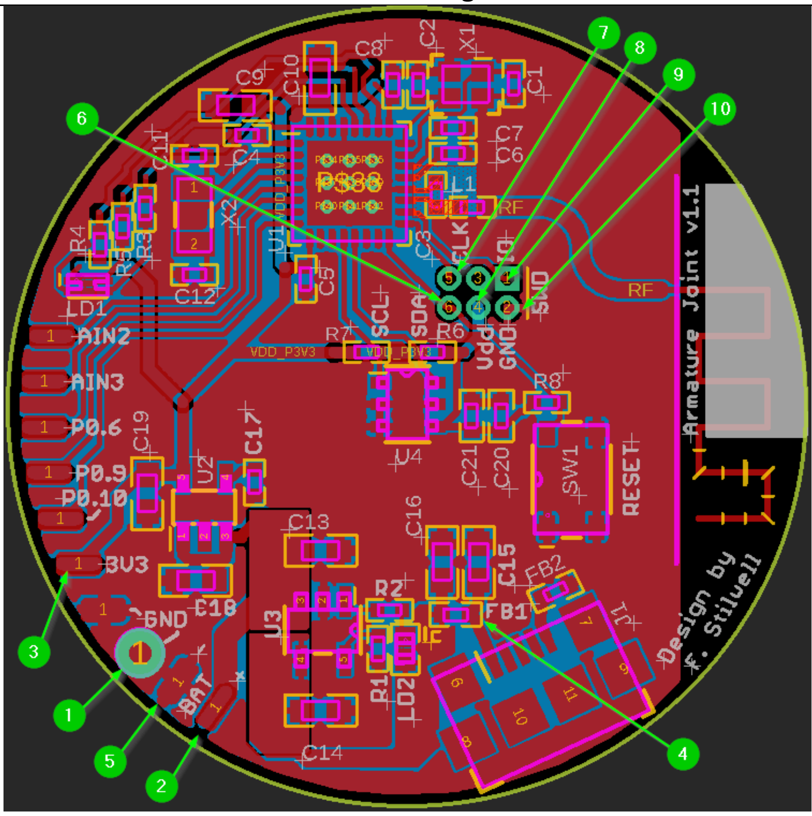
Notes:

11.11.20

- RGB LED orientation is wrong
- Button is not functioning due to schematic error, should be removed

PCB Name	Armature Joint V1.1
Serial Number	5
Tester	Flynn Stilwell

Test Point Diagram



Top View

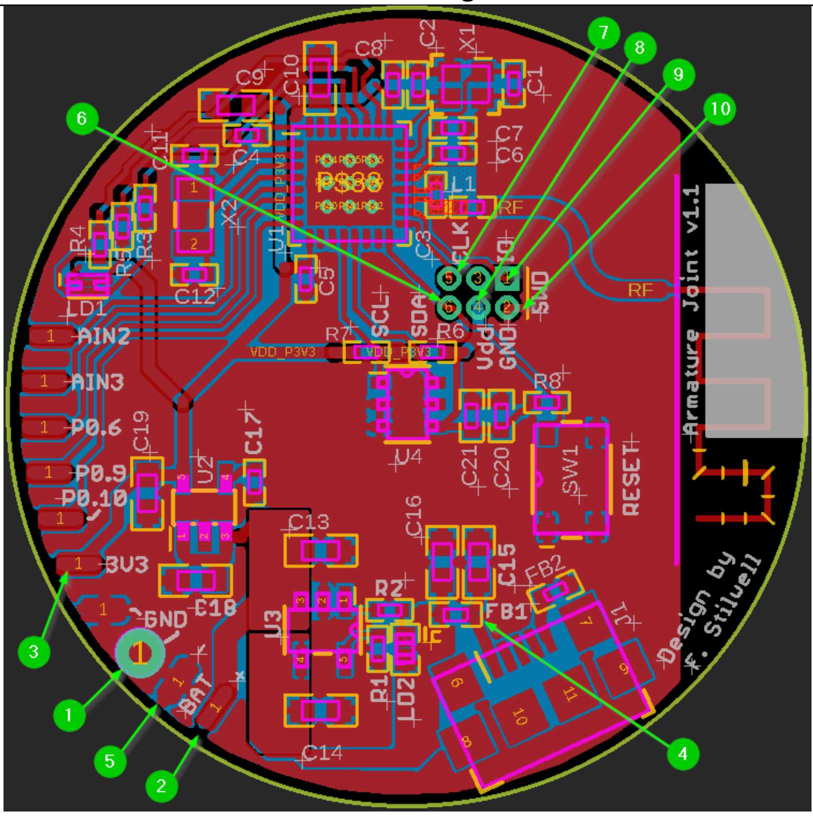
Test Points	**Test Parameters / Conditions**	**Criterion, Units**	**Value**	**Comments**
Multimeter Probes	No power connected	Resistance, Ω		
1-2			OL	
1-3			9.84k	Value from pullup resistor R8, related to GND'd switch
1-4			OL	
1-10			0.2	
1-6			0.2	Check button, all pins connected to GND. Button pin 4 connected to GND
1-7			47.48k	
1-8			9.84k	
1-9			152.0k	
1-2	USB Power Only	Voltage, V		
1-3				
1-4				
1-2	Battery Power Only	Voltage, V		
1-3				
1-4				

Notes:
11.11.20

- LD2, charger indicator orientation is wrong
- BLE Blinky program upload: Indicates advertisement but cannot be found with nRF Connect Android
- C1 possibly disconnected
- Button is not functioning due to schematic error, should be removed

PCB Name	Armature Joint V1.1
Serial Number	6
Tester	Flynn Stilwell

Test Point Diagram



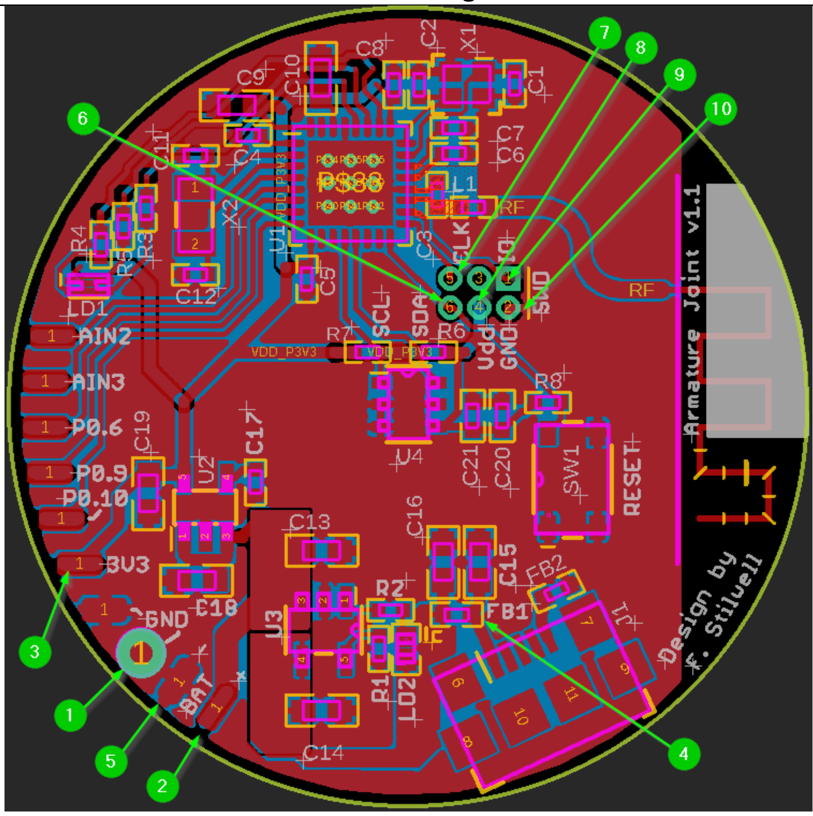
Top View

Test Points	Test Parameters / Conditions	Criterion, Units	Value	Comments
Multimeter Probes	No power connected	Resistance, Ω	0.2	
1-2			OL	
1-3			9.80k	Value from pullup resistor R8, related to GND'd switch
1-4			OL	
1-10			0.1	
1-6			0.2	Check button, all pins connected to GND. Button pin 4 connected to GND
1-7			47.92k	
1-8			9.80k	
1-9			156.0k	Lower than S/N 3 and 4
1-2	USB Power Only	Voltage, V	4.060	
1-3			3.328	
1-4			5.072	
USB Tester		Current, A	0.000	No current draw, no shorts
1-2	Battery Power Only	Voltage, V		
1-3				
1-4				

<table>
<tr><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td colspan="5">Notes:
- Removed Button
- R4 and R5 needed more solder on their uC pin pads</td></tr>
</table>

PCB Name	Armature Joint V1.1
Serial Number	1
Tester	Flynn Stilwell

Test Point Diagram



Top View

Test Points	Test Parameters / Conditions	Criterion, Units	Value	Comments
Multimeter Probes				
1-2			OL	
1-3			9.82k	Value from pullup resistor R8, related to GND'd switch
1-4			OL	
1-10	No power connected	Resistance, Ω	0.2	
1-6			0.2	Check button, all pins connected to GND. Button pin 4 connected to GND
1-7			45.53k	
1-8			9.82k	
1-9			148.7k	
1-2			4.062	
1-3	USB Power Only	Voltage, V	3.327	
1-4			5.048	
1-2			3.860	
1-3	Battery Power Only	Voltage, V	3.328	
1-4			0.515	

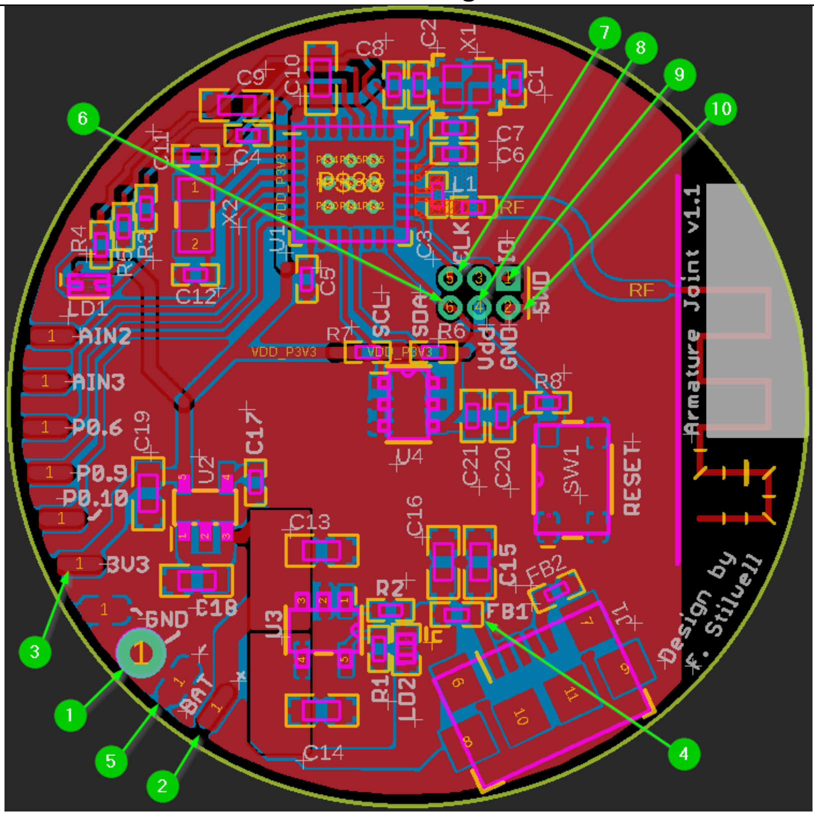
Notes:

11.11.20

- RGB LED orientation is wrong
- Button is not functioning due to schematic error, should be removed

PCB Name	Armature Joint V1.1
Serial Number	2
Tester	Flynn Stilwell

Test Point Diagram



Top View

Test Points	**Test Parameters / Conditions**	**Criterion, Units**	**Value**	**Comments**
Multimeter Probes	No power connected	Resistance, Ω		
1-2			OL	
1-3			9.80k	Value from pullup resistor R8, related to GND'd switch
1-4			OL	
1-10			0.2	
1-6			0.2	Check button, all pins connected to GND. Button pin 4 connected to GND
1-7			47.64k	
1-8			9.80k	
1-9			0.955M	Lower than S/N 3 and 4, higher than 6
1-2	USB Power Only	Voltage, V	-0.155	Issue with charger IC. Pin 1, Stat = ~0V Pin 2, GND = 0.0V Pin 3, VBAT = -120mV Pin 4, VDD = 5.109V Pin 5, Prog = ~0V

1-3			0.0	
1-4			5.109	
1-2	Battery Power Only	Voltage, V		
1-3				
1-4				

Notes:

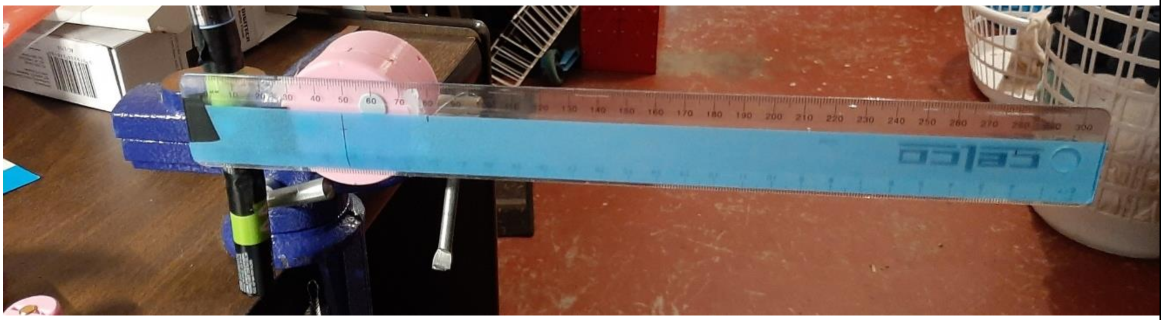
11.11.20

- LD2, charger indicator orientation is wrong
- Charger IC is not working
- Button is not functioning due to schematic error, should be removed

Appendix E – Holding Torque Tests

Test article: V0.1 Link 1 and 2 (three cantilever clip design)

Mass, m = 0.085kg



Trial Number	Distance from mounting point, d (m)
1	0.180
2	0.200
3	0.200
4	0.160
5	0.170
Average	0.182

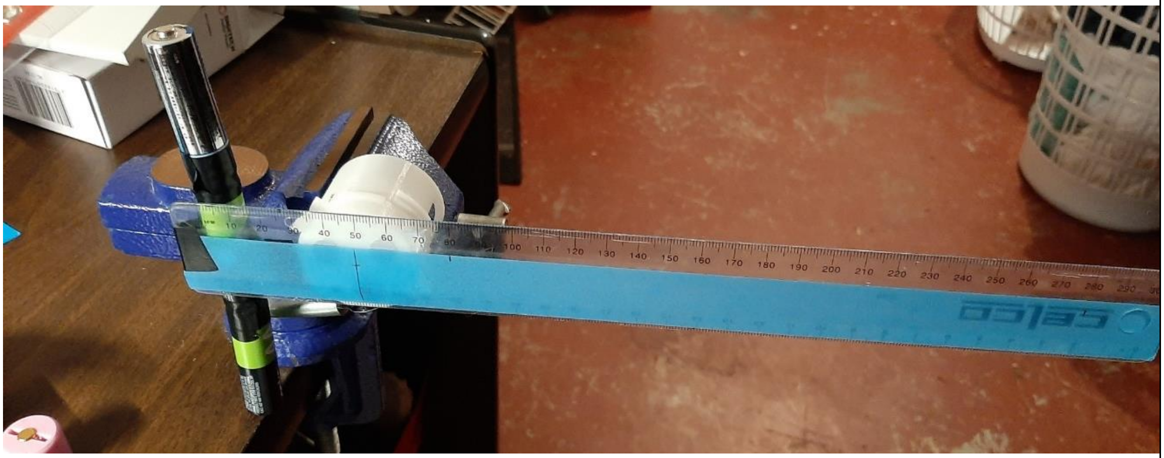
Test article: Prototype V1.0 - Link 1 V1.3 and Link 2 V1.1

Mass, m = 0.085kg

Trial Number	Distance from mounting point, d (m)
1	0.155
2	0.150
3	0.155
4	0.155
Average	0.1538

Test article: Prototype V1.0 - Link 1 V1.3 and Link 2 V1.3

Mass, m = 0.154kg



Trial Number	Distance from mounting point, d (m)
1	0.230
2	0.235
3	0.240
4	0.240
5	0.240
Average	0.237

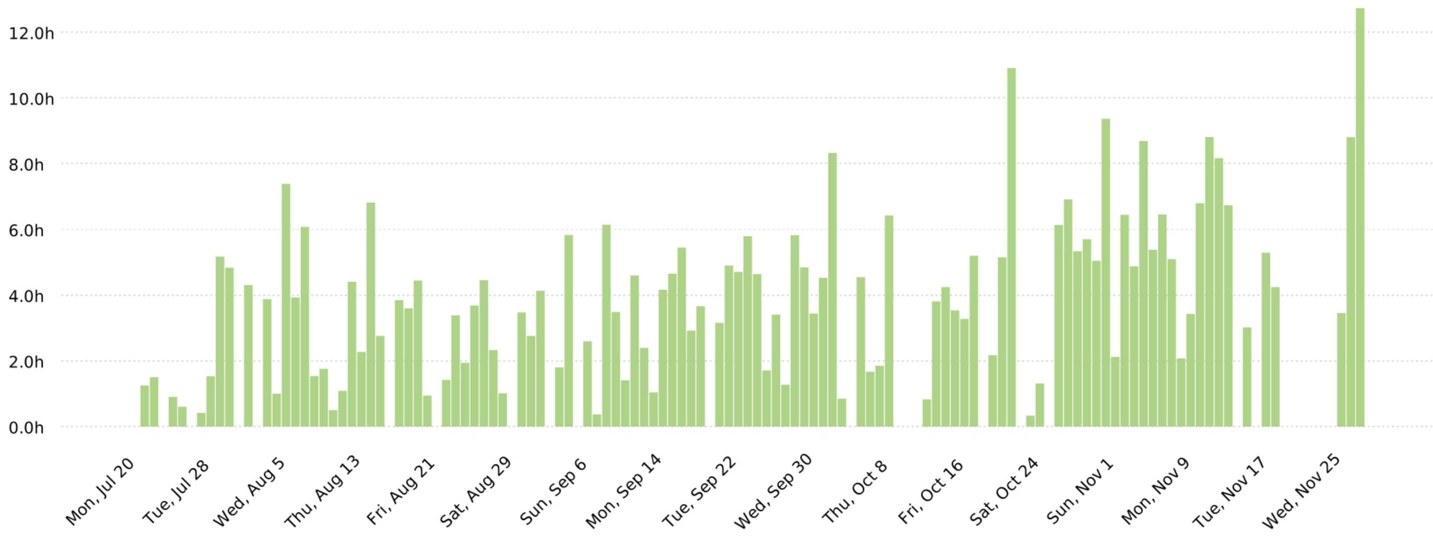
Appendix F – Clockify project timeline report

See the following pages for the Clockify project timeline report.

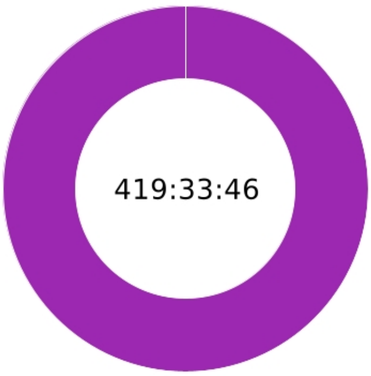
Summary report

07/20/2020 - 11/27/2020

Total: 419:33:46 Billable: 00:00:00 Amount: 0.00 USD



Project



MG7101 Engineering Development Project	419:33:46	100.00%

Description

Meeting on Zoom with Frank and Chris	00:25:00	0.10%
Met with Frank: Figured out virtual PCB model, spoke about synopsis and product to demonstrate	01:20:00	0.32%
Sent V1 features to Frank	00:28:49	0.11%
Started eagle schematic design, sourced Hall sensor components	03:23:27	0.81%

Presentation feedback, discussed next steps (mechanical, programmer, UART)	00:32:00	0.13%
Watched pcb assembly video	01:36:33	0.38%
Met course coordinator, introduction to course	01:15:00	0.30%
Weighing options on PCB manufacturing	01:54:20	0.46%
PCB manufacturing issue found, laying out components in PCB	01:36:16	0.38%
Charger IC copper pours	01:01:40	0.25%
New SoC selection	01:07:40	0.27%
Sourcing parts on digi-key from BOM	00:56:31	0.22%
Modelling PCB USB, USB connector cutout, PCB tabs and stops	02:10:51	0.52%
Introduction finished, development section started	01:34:57	0.38%
Week 14 diary entries and submission	00:54:43	0.22%
Attended 3 Presentations	01:00:00	0.24%
Discussed new SoC options, progress update on PCB	00:38:00	0.15%
Reinstaled nRF52v17 SDK for fresh start, found guide for developing for nRF52810, looked at TWI documentation	00:20:42	0.08%
Analysing structure v1.0 test print, made changes for v1.1 test print	01:58:57	0.47%
Build attempts	01:15:21	0.30%
Determined max needed power supply current	00:46:57	0.19%
Softdevice webinar note taking	01:11:33	0.28%
Power supply layout approx. done, added reset pullup resistor	02:51:47	0.68%
Attempted uploading blink example. Setting up board files	01:15:34	0.30%
Producing Mock-up drawings for write-up	00:40:18	0.16%
Started writing up the mock-up design choices	00:34:09	0.14%
Setup nRF52840 reference Design in Eagle	01:17:58	0.31%
Week 7 entry	00:04:07	0.02%
Researching debuggers and connections, Traveler's Atmel-ICE	01:12:29	0.29%
TLV493 Placement	00:54:23	0.22%

Running into Unplaced sections overflow error	02:45:54	0.66%
Tried offset mag sensor idea	01:06:43	0.26%
Calculated torque results	01:05:17	0.26%
Wrote up test results of the magnet on plate mock-up	01:29:07	0.36%
Time estimates for future work, emailed coordinator about concerns with finishing	00:39:18	0.16%
Finalised first draft and sent to James	02:10:18	0.52%
Followed tutorial on short blink program, programmed board, couldn't debug	01:26:43	0.35%
Week 8 entry and submission	00:30:03	0.12%
Uploaded BLE UART example, started modifying it	02:08:28	0.51%
Drawing mechanical concept idea, tested potentiometer with teensy	01:42:13	0.41%
Presentation Powerpoint, Gantt Chart Template Implemented	02:02:37	0.49%
Completed and sent structure v1.1 test print to Frank	02:27:08	0.58%
Ordered Diametrically magnetized cylindrical magnets	00:11:13	0.05%
Learning Seggers IDE and getting it working	03:08:43	0.75%
Function structure analysis	02:11:58	0.52%
Communication with Frank and Traveler	00:20:00	0.08%
Produced and sent Frank STL files for 2nd mockup mechanical interface	00:25:23	0.10%
Wrote up the mock-up design choices	00:59:09	0.24%
Investigating nRF52 SDK examples	00:45:00	0.18%
continued development section and photoshoot	00:29:59	0.12%
Week 15 entry	00:28:24	0.11%
Read and signed a confidentiality agreement with MechAdept	00:51:52	0.20%
Configuring VS Code intellisense	01:50:53	0.44%
PCB general layout	02:47:16	0.66%
Week 4 entry	00:09:11	0.04%

Assembled 6 Armature joint boards	08:00:00	1.91%
Voltage Regulator Selection, passives selection	00:46:18	0.18%
Scope definition	00:38:40	0.15%
Programmed nrf52 blinky example	02:36:00	0.62%
Modelling V1.0 mechanical structure link 1	01:04:06	0.26%
Watching C programming tutorial	00:19:11	0.08%
Call with Frank and Traveler: Feedback on concept ideas, questions on I2C, Debuggers,	00:35:00	0.14%
Fixed export issues and ordered boards	00:40:00	0.16%
Week 11 entries and submission	00:44:22	0.18%
Acquired files from Traveler	00:12:41	0.05%
Compared BLE central and peripheral programs	00:36:30	0.15%
setting up experiment	01:33:31	0.37%
Presentation Powerpoint	01:08:10	0.27%
Investigating features	00:44:18	0.18%
Started presentation slides	00:53:53	0.21%
Started working on program design. Stepped through BLE UART example with debugger while taking notes.	03:29:22	0.83%
Wrote up some requirements and started planning	00:38:45	0.15%
Voltage Regulator Selection	00:22:04	0.09%
Submitted week 5 diary, setup week 6 diary	00:22:22	0.09%
Requested report deadline extension	00:08:00	0.03%
Getting joints fitting together properly for showcase	01:16:35	0.31%
Preparing for PCB assembly at MechAdept	00:12:32	0.05%
Routing almost complete	01:37:03	0.39%
Updated time tracker tags	00:05:59	0.02%
Almost finished modelling link2	01:00:04	0.24%
Spoke with Traveler about deadlines, help with software. Watching webinar on softdevice	01:37:00	0.39%

Project Planning	00:46:00	0.18%
Travelled to campus, picked up 3D Print, made leads for sensor	03:25:00	0.82%
Component research - Battery holder, switches, SoC power considerations	02:22:34	0.57%
SWD connector finished, fixed pad size issue in eagle	01:13:59	0.29%
Assembled and glued mockup 3	01:34:08	0.37%
Outlined Mechanical, Electrical, and software considerations. Started on objectives	01:15:34	0.30%
Electronics block diagram, battery and voltage regulator research	01:24:28	0.34%
Design methodology	01:53:11	0.45%
3D modeled and exported mockup files for printing	03:28:21	0.83%
Downloading Jlink V6.2d and SES v3.3	00:56:23	0.22%
Potentiometer interface concept drawings and analysis	01:14:46	0.30%
Exported mock-up 2 iteration files and sent to Frank	00:22:36	0.09%
Making up SWD cable, removed button	03:19:14	0.79%
Reading about QFD	00:24:05	0.10%
Communication with Traveler, looked into linker command and ram and flash placement	00:59:46	0.24%
Week 1	00:19:08	0.08%
Added solder to SN6 boar'd R4 and R5 resistors	00:29:13	0.12%
Ethical Issues	00:20:24	0.08%
3D modeled mag sensor holder and sent to Frank to print. Started on link 1 mock-up	03:08:22	0.75%
Charger IC component selection	02:03:13	0.49%
Exporting 3d circuitboard file	00:24:12	0.10%
Board SN1: Tested power, modified button connections, solder battery on, tested voltages	01:26:00	0.34%
Imported new SoC reference circuitry into Eagle	00:49:01	0.20%

•	Silkscreen names and font consideration, altered 'VBUS' polygon	01:21:38	0.32%
•	Prepared power test sheet	01:01:40	0.25%
•	Looking through UART examples. Installed meld to compare code between examples	00:42:08	0.17%
•	Magnetic sensor with teensy - testing concept idea	00:28:50	0.11%
•	Researching sensor solutions	00:38:54	0.15%
•	Charging IC selection, Nordic SoC variant 2 selection	01:21:22	0.32%
•	Presented and attended presentations	00:40:00	0.16%
•	Editing library components, researching silkscreen considerations	01:07:52	0.27%
•	Finished week 3 diary, started week 4	00:11:47	0.05%
•	Presentation Powerpoint almost completed	01:48:08	0.43%
•	Changing DRC rules, Updating PCB V1.1 BOM, sourcing components from digikey	02:49:39	0.67%
•	Shifted solder pad for clearance while soldering	00:34:07	0.14%
•	Trying to understand ble examples	02:10:28	0.52%
•	Showcase evening event	02:45:00	0.66%
•	Sent continuous ble strings over ble uart	01:48:09	0.43%
•	Showed Frank my new test print, went through BLE UART program	00:30:00	0.12%
•	Sanded 3D prints to fit	00:31:00	0.12%
•	Uploading and debugging BLE application	01:46:11	0.42%
•	Completed week 12 diary entries, started week 13 diary	01:29:56	0.36%
•	Remapped SWD connector pins, adding more ground vias	00:18:33	0.07%
•	Finished and submitted week 7, started on week 8	00:22:17	0.09%
•	Week 8 entry	01:03:06	0.25%
•	Plotted pot range, calculated angles	01:01:27	0.24%
•	Soldered pin headers onto nRF52 Dongle	00:30:00	0.12%
•	In-person meeting with Frank: Report structure change, progress update, vscode compiling discussion	00:30:00	0.12%

SWD connector design, I2C interface pins selected	00:43:03	0.17%
Meeting with Frank and Traveler. Clarified some things, got help on others	01:07:00	0.27%
Troubleshooting VSCode errors	01:54:58	0.46%
Meeting with Traveler and Frank: Discussed PCB Manufacturing issue	01:06:00	0.26%
Blinked RGB LED all colours	00:10:41	0.04%
Edited gantt chart, started progress report document	00:37:48	0.15%
Peripherals and connectors schematic design	01:20:35	0.32%
Looked for debugging tutorials. Found I forgot to change memory segments, found current limiting resistors not soldered enough	02:42:45	0.65%
installed v3.3, found it's licensing is outdated. Started downloading V4.2	00:11:55	0.05%
Attempted debugging hardfault	01:20:16	0.32%
3D modelling parts for concept 2	02:04:52	0.50%
PCB general layout, gathered parameter from PCBOne	00:57:43	0.23%
Speaking to Frank about Eagle licensing	00:24:00	0.10%
Methodology, Milestones, Planning	01:13:09	0.29%
Looked into via considerations (thru-hole to inner layers), encountered stock issue with nRF52840	01:57:00	0.46%
Finished charger IC schematic	01:07:58	0.27%
Collected 3D prints from Frank, looked at BLE examples and library documentation	01:25:55	0.34%
Exporting v1.4 prints	00:34:17	0.14%
Drawing full system diagram, started GitHub repo, started plan doc, found central USBD ex. for dongle, fixing build errors	04:14:38	1.01%
Wrote mock-up2 design choices, started 3D modelling iteration of Mock-up2	01:51:22	0.44%
Mechanical interface initial concept drawings, analysis and tests	03:12:43	0.77%

Battery research	00:52:36	0.21%
Uploaded BLE UART example	00:36:41	0.15%
Sketched product, started function structure analysis	01:16:00	0.30%
Learning about software development with nrf	00:59:29	0.24%
Successfully modified and uploaded TWI scanner and found the I2C address. Realised i may need floating point	02:47:50	0.67%
Watched videos on PCB assembly process	00:29:13	0.12%
Report writing	24:59:20	5.96%
Week 10 diary entry and submission, Week 10A diary started	01:01:30	0.25%
Finished week 2 entry. Started week 3 entry	00:05:20	0.02%
Started slide structure	01:10:00	0.28%
Researching high-speed PCB considerations	01:55:16	0.46%
Stdint.h errors in vscode	01:23:43	0.33%
Added extra ground vias	00:31:00	0.12%
Compiled a list of features for V1 prototype	01:20:20	0.32%
carried out experiment	02:25:09	0.58%
Sourcing digi-key components, calculated RGB LED series resistor values	01:02:48	0.25%
Slideshow finished and practicing presentation	03:44:13	0.89%
started attempting BLE UART Example	00:36:38	0.15%
Sent report extension request form	00:24:33	0.10%
Looking at Segger Embedded Studio again	00:48:32	0.19%
Editing Gantt chart and time estimates	00:18:20	0.07%
3D magnetic Sensor datasheet overview	00:58:25	0.23%
Calculated RF transmission impedance, added meandered antenna	01:47:47	0.43%
Defined objectives, posed research questions	01:39:00	0.39%
Uploading and debugging BLE application. Successfully implemented. Tested at range of 7m	02:06:28	0.50%

•	Read up on BLE, sucessfully modified example project	01:32:24	0.37%
•	(Without Description)	11:51:31	2.83%
•	Designed and modelled another mechanical interface concept	01:59:29	0.47%
•	Presented and attended peers' presentations	03:00:00	0.72%
•	Test protractor modelling	01:20:00	0.32%
•	Compiling list of features	00:44:17	0.18%
•	Attempted uploading example from youtube video	01:59:34	0.47%
•	V1.0 Requirements	00:57:24	0.23%
•	Looked for micro USB 3d package in eagle	00:06:00	0.02%
•	Testing potentiometer range using teensy	01:10:05	0.28%
•	resolved 'unplaced sections' error. Successfully blinked LED	01:26:09	0.34%
•	Examined and fixed reflow errors on all boards. Electrically tested board serial number 4	01:00:00	0.24%
•	Applied mag sensor to mock-up 1	01:46:57	0.42%
•	Updated packages (added sild outline), Started routing, added outer fence vias	02:55:34	0.70%
•	Checked BOM quantities, exported digikey shopping cart, sent to Traveler	00:55:11	0.22%
•	Setup the template	00:18:23	0.07%
•	Sent print models to Frank	00:03:32	0.01%
•	Attempted to modify BLE UART example - unsuccessful, don't know enough	00:57:37	0.23%
•	Testing programs on boards; S/N 1, 2, 3, 4, 5	02:13:27	0.53%
•	Updating PCB V1.1 BOM, sourcing components from digikey	01:33:37	0.37%
•	attmpted ble uart again, bricked the dongle	01:12:39	0.29%
•	Meeting with Traveler and Frank	01:29:46	0.36%
•	Added more planning considerations	00:10:53	0.04%
•	Voltage regulator research	01:02:27	0.25%

•	MicroUSB and Charger IC Placement	00:22:04	0.09%
•	Voltage regulator pours and battery terminals	00:59:29	0.24%
•	Decision to wait for Frank's J-links to arrive, asked him about included connectors	00:48:31	0.19%
•	Updated RGB LED and micro USB footprint silkscreen	00:21:39	0.09%
•	Finished poster draft and sent to Frank and James	01:56:29	0.46%
•	Testing BLE UART, power consumption	00:56:31	0.22%
•	Labelled I2C signals	00:06:00	0.02%
•	Discovered license issue with Eagle, communicated with supervisor	00:44:05	0.17%
•	Breakout pins schematic, added pads	01:11:31	0.28%
•	Drafting needs statement	00:36:03	0.14%
•	Attempted modifying UART example	02:55:03	0.70%
•	Wrote instructions for modifying SDK examples for armature board	01:34:07	0.37%
•	Writing up potentiometer results	01:52:29	0.45%
•	Reinstalled GCC compiler, uploaded BLE UART axample and send data from Putty and Android	01:24:00	0.33%
•	BLE notification research	00:14:40	0.06%
•	Gantt Chart	01:15:26	0.30%
•	Sourced Crystal oscillators, determined capacitance, sourced various capacitors	02:26:32	0.58%
•	Started email about mechanical lab weights	00:02:28	0.01%
•	Modelled link 1 cap, adjusted magnet position, exported model and sent to Frank. Exported files to print	02:03:03	0.49%
•	Started laying out MicroUSB, Charger IC and their passives	01:34:51	0.38%
•	Successfully built and uploaded program. However, no board activity	01:26:17	0.34%
•	Posed research questions, started research	00:45:37	0.18%
•	Compiling errors - redeclaration of m4 core symbols	00:53:20	0.21%
•	Started PCB stops, wedge slots	00:45:23	0.18%

Sourced new micro USB, researched trace antennas	01:42:32	0.41%
Aquired eagle reference files for nRF52840 chip	00:59:19	0.24%
Exported parts to 3D print - Snap fit concept	01:16:09	0.30%
Looked through nRF51x22 datasheet	01:00:24	0.24%
Programmed BLE blinky example on dongle	01:22:29	0.33%
Experimenting with 3D printed parts and magnetic sensor	01:00:00	0.24%
Reading through BLE Example. Successfully continuously streamed 8bit data	02:58:55	0.71%
Sent solidworks model to frank	00:12:19	0.05%
Antenna considerations research	01:04:10	0.26%
Eagle schematic	01:47:51	0.43%
Drew mechanical interface idea, planned work for tomorrow	00:24:35	0.10%
Learning about software development with nrf. Raised concerns about software functionality with Frank	01:11:15	0.28%
Getting started guides for programming dongle	00:49:33	0.20%
Structuring notes	00:43:28	0.17%
Attempted adding logo to eagle, exported, checked and sent gerbers	03:00:00	0.72%
Reading up on possible report structures	00:46:24	0.18%
Meeting with Frank. Harjeev was present	00:20:00	0.08%
Completed and submitted week 9 diary, setup week 10 diary	00:56:00	0.22%
Sensors research	01:47:19	0.43%
PCB Planning, component selection	01:36:13	0.38%
Configuring VS Code intellisense - got rid of most errors	00:15:46	0.06%
Finished charger IC copper pours, started regulator layout	01:25:15	0.34%
Tested Jlink commander and Jlink flash lite	00:26:55	0.11%
Searching for UART examples	00:34:08	0.14%
Solidworks design work on snap fit concept	01:34:40	0.38%
Looked into custom nRF board programming	00:39:19	0.16%

•	Collected mechanical structure v1.1 test print and J-link	01:10:00	0.28%
•	Presentation Practice	01:34:00	0.37%
•	Designed roller bearing	01:07:59	0.27%
•	Trying to solve error	03:29:34	0.83%
•	Imported PCB model, modelled magnet	01:48:18	0.43%
•	Supply voltage PCB requirements	01:04:55	0.26%
•	Time tracker planning, File organisation, Compiled Slack communication	01:30:11	0.36%
•	setting up design rules for PCBZone, fixing packages	01:05:55	0.26%
•	Looked into BLE blinky and UART examples with Frank	00:50:00	0.20%
•	Preparation	03:45:00	0.89%
•	Week 3 entry	00:32:15	0.13%
•	Call with Traveler: Walked through his thought process - I2c hardware, software implemention	01:17:00	0.31%
•	Submitted week 6, setup week 7 diary	00:31:57	0.13%
•	rendered exploded view	02:21:53	0.56%
•	Soldered battery onto different board	00:30:00	0.12%
•	FInished week 4. Week 5 entry	00:26:31	0.10%
•	Researching magnetic field simulators	00:58:08	0.23%
•	Sourced resistors from Digi-key, determining specs of high frequency crystal oscillator	02:12:38	0.53%
•	Finished and submitted synopsis	01:17:12	0.31%
•	Switch schematic done, TLV493D schematic started	01:24:44	0.34%
•	Sanded PCB tabs to fit in the test print, tested battery fit	00:56:15	0.22%
•	installed nRF52 v14.2 SDK, SoftDevice s132 v5.1, started custom ble service example tutorial	01:39:45	0.40%
•	Week 6 entry	00:33:17	0.13%
•	Found PCBOne's design rules	00:11:58	0.05%

continued development section	00:47:27	0.19%
Specification/performance evaluation methods	01:05:31	0.26%
Added and positioned component names, edited pin 1 marks, added board name	02:08:23	0.51%
General Layout	01:55:29	0.46%
Testing boards' power/gnd resistances. Found error in button schematic	02:00:51	0.48%
Setup 3D magnetic sensor with Teensy. Read through libraries	01:15:54	0.30%
Jitsi Meeting with Frank and Traveler. Discussed Project	00:35:00	0.14%
setting up design rules for PCBZone	00:15:00	0.06%
Learning about magnetic fields - Youtube, Khan Academy, wikipedia	01:47:22	0.43%
Researching mechanical/manufacturing considerations	01:07:04	0.27%
Assembling PCB BOM, sourcing parts on digi-key	01:13:46	0.29%
Researching antenna tuning considerations, checking BOM and datasheets	01:15:16	0.30%
Proposal Ethical Issues, communication with Frank and Traveler	01:34:48	0.38%
Updated weekly diaries	00:27:28	0.11%
Diary entries, compiled and submitted	00:14:07	0.06%
Researching design rules	00:53:35	0.21%
Modelled and laser cut custom protractor	02:15:56	0.54%
Attempted setting up dongle with PlatformIO	00:45:44	0.18%
Exported 3D prints	00:51:27	0.20%
Finishing and submitting proposal report	03:45:10	0.89%
Diary entry	00:18:48	0.07%
Successfully built project from vs code	03:22:03	0.80%
Noting communication on confidentiality and IP issues	00:04:21	0.02%
Testing mock-up 1 with teensy, researching alternative magnets	02:45:17	0.66%
Weeks 10A and 10B entries and submission	01:28:14	0.35%

●	Collected 3D printed parts, spoke about magnets, PCB antenna, 3D printed bearing	00:30:00	0.12%
●	Sourcing debug cables/communication with Traveler	02:05:38	0.50%
●	Week 2 entries	00:15:00	0.06%
●	Modelled pegs for v1.0 struct. test print, prepared test print files, sanded down v0.1 print	02:17:33	0.55%
●	Submitted week 12 diary	00:08:40	0.03%

Project / Description	Duration	Amount
MG7101 Engineering Development Project	419:33:46	0.00 USD
Meeting on Zoom with Frank and Chris	00:25:00	0.00 USD
Met with Frank: Figured out virtual PCB model, spoke about synopsis and product to demonstrate	01:20:00	0.00 USD
Sent V1 features to Frank	00:28:49	0.00 USD
Started eagle schematic design, sourced Hall sensor components	03:23:27	0.00 USD
Presentation feedback, discussed next steps (mechanical, programmer, UART)	00:32:00	0.00 USD
Watched pcb assembly video	01:36:33	0.00 USD
Met course coordinator, introduction to course	01:15:00	0.00 USD
Weighing options on PCB manufacturing	01:54:20	0.00 USD
PCB manufacturing issue found, laying out components in PCB	01:36:16	0.00 USD
Charger IC copper pours	01:01:40	0.00 USD
New SoC selection	01:07:40	0.00 USD
Sourcing parts on digi-key from BOM	00:56:31	0.00 USD
Modelling PCB USB, USB connector cutout, PCB tabs and stops	02:10:51	0.00 USD
Introduction finished, development section started	01:34:57	0.00 USD

Week 14 diary entries and submission	00:54:43	0.00 USD
Attended 3 Presentations	01:00:00	0.00 USD
Discussed new SoC options, progress update on PCB	00:38:00	0.00 USD
Reinstaled nRF52v17 SDK for fresh start, found guide for developing for nRF52810, looked at TWI documentation	00:20:42	0.00 USD
Analysing structure v1.0 test print, made changes for v1.1 test print	01:58:57	0.00 USD
Build attempts	01:15:21	0.00 USD
Determined max needed power supply current	00:46:57	0.00 USD
Softdevice webinar note taking	01:11:33	0.00 USD
Power supply layout approx. done, added reset pullup resistor	02:51:47	0.00 USD
Attempted uploading blink example. Setting up board files	01:15:34	0.00 USD
Producing Mock-up drawings for write-up	00:40:18	0.00 USD
Started writing up the mock-up design choices	00:34:09	0.00 USD
Setup nRF52840 reference Design in Eagle	01:17:58	0.00 USD
Week 7 entry	00:04:07	0.00 USD
Researching debuggers and connections, Traveler's Atmel-ICE	01:12:29	0.00 USD
TLV493 Placement	00:54:23	0.00 USD
Running into Unplaced sections overflow error	02:45:54	0.00 USD
Tried offset mag sensor idea	01:06:43	0.00 USD
Calculated torque results	01:05:17	0.00 USD
Wrote up test results of the magnet on plate mock-up	01:29:07	0.00 USD
Time estimates for future work, emailed coordinator about concerns with finishing	00:39:18	0.00 USD
Finalised first draft and sent to James	02:10:18	0.00 USD
Followed tutorial on short blink program, programmed board, couldn't debug	01:26:43	0.00 USD

Week 8 entry and submission	00:30:03	0.00 USD
Uploaded BLE UART example, started modifying it	02:08:28	0.00 USD
Drawing mechanical concept idea, tested potentiometer with teensy	01:42:13	0.00 USD
Presentation Powerpoint, Gantt Chart Template Implemented	02:02:37	0.00 USD
Completed and sent structure v1.1 test print to Frank	02:27:08	0.00 USD
Ordered Diametrically magnetized cylindrical magnets	00:11:13	0.00 USD
Learning Seggers IDE and getting it working	03:08:43	0.00 USD
Function structure analysis	02:11:58	0.00 USD
Communication with Frank and Traveler	00:20:00	0.00 USD
Produced and sent Frank STL files for 2nd mockup mechanical interface	00:25:23	0.00 USD
Wrote up the mock-up design choices	00:59:09	0.00 USD
Investigating nRF52 SDK examples	00:45:00	0.00 USD
continued development section and photoshoot	00:29:59	0.00 USD
Week 15 entry	00:28:24	0.00 USD
Read and signed a confidentiality agreement with MechAdept	00:51:52	0.00 USD
Configuring VS Code intellisense	01:50:53	0.00 USD
PCB general layout	02:47:16	0.00 USD
Week 4 entry	00:09:11	0.00 USD
Assembled 6 Armature joint boards	08:00:00	0.00 USD
Voltage Regulator Selection, passives selection	00:46:18	0.00 USD
Scope definition	00:38:40	0.00 USD
Programmed nrf52 blinky example	02:36:00	0.00 USD
Modelling V1.0 mechanical structure link 1	01:04:06	0.00 USD
Watching C programming tutorial	00:19:11	0.00 USD

Call with Frank and Traveler: Feedback on concept ideas, questions on I2C, Debuggers,	00:35:00	0.00 USD
Fixed export issues and ordered boards	00:40:00	0.00 USD
Week 11 entries and submission	00:44:22	0.00 USD
Acquired files from Traveler	00:12:41	0.00 USD
Compared BLE central and peripheral programs	00:36:30	0.00 USD
setting up experiment	01:33:31	0.00 USD
Presentation Powerpoint	01:08:10	0.00 USD
Investigating features	00:44:18	0.00 USD
Started presentation slides	00:53:53	0.00 USD
Started working on program design. Stepped through BLE UART example with debugger while taking notes.	03:29:22	0.00 USD
Wrote up some requirements and started planning	00:38:45	0.00 USD
Voltage Regulator Selection	00:22:04	0.00 USD
Submitted week 5 diary, setup week 6 diary	00:22:22	0.00 USD
Requested report deadline extension	00:08:00	0.00 USD
Getting joints fitting together properly for showcase	01:16:35	0.00 USD
Preparing for PCB assembly at MechAdept	00:12:32	0.00 USD
Routing almost complete	01:37:03	0.00 USD
Updated time tracker tags	00:05:59	0.00 USD
Almost finished modelling link2	01:00:04	0.00 USD
Spoke with Traveler about deadlines, help with software. Watching webinar on softdevice	01:37:00	0.00 USD
Project Planning	00:46:00	0.00 USD
Travelled to campus, picked up 3D Print, made leads for sensor	03:25:00	0.00 USD
Component research - Battery holder, switches, SoC power considerations	02:22:34	0.00 USD

SWD connector finished, fixed pad size issue in eagle	01:13:59	0.00 USD
Assembled and glued mockup 3	01:34:08	0.00 USD
Outlined Mechanical, Electrical, and software considerations. Started on objectives	01:15:34	0.00 USD
Electronics block diagram, battery and voltage regulator research	01:24:28	0.00 USD
Design methodology	01:53:11	0.00 USD
3D modeled and exported mockup files for printing	03:28:21	0.00 USD
Downloading Jlink V6.2d and SES v3.3	00:56:23	0.00 USD
Potentiometer interface concept drawings and analysis	01:14:46	0.00 USD
Exported mock-up 2 iteration files and sent to Frank	00:22:36	0.00 USD
Making up SWD cable, removed button	03:19:14	0.00 USD
Reading about QFD	00:24:05	0.00 USD
Communication with Traveler, looked into linker command and ram and flash placement	00:59:46	0.00 USD
Week 1	00:19:08	0.00 USD
Added solder to SN6 boar'd R4 and R5 resistors	00:29:13	0.00 USD
Ethical Issues	00:20:24	0.00 USD
3D modeled mag sensor holder and sent to Frank to print. Started on link 1 mock-up	03:08:22	0.00 USD
Charger IC component selection	02:03:13	0.00 USD
Exporting 3d circuitboard file	00:24:12	0.00 USD
Board SN1: Tested power, modified button connections, solder battery on, tested voltages	01:26:00	0.00 USD
Imported new SoC reference circuitry into Eagle	00:49:01	0.00 USD
Silkscreen names and font consideration, altered 'VBUS' polygon	01:21:38	0.00 USD
Prepared power test sheet	01:01:40	0.00 USD

Looking through UART examples. Installed meld to compare code between examples	00:42:08	0.00 USD
Magnetic sensor with teensy - testing concept idea	00:28:50	0.00 USD
Researching sensor solutions	00:38:54	0.00 USD
Charging IC selection, Nordic SoC variant 2 selection	01:21:22	0.00 USD
Presented and attended presentations	00:40:00	0.00 USD
Editing library components, researching silkscreen considerations	01:07:52	0.00 USD
Finished week 3 diary, started week 4	00:11:47	0.00 USD
Presentation Powerpoint almost completed	01:48:08	0.00 USD
Changing DRC rules, Updating PCB V1.1 BOM, sourcing components from digikey	02:49:39	0.00 USD
Shifted solder pad for clearance while soldering	00:34:07	0.00 USD
Trying to understand ble examples	02:10:28	0.00 USD
Showcase evening event	02:45:00	0.00 USD
Sent continuous ble strings over ble uart	01:48:09	0.00 USD
Showed Frank my new test print, went through BLE UART program	00:30:00	0.00 USD
Sanded 3D prints to fit	00:31:00	0.00 USD
Uploading and debugging BLE application	01:46:11	0.00 USD
Completed week 12 diary entries, started week 13 diary	01:29:56	0.00 USD
Remapped SWD connector pins, adding more ground vias	00:18:33	0.00 USD
Finished and submitted week 7, started on week 8	00:22:17	0.00 USD
Week 8 entry	01:03:06	0.00 USD
Plotted pot range, calculated angles	01:01:27	0.00 USD
Soldered pin headers onto nRF52 Dongle	00:30:00	0.00 USD
In-person meeting with Frank: Report structure change, progress update, vscode compiling discussion	00:30:00	0.00 USD

SWD connector design, I2C interface pins selected	00:43:03	0.00 USD
Meeting with Frank and Traveler. Clarified some things, got help on others	01:07:00	0.00 USD
Troubleshooting VSCode errors	01:54:58	0.00 USD
Meeting with Traveler and Frank: Discussed PCB Manufacturing issue	01:06:00	0.00 USD
Blinked RGB LED all colours	00:10:41	0.00 USD
Edited gantt chart, started progress report document	00:37:48	0.00 USD
Peripherals and connectors schematic design	01:20:35	0.00 USD
Looked for debugging tutorials. Found I forgot to change memory segments, found current limiting resistors not soldered enough	02:42:45	0.00 USD
installed v3.3, found it's licensing is outdated. Started downloading V4.2	00:11:55	0.00 USD
Attempted debugging hardfault	01:20:16	0.00 USD
3D modelling parts for concept 2	02:04:52	0.00 USD
PCB general layout, gathered parameter from PCBOne	00:57:43	0.00 USD
Speaking to Frank about Eagle licensing	00:24:00	0.00 USD
Methodology, Milestones, Planning	01:13:09	0.00 USD
Looked into via considerations (thru-hole to inner layers), encountered stock issue with nRF52840	01:57:00	0.00 USD
Finished charger IC schematic	01:07:58	0.00 USD
Collected 3D prints from Frank, looked at BLE examples and library documentation	01:25:55	0.00 USD
Exporting v1.4 prints	00:34:17	0.00 USD
Drawing full system diagram, started GitHub repo, started plan doc, found central USBD ex. for dongle, fixing build errors	04:14:38	0.00 USD
Wrote mock-up2 design choices, started 3D modelling iteration of Mock-up2	01:51:22	0.00 USD
Mechanical interface initial concept drawings, analysis and tests	03:12:43	0.00 USD

Battery research	00:52:36	0.00 USD
Uploaded BLE UART example	00:36:41	0.00 USD
Sketched product, started function structure analysis	01:16:00	0.00 USD
Learning about software development with nrf	00:59:29	0.00 USD
Successfully modified and uploaded TWI scanner and found the I2C address. Realised i may need floating point	02:47:50	0.00 USD
Watched videos on PCB assembly process	00:29:13	0.00 USD
Report writing	24:59:20	0.00 USD
Week 10 diary entry and submission, Week 10A diary started	01:01:30	0.00 USD
Finished week 2 entry. Started week 3 entry	00:05:20	0.00 USD
Started slide structure	01:10:00	0.00 USD
Researching high-speed PCB considerations	01:55:16	0.00 USD
Stdint.h errors in vscode	01:23:43	0.00 USD
Added extra ground vias	00:31:00	0.00 USD
Compiled a list of features for V1 prototype	01:20:20	0.00 USD
carried out experiment	02:25:09	0.00 USD
Sourcing digi-key components, calculated RGB LED series resistor values	01:02:48	0.00 USD
Slideshow finished and practicing presentation	03:44:13	0.00 USD
started attempting BLE UART Example	00:36:38	0.00 USD
Sent report extension request form	00:24:33	0.00 USD
Looking at Segger Embedded Studio again	00:48:32	0.00 USD
Editing Gantt chart and time estimates	00:18:20	0.00 USD
3D magnetic Sensor datasheet overview	00:58:25	0.00 USD
Calculated RF transmission impedance, added meandered antenna	01:47:47	0.00 USD

Defined objectives, posed research questions	01:39:00	0.00 USD
Uploading and debugging BLE application. Successfully implemented. Tested at range of 7m	02:06:28	0.00 USD
Read up on BLE, sucessfully modified example project	01:32:24	0.00 USD
(Without Description)	11:51:31	0.00 USD
Designed and modelled another mechanical interface concept	01:59:29	0.00 USD
Presented and attended peers' presentations	03:00:00	0.00 USD
Test protractor modelling	01:20:00	0.00 USD
Compiling list of features	00:44:17	0.00 USD
Attempted uploading example from youtube video	01:59:34	0.00 USD
V1.0 Requirements	00:57:24	0.00 USD
Looked for micro USB 3d package in eagle	00:06:00	0.00 USD
Testing potentiometer range using teensy	01:10:05	0.00 USD
resolved 'unplaced sections' error. Successfully blinked LED	01:26:09	0.00 USD
Examined and fixed reflow errors on all boards. Electrically tested board serial number 4	01:00:00	0.00 USD
Applied mag sensor to mock-up 1	01:46:57	0.00 USD
Updated packages (added sild outline), Started routing, added outer fence vias	02:55:34	0.00 USD
Checked BOM quantities, exported digikey shopping cart, sent to Traveler	00:55:11	0.00 USD
Setup the template	00:18:23	0.00 USD
Sent print models to Frank	00:03:32	0.00 USD
Attempted to modify BLE UART example - unsuccessful, don't know enough	00:57:37	0.00 USD
Testing programs on boards; S/N 1, 2, 3, 4, 5	02:13:27	0.00 USD
Updating PCB V1.1 BOM, sourcing components from digikey	01:33:37	0.00 USD

attmpted ble uart again, bricked the dongle	01:12:39	0.00 USD
Meeting with Traveler and Frank	01:29:46	0.00 USD
Added more planning considerations	00:10:53	0.00 USD
Voltage regulator research	01:02:27	0.00 USD
MicroUSB and Charger IC Placement	00:22:04	0.00 USD
Voltage regulator pours and battery terminals	00:59:29	0.00 USD
Decision to wait for Frank's J-links to arrive, asked him about included connectors	00:48:31	0.00 USD
Updated RGB LED and micro USB footprint silkscreen	00:21:39	0.00 USD
Finished poster draft and sent to Frank and James	01:56:29	0.00 USD
Testing BLE UART, power consumption	00:56:31	0.00 USD
Labelled I2C signals	00:06:00	0.00 USD
Discovered license issue with Eagle, communicated with supervisor	00:44:05	0.00 USD
Breakout pins schematic, added pads	01:11:31	0.00 USD
Drafting needs statement	00:36:03	0.00 USD
Attempted modifying UART example	02:55:03	0.00 USD
Wrote instructions for modifying SDK examples for armature board	01:34:07	0.00 USD
Writing up potentiometer results	01:52:29	0.00 USD
Reinstalled GCC compiler, uploaded BLE UART axample and send data from Putty and Android	01:24:00	0.00 USD
BLE notification research	00:14:40	0.00 USD
Gantt Chart	01:15:26	0.00 USD
Sourced Crystal oscillators, determined capacitance, sourced various capacitors	02:26:32	0.00 USD
Started email about mechanical lab weights	00:02:28	0.00 USD
Modelled link 1 cap, adjusted magnet position, exported model and sent to Frank. Exported files to print	02:03:03	0.00 USD

Started laying out MicroUSB, Charger IC and their passives	01:34:51	0.00 USD
Successfully built and uploaded program. However, no board activity	01:26:17	0.00 USD
Posed research questions, started research	00:45:37	0.00 USD
Compiling errors - redeclaration of m4 core symbols	00:53:20	0.00 USD
Started PCB stops, wedge slots	00:45:23	0.00 USD
Sourced new micro USB, researched trace antennas	01:42:32	0.00 USD
Aquired eagle reference files for nRF52840 chip	00:59:19	0.00 USD
Exported parts to 3D print - Snap fit concept	01:16:09	0.00 USD
Looked through nRF51x22 datasheet	01:00:24	0.00 USD
Programmed BLE blinky example on dongle	01:22:29	0.00 USD
Experimenting with 3D printed parts and magnetic sensor	01:00:00	0.00 USD
Reading through BLE Example. Successfully continuously streamed 8bit data	02:58:55	0.00 USD
Sent solidworks model to frank	00:12:19	0.00 USD
Antenna considerations research	01:04:10	0.00 USD
Eagle schematic	01:47:51	0.00 USD
Drew mechanical interface idea, planned work for tomorrow	00:24:35	0.00 USD
Learning about software development with nrf. Raised concerns about software functionality with Frank	01:11:15	0.00 USD
Getting started guides for programming dongle	00:49:33	0.00 USD
Structuring notes	00:43:28	0.00 USD
Attempted adding logo to eagle, exported, checked and sent gerbers	03:00:00	0.00 USD
Reading up on possible report structures	00:46:24	0.00 USD
Meeting with Frank. Harjeev was present	00:20:00	0.00 USD
Completed and submitted week 9 diary, setup week 10 diary	00:56:00	0.00 USD

Sensors research	01:47:19	0.00 USD
PCB Planning, component selection	01:36:13	0.00 USD
Configuring VS Code intellisense - got rid of most errors	00:15:46	0.00 USD
Finished charger IC copper pours, started regulator layout	01:25:15	0.00 USD
Tested Jlink commander and Jlink flash lite	00:26:55	0.00 USD
Searching for UART examples	00:34:08	0.00 USD
Solidworks design work on snap fit concept	01:34:40	0.00 USD
Looked into custom nRF board programming	00:39:19	0.00 USD
Collected mechanical structure v1.1 test print and J-link	01:10:00	0.00 USD
Presentation Practice	01:34:00	0.00 USD
Designed roller bearing	01:07:59	0.00 USD
Trying to solve error	03:29:34	0.00 USD
Imported PCB model, modelled magnet	01:48:18	0.00 USD
Supply voltage PCB requirements	01:04:55	0.00 USD
Time tracker planning, File organisation, Compiled Slack communication	01:30:11	0.00 USD
setting up design rules for PCBZone, fixing packages	01:05:55	0.00 USD
Looked into BLE blinky and UART examples with Frank	00:50:00	0.00 USD
Preparation	03:45:00	0.00 USD
Week 3 entry	00:32:15	0.00 USD
Call with Traveler: Walked through his thought process - I2c hardware, software implemention	01:17:00	0.00 USD
Submitted week 6, setup week 7 diary	00:31:57	0.00 USD
rendered exploded view	02:21:53	0.00 USD
Soldered battery onto different board	00:30:00	0.00 USD

FInished week 4. Week 5 entry	00:26:31	0.00 USD
Researching magnetic field simulators	00:58:08	0.00 USD
Sourced resistors from Digi-key, determining specs of high frequency crystal oscillator	02:12:38	0.00 USD
Finished and submitted synopsis	01:17:12	0.00 USD
Switch schematic done, TLV493D schematic started	01:24:44	0.00 USD
Sanded PCB tabs to fit in the test print, tested battery fit	00:56:15	0.00 USD
installed nRF52 v14.2 SDK, SoftDevice s132 v5.1, started custom ble service example tutorial	01:39:45	0.00 USD
Week 6 entry	00:33:17	0.00 USD
Found PCBOne's design rules	00:11:58	0.00 USD
continued development section	00:47:27	0.00 USD
Specification/performance evaluation methods	01:05:31	0.00 USD
Added and positioned component names, edited pin 1 marks, added board name	02:08:23	0.00 USD
General Layout	01:55:29	0.00 USD
Testing boards' power/gnd resistances. Found error in button schematic	02:00:51	0.00 USD
Setup 3D magnetic sensor with Teensy. Read through libraries	01:15:54	0.00 USD
Jitsi Meeting with Frank and Traveler. Discussed Project	00:35:00	0.00 USD
setting up design rules for PCBZone	00:15:00	0.00 USD
Learning about magnetic fields - Youtube, Khan Academy, wikipedia	01:47:22	0.00 USD
Researching mechanical/manufacturing considerations	01:07:04	0.00 USD
Assembling PCB BOM, sourcing parts on digi-key	01:13:46	0.00 USD
Researching antenna tuning considerations, checking BOM and datasheets	01:15:16	0.00 USD
Proposal Ethical Issues, communication with Frank and Traveler	01:34:48	0.00 USD

Updated weekly diaries	00:27:28	0.00 USD
Diary entries, compiled and submitted	00:14:07	0.00 USD
Researching design rules	00:53:35	0.00 USD
Modelled and laser cut custom protractor	02:15:56	0.00 USD
Attempted setting up dongle with PlatformIO	00:45:44	0.00 USD
Exported 3D prints	00:51:27	0.00 USD
Finishing and submitting proposal report	03:45:10	0.00 USD
Diary entry	00:18:48	0.00 USD
Successfully built project from vs code	03:22:03	0.00 USD
Noting communication on confidentiality and IP issues	00:04:21	0.00 USD
Testing mock-up 1 with teensy, researching alternative magnets	02:45:17	0.00 USD
Weeks 10A and 10B entries and submission	01:28:14	0.00 USD
Collected 3D printed parts, spoke about magnets, PCB antenna, 3D printed bearing	00:30:00	0.00 USD
Sourcing debug cables/communication with Traveler	02:05:38	0.00 USD
Week 2 entries	00:15:00	0.00 USD
Modelled pegs for v1.0 struct. test print, prepared test print files, sanded down v0.1 print	02:17:33	0.00 USD
Submitted week 12 diary	00:08:40	0.00 USD

Appendix G – Weekly Diaries

See the following pages for the project’s weekly diaries.

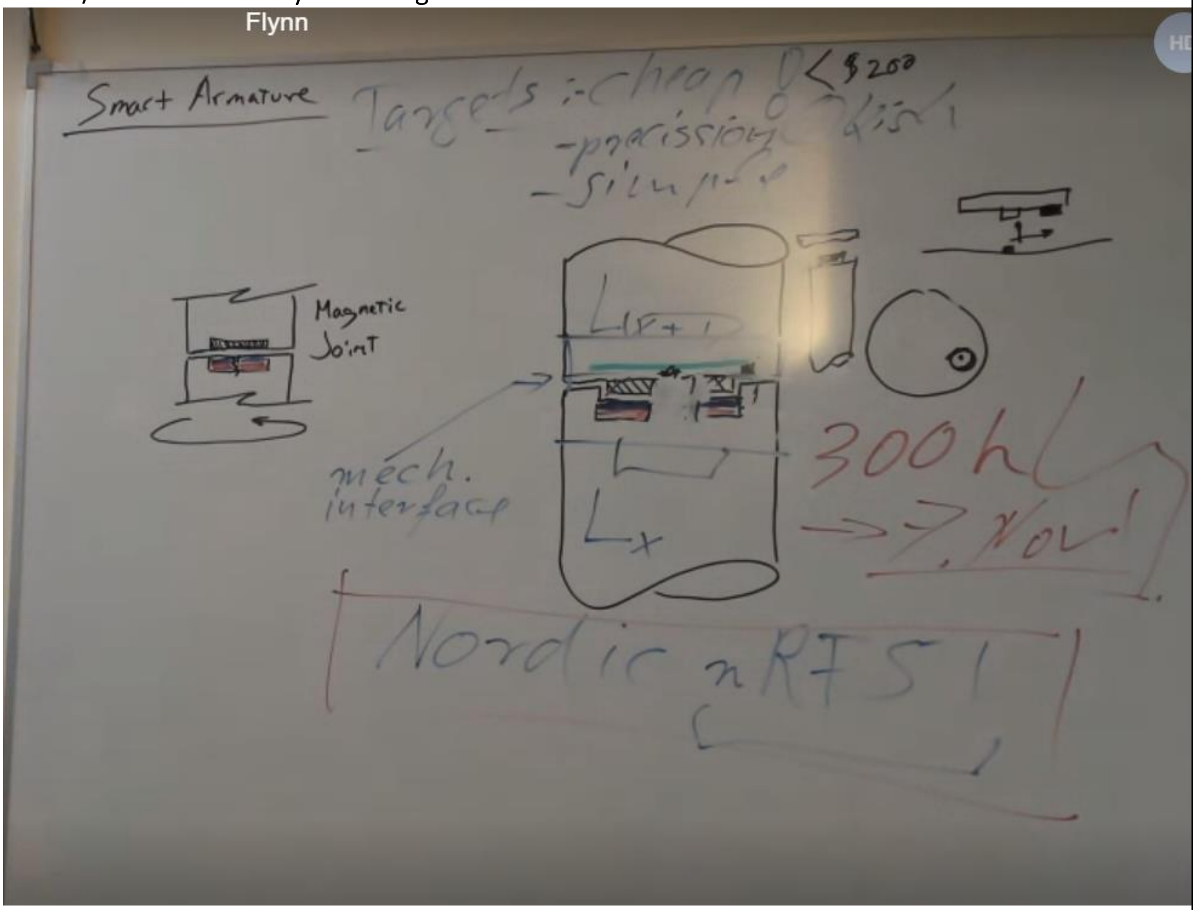
MG 7101 Engineering Development Project – Weekly Diary 1

Student Name: Flynn Stilwell   Supervisor: Frank Beinersdorf

Week Number   From: **20/07/2020**   To: **26/07/2020**

Day	**Work Done** (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Reached out to Frank Beinersdorf to be my supervisor – he accepted. Confirmed a time to meet Frank and Traveler Hauptman virtually. Traveler is the Principal and Director of MechAdept.
Tuesday	
Wednesday	
Thursday	
Friday	Had a meeting with Frank and Traveler at 11am. They presented an idea for my project – a smart armature.
Saturday	Drafted needs/wishes statement from meeting

Sketch/notes from Friday's meeting:



Number of Project Hours This week 4.75	Cumulative Project Hours 4.75	Student's Signature ………………………………	Supervisor's Signature ………………………………

MG 7101 Engineering Development Project – Weekly Diary 2

Student Name: Flynn Stilwell   Supervisor: Frank Beinersdorf

Week Number   From: **27/07/2020**   To: **02/07/2020**

Day	Work Done (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Met Frank at 12pm. Chris and Harjeev were present. Received hardware from Frank; 3D printed joint mockup with magnets, 2 3D magnetic sensors, 2 bluetooth dongles (nRF51)
Tuesday	Given Traveler's contact email.
Wednesday	Meeting with Frank and Traveler at 3:50pm. Briefly checked feasibility of constructing PCB, advice on where to look for information re. software and hardware clarified a focus on mechanical and PCB rather than software.
Thursday	Meeting with Frank at 3:10pm. Advice on scope definition, research question definition, clarification on use for armature. Defined objectives, posed some research questions
Friday	
Saturday	Proposal report work. Function structure diagram

Use this space for any sketches, supervisor comments etc.

Frank suggested having dedicate weekly meeting times with Chris, Harjeev and I present

Q's for Frank:

Q: Is the use of libraries sufficient? Or write my own? E.g. for the 3D magnetic sensor

Q: Approach for system diagram?

A: Follow a function structure analysis approach

Q's for Frank and Traveler:

Q: How could I narrow my scope?

A: Can focus more on PCB and Mechanical design then just get the software running. PCB doesn't have to be perfect – first iteration is enough. Possibility of improving if there's time. Keep focus on mechanical as this is where you can get creative with design – PCB is basically combining existing boards.

Q: How long does PCB manufacturing take? Who with? Where to get components?

A: Should get PCBs back within two weeks with pcbzone.net. Digikey tends to be quickest.

Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
15.5	20.25	………………………………	………………………………

MG 7101 Engineering Development Project – Weekly Diary 3

Student Name: Flynn Stilwell   Supervisor: Frank Beinersdorf

Week Number   From: **03/08/2020**   To: **09/08/2020**

Day	Work Done (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Configuring Segger IDE, worked on proposal report scope definition
Tuesday	Attended 3 presentations in class
Wednesday	Proposal report methodology, finished most of proposal presentation, communicated with Frank and Traveler on Element, practiced presentation
Thursday	Gantt chart complete, practiced presentation with Harjeev, presented, met with Frank at 3:15pm and discussed next steps (mechanical, programmer, UART)
Friday	Mechanical/manufacturing research, sensors research, investigated features of nRF52840 chip, uploaded a BLE UART example
Saturday	Read up on Bluetooth low energy, successfully modified an example project
Sunday	Edited Gantt Chart estimates, compiled a list of v1 features

Use this space for any sketches, supervisor comments etc.

Q: Are there any facilities I could use for the PCB assembly? I have only assembled 0805 packages with a soldering iron.
A: yes we have a small reflow oven (manual placing of the parts).... assembly can be done by you (and we help :))

Q: Are the dongles and sensors that have already been provided included in the budget?
A: Yes sure

Thursday 06/08 Meeting with Frank:
He would be happy with my 'V0.1' prototype with the breadboarded system separate from the mechanical structure. Raised concerns over no on-board debugger for the nRF52 dongle (no dedicated UART module either). Discussed ways to terminate dongle pins to breadboard: Either just solder male pin headers onto dongle or engrave a breakout board with LPKF CNC engraver at Weltec.
Tasks to be done by next week

- Investigate UART over Bluetooth
- Compile a list of features to add on PCB (with justification) to suggest to Traveler
- Compile a list of mechanical features to suggest
- Start working on mechanical structure concepts

Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
25.5	45.75	………………………………	………………………………

MG 7101 Engineering Development Project – Weekly Diary 4

Student Name: Flynn Stilwell Supervisor: Frank Beinersdorf

Week Number From: **10/08/2020** To: **16/08/2020**

Day	**Work Done** (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Soldered pin headers onto nRF52 Dongle
Tuesday	Attended 2 presentations online via zoom
Wednesday	Proposal report work – Ethical issues section, methodology, planning. 3D Hall sensor configuration with teensy, presented features list to Traveler and Frank
Thursday	Meeting with Frank with Chris present. Potentiometer configuration with teensy
Friday	Jitsi all with Frank and Traveler: Feedback on concept ideas, questions on I2C, Debuggers. Call with Traveler: walkthrough of Traveler's thinking regarding hardware configuration, libraries, example adaption all in nRF52840 chip/dongle
Saturday	Khan academy magnetic fields, feedback from Frank on magnetic field simulators – probably won't have access to them in industry
Sunday	

11/08/20
To do tomorrow:

- Work on proposal report
  - Project Methodology – produce outline for design phases
  - Ethical Issues – Write about Traveler's comments, safety issues (COVID 19), sustainability
- Aim to experiment with Hall sensors

13/08/2020
Supervisor Meeting:
Asked about I2C – Frank recommended asking Traveler. Notified Frank of Hall sensor progress, he mentioned Traveler got the same sensor going on STM32 chip – Could be useful for Nordic Chips. Presented idea on annular snap fit mech. interface, he said it would be a good discussion point to bring up with Traveler.

14/08/2020
Traveler Meeting:
Went through procedure of configuring software and hardware for the nRF52840 Dongle.
Useful software for comparing files: Meld
Two wire interface config – use pins 0.15 and 0.17 on dongle for I2C

Weekend work:
Aim to get UART communication going and start configuring example for I2C

Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
18	78.25	FStilwell …………………………………	…………………………………

MG 7101 Engineering Development Project – Weekly Diary

Student Name: Flynn Stilwell   Supervisor: Frank Beinersdorf

Week Number 5   From: **17/08/2020**   To: **23/08/2020**

Day	**Work Done** (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Played with mag sensor using Teensy, successfully compiled program using vscode as IDE
Tuesday	Tested and plotted potentiometer angle range, calculated angle. Programmed BLE blinky on dongle – controlled LED from my phone.
Wednesday	Wrote up potentiometer test results, researched antenna solutions, researched high speed PCB considerations
Thursday	Meeting with Frank - In-person meeting with Frank: Report structure change, progress update, vscode compiling discussion
Friday	
Saturday	Wrote up some V0.1 requirements and planning, looked into report structures. Will seek advice from James on Tuesday
Sunday	Looked into Quality Function Deployment Method of gathering performance/specifications with a House of Quality table – should be useful. Configured VS code's Intellisense for error highlighting

Use this space for any sketches, supervisor comments etc.

20/0820 - In-person meeting with Frank:
Gave Frank a progress update. He reminded me of phase shifting two potentiometers. Talked about using VSCode intelliSense for cross-compiling. Suggested that AIMRaD type report format is probably not very good for engineering – I will investigate other formats or use a format that makes sense for me.
To do on Friday (spend >= 4 hours):

- Write up v0.1 requirements
- Figure out report structure
- Have a serious attempt at coming up with concepts

Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
17.75	93	FStilwell …………………………	………………………………

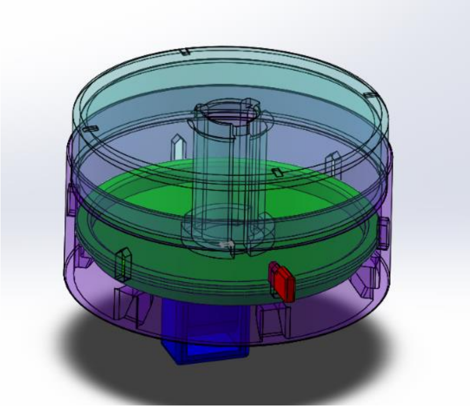
MG 7101 Engineering Development Project – Weekly Diary

Student Name: Flynn Stilwell   Supervisor: Frank Beinersdorf

Week Number 6   From: **24/08/2020**   To: **30/08/2020**

Day	**Work Done** (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	V1.0 requirements were tabulated and room was left for importance ratings to be set by the stakeholders. This is to be used in a House of Quality table for evaluating specifications. Specifications are yet to be set.
Tuesday	While configuring the cross compilation with Intellisense on VS code on 23 August, I tinkered with some library files in the compiler's folder. This led to errors with compiling previously working programs using MINGW64 and Makefile. A reinstall of the gcc arm none eabi fixed the issue. A USBD BLE UART example was then successfully uploaded and used on the nRF42 dongle. The virtual com port was opened with a PuTTY terminal and the dongle would toggle an LED when it received data from the terminal. An android phone was also connected to the dongle at the same time and data was sent using the nRF Connect android app which also toggled the LED and displayed the message on the terminal. An attempt was made at modifying the program to get the dongle to continuously display data but this was unsuccessful due to little knowledge on how the program works.
Wednesday	Drew up and analysed a concept for the mechanical interface using a magnet and steel plate, and a sensor concept using two piggy-backed potentiometers.
Thursday	3D modelled a part to hold the given 3D magnetic hall sensor. This is to conveniently hold the sensor for testing.
Friday	Started modelling a 3D printed mock-up for a mechanical interface concept
Sunday	Finished 3D modelling the mock-up in Solidworks which included 2 links and some wedges to hold steel lids for testing the magnetic field. Sent the files off to Frank to 3D print.

Haven't been able to find how to import standard c library files.
Below is a screenshot of the Mock-up 3D model.



Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
17	110	FStilwell	....

MG 7101 Engineering Development Project – Weekly Diary

Student Name: Flynn Stilwell Supervisor: Frank Beinersdorf

Week Number 7 From: **31/08/2020** To: **06/09/2020**

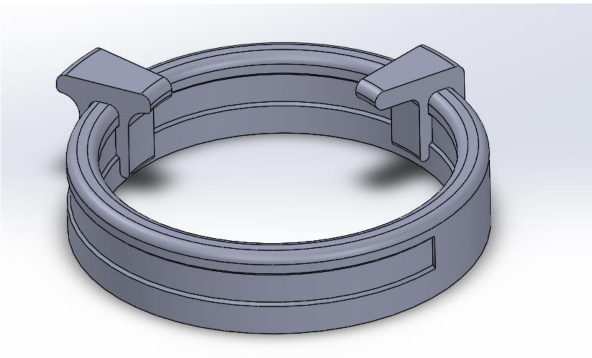
Day	Work Done (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Produced drawings for mock-up model to help explain design decisions, wrote about design decisions for the mock-up.
Tuesday	3D print of mock-up failed due to printer skipping steps. Gelasio said he might be able to get it printed by tomorrow.
Wednesday	
Thursday	Meeting with Frank, Harjeev and Chris were present. Showed Frank a new mechanical interface concept. Discussed issues with BLE UART example. Setup Nordic reference circuitry design in Eagle.
Friday	Travelled to campus to pick up first mock up 3D print, constructed longer leads for the test mag sensor. Worked on interim progress report document, edited gantt chart slightly. Applied test mag. Sensor to the mock-up.
Saturday	
Sunday	Produced and sent second mock-up STL files to Frank for 3D printing. Started assembling BOM for PCB design, mostly includes the Nordic reference circuitry parts, some from dongle circuit. Sourced most reference design parts for PCB on digi-key, wrote down manufacturer names and part numbers.

Bill of material

Board: Armature_Joint_PCB_v1.0

Version: 1.0.0

Designator	Value	Description	Assembly	Footprint Doc	Manufacturer	Manufacturer Part Numb
C1, C2, C17, C18	12pF	Capacitor, NP0, ±2% ~ 12pF ±2% 50V Ceramic Capacitor C0G, NP0 0402 (1005 Metric)		0402	Yageo	CC0402GRNPO9BN120
C3	0.8pF	Capacitor, NP0, ±10% ~0.8pF ±0.05pF 50V Ceramic Capacitor C0G, NP0 0402 (1005 Metric)		0402	Murata Electronics	GJM1555C1HR80WB01D
C4	0.5pF	Capacitor, NP0, ±10% ~ 0.5pF ±0.05pF 50V Ceramic Capacitor C0G, NP0 0402 (1005 Metric)		0402	Murata Electronics	GJM1555C1HR50WB01D
C5, C7, C8, C12	100nF	Capacitor, X7R, ±10% ~ 0.1µF ±10% 16V Ceramic Capacitor X7R 0402 (1005 Metric)		0402	Murata Electronics	GCM155R71C104KA55J
C6, C20	4.7µF	Capacitor, X7R, ±10% ~ 4.7µF ±10% 6.3V Ceramic Capacitor X7R 0603 (1608 Metric)		0603	Samsung Electro-Mechanics	CL10B475KQ8NQNC
C9	820pF	Capacitor, X7R, ±10% ~ 820pF ±10% 50V Ceramic Capacitor X7R 0402 (1005 Metric)		0402	Walsin Technology Corporation	0402B821K500CT
C10, C13, C22	N.C.	Not mounted	Not mounted	0402		
C11	100 pF	Capacitor, NP0, ±5% ~ 100pF ±5% 50V Ceramic Capacitor C0G, NP0 0402 (1005 Metric)		0402	Yageo	CC0402JRNPO9BN101
C14, C15	1.0µF	Capacitor, X7R, ±10% ~ 1µF ±10% 16V Ceramic Capacitor X7R 0603 (1608 Metric)		0603	Yageo	CC0603KRX7R7BB105
C19	4.7 µF	Capacitor, X7S, ± 10 % ~ 4.7µF ±10% 6.3V Ceramic Capacitor X7S 0603 (1608 Metric)		0603	Murata Electronics	GRJ188C70J475KE11D
D1	PRTR5V0U2X	Ultra low capacitance double rail-to-rail ESD protection diode		SOT-143B	NXP	PRTR5V0U2X
FB1, FB2	120R/0.85A	Ferrite Bead, 120 Ohm @ 100MHz, 850mA, 150 mOhm Max		0402	Taiyo Yuden	BKP0603HS121-T
L1	4.7 nH	High frequency chip inductor ±5 %		0402		
LD1	L0603G	LED, Green, 0603, 573nm, Vf=2.0V, 24mcd, - 40 to +85°C		0603	Victory Electronics	VS 25C7M5
LD2	LED RGB	RGB LED 1.60 mm x 0.80 mm SMD		LED_RGB_0603	Kingbright Company LLC	APHF1608LSEEQBDZGK(
P1	Pin Header 2x5, 1.27mm	Pin Header 2x5, 1.27mm (50mil), SMD, Keying Shroud	Not mounted	HDR_2x5-SMD-1.27mm	W-CON	3132-10MS0BK00R1
R1	4R7	Resistor ±1 %, 0.063W		0402		
R2	0R	Resistor, ±5%, 0.05W		0402		
SW1	PB SW	Tactile Switch, SPNO, SMD, 260gf, 4.2x3.2x2.5mm		PTS810	C&K Components	PTS810 SJK 250 SMTR LF
SW2	PTS840 GK SMTR LFS	Microminiature SMT Switch, Side Actuated		PTS840 Gx	C&K Components	PTS840 GK SMTR LFS
U1	nRF52840-QIAA	Multi-protocol Bluetooth Low Energy, IEEE 802.15.4, ANT and 2.4GHz proprietary system-on-chip		AQFN-73	Nordic Semiconductor	nRF52840-QIAAC0
X1	32MHz	XTAL SMD 2016, 32 MHz, Cl=8 pF, Total Tol: ±40 ppm		XTAL_2016		
X2	32.768kHz	XTAL SMD 3215, 32.768 kHz, Cl=9 pF, Total Tol: ±50 ppm		XTAL_3215		



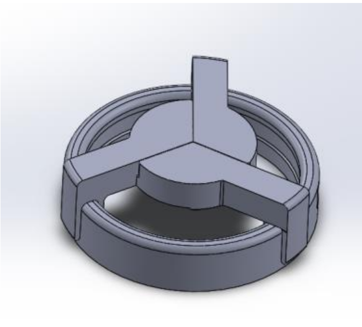
Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
17	Last week: 127	FStilwell ………………………………	………………………………

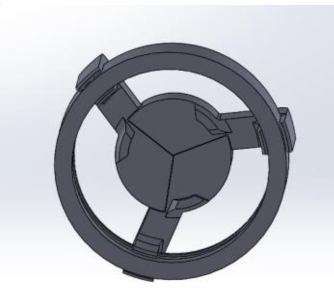
MG 7101 Engineering Development Project – Weekly Diary

Student Name: Flynn Stilwell   Supervisor: Frank Beinersdorf

Week Number   From: **07/09/2020**   To: **13/09/2020**

Day	**Work Done** (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	
Tuesday	3D modelled an iteration of the snap-fit concept. Worked on BLE UART communication again, only I was running into errors in VSCode and was unable to find a working C library file. Decided to look into segger embedded studio again as I spent most of my time trying to fix vscode problems.
Wednesday	Wrote up PCB planning considerations such as Eagle software aspects, PCBZone manufacturing constraints, supplier shipping.
Thursday	Drew a simple electronics block diagram, started researching voltage regulators
Friday	Component research on Battery holders, switches, SoC power supply considerations. Meeting with Frank and Traveler
Saturday	Research for PCB design – batteries and max current needed
Sunday	Voltage regulator research





11/09/2020 Meeting with Frank and Traveler:
Traveler mentioned PCB 'mouse bites' for adding in temporary circuits such as a USB connector. These can be easily removed once. Traveler also mentioned using small Li-Po batteries and referred me to an Aliexpress supplier https://www.aliexpress.com/store/215388. A charger IC could also be added – Traveler suggested the Microchip MCP73831/2. Having a rechargeable battery means that a power switch won't be needed – battery can run dead and recharge. Saves on switch component and wasting non-rechargeable batteries. Digi-key and PCBZone shipping time is expected to be no more than 2 weeks – MechAdept has accounts with both. Traveler offered a crash-course on C firmware programming when I'm up to that. Frank recommended printing the above clips and carriers as one part - offered trying ABS printing for extra strength if needed.

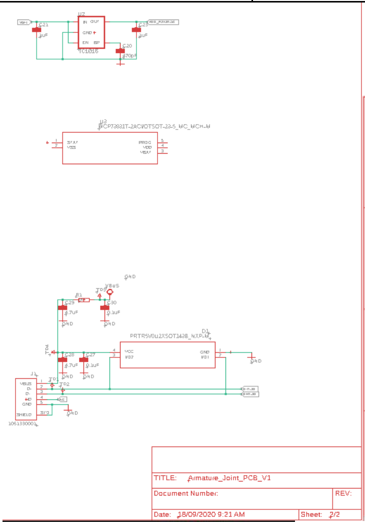
Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
19.5	146.5	FStilwell …………………………	…………………………….

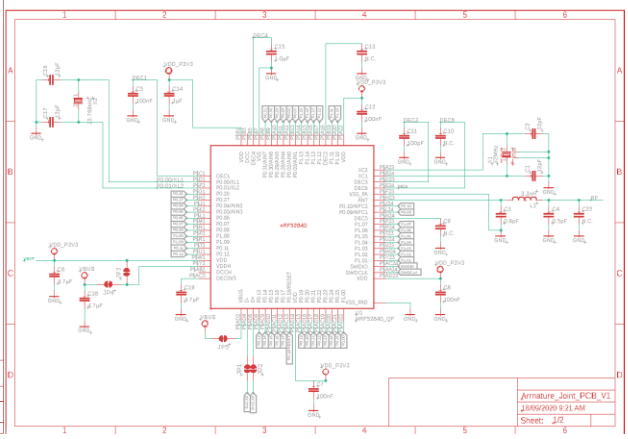
MG 7101 Engineering Development Project – Weekly Diary

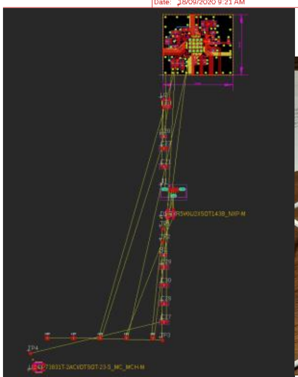
Student Name: Flynn Stilwell Supervisor: Frank Beinersdorf

Week Number From: **14/09/2020** To: **20/09/2020**

Day	Work Done (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Li-Po battery was selected, voltage regulators were researched and one was selected and sourced from digi-key, LED resistor values were determined, started charger IC selection
Tuesday	Sourced resistors from digi-key, determined the specs for the high frequency crystal oscillator, determined and sourced capacitors
Wednesday	Started eagle design schematic, sourced magnetic sensor components, contacted PCBZone for their 4 layer PCB stack up for impedance considerations
Thursday	Troubleshooted VSCode errors, experimented with 3D printed parts and magnetic sensor
Friday	Continued work on schematic, edited library components, discovered licensing issue with eagle and reached out to supervisor. Found that free version of Eagle does not accept more than 2 layers of PCB.
Saturday	
Sunday	Tested 3D printed mockup with teensy, researched alternative magnets, spoke to supervisor about educational version of Eagle







Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
24	170.5	………………………………	………………………………

MG 7101 Engineering Development Project – Weekly Diary

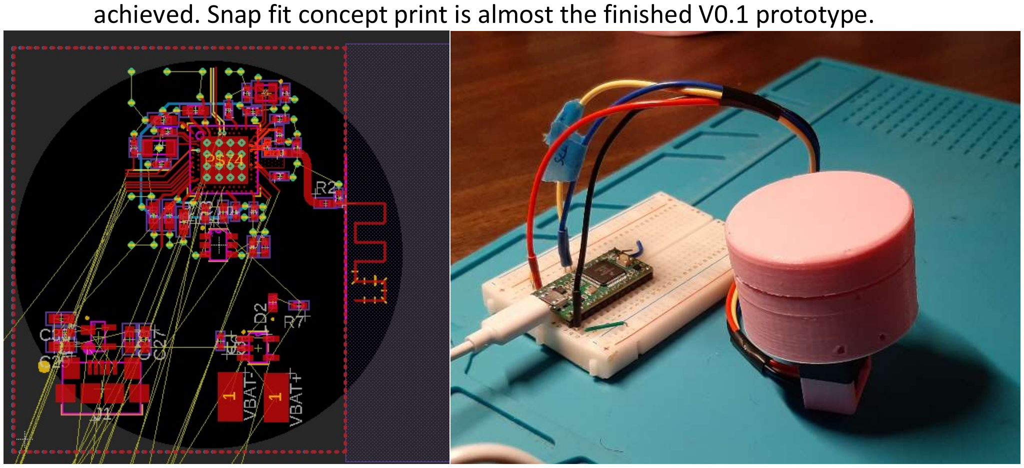
Student Name: Flynn Stilwell Supervisor: Frank Beinersdorf

Week Number From: **21/09/2020** To: **27/09/2020**

Day	Work Done (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Completed and submitted week 9 diary. Wrote up test results of magnet on plate concept mock up, set up design rules for PCBZone in eagle, started schematic design of charger IC on PCB.
Tuesday	Schematic design: Peripherals and connectors, started TLV493 sensor schematic, SWD pad connector design. Reached out to Traveler for his preference on SWD connector, he recommended 1mm pitch, 6 pin headers thru hole.
Wednesday	Assembled and glued snap-fit concept mockup, Solidworks design on next snap fit parts, exported parts to 3D print, tried offset magnetic sensor idea and was unsuccessful.
Thursday	Designed a 3D printed roller bearing for a new concept, Antenna considerations research. Had supervisor meeting with Frank, collected 3D printed parts, spoke about magnets, PCB antenna, 3D printed bearing. sent Solidworks model to Frank. Breakout pins PCB schematic designed, added solder pads for those. Sanded 3D printed parts to fit.
Friday	Sourced new SMD micro USB component, imported CAD model for which in eagle.
Saturday	Calculated RF transmission trace impedance, added meandered antenna from nRF library, started laying out microUSB, IC charger, and their passives in PCB.
Sunday	TLV493 Sensor, MicroUSB and Charger IC placement on PCB

Goals for the week:

- Get the Eagle schematic done and start on PCB layout – achieved, PCB layout started
- Finish a prototype V0.1 design in Solidworks – get ready for printing – approximately achieved. Snap fit concept print is almost the finished V0.1 prototype.



Early stage of PCB layout (left), new 3D printed snap fit concept (right)

Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
26.5	197	FStilwell	....

MG 7101 Engineering Development Project – Weekly Diary

Student Name: Flynn Stilwell   Supervisor: Frank Beinersdorf

Week Number   From: **28/09/2020**   To: **04/10/2020**

Day	Work Done (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Week 10 diary entry and submission. Ordered diametrically magnetized cylindrical magnet from AMFMagnets.co.nz. Discovered PCBZone could not provide the precision needed for the nRF52840 SoC footprint on the PCB. Had meeting with Traveler and switched focus to PCBOne, a Chinese PCB manufacturing company.
Tuesday	Researched PCBOne's design rules. Received PCBOne's standard 4 layer PCB stackup and a full spreadsheet containing their capabilities. Did some general PCB layout.
Wednesday	Implemented the charger IC's recommended layout including copper pours, started on some voltage regulator copper pours and battery terminals.
Thursday	Looked into via considerations (thru-holes to inner layers), found that nRF52840 SoC was out of stock everywhere. Had meeting with Traveler to discuss new SoC solutions, gave a progress update on the PCB. Selected the nRF52810 SoC and imported reference circuitry into the PCB.
Friday	Updated the BOM to V1.1 to account for new SoC and sourced new components on digi-key. This required fewer components. Changed eagle design rules back to PCBZone as they could manufacture the footprint of the new SoC. Worked on general PCB layout. Approximately completed power supply layout and added external pullup resistor to reset button.
Saturday	General PCB layout.
Sunday	

Use this space for any sketches, supervisor comments etc.

Neodymium Cylinder Magnet 6mm x 12mm GOLD N45 Diametrically Mag

Product Code:	C-D6H12-N45-GS
Pull Force:	1.64 Kg
Direction of Magnetisation:	Diametrically (through diameter)
Grade:	N45
Gauss:	6643
Coating:	Gold
Weight:	12 gm
Style:	Neodymium Cylinder
Dimensions:	Diameter: 6mm Height: 12mm

Charger IC recommended layout:

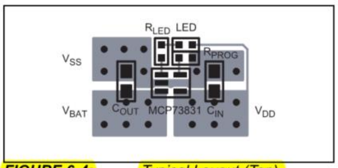


FIGURE 6-4: *Typical Layout (Top).*

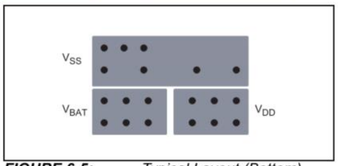
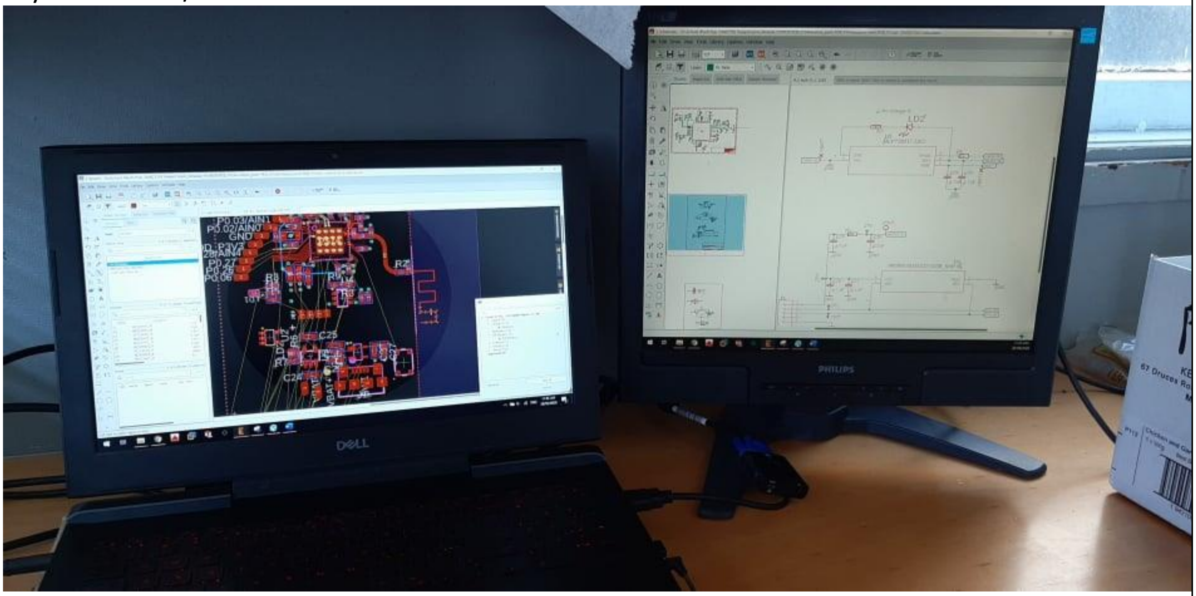
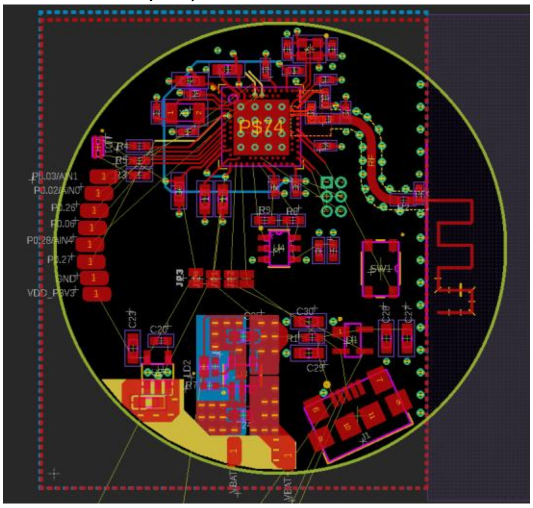


FIGURE 6-5: *Typical Layout (Bottom).*

Layout as of 29/09:



PCB as of 01/10, before the switch to new SoC:



Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
28	225	………………………………..	………………………………..

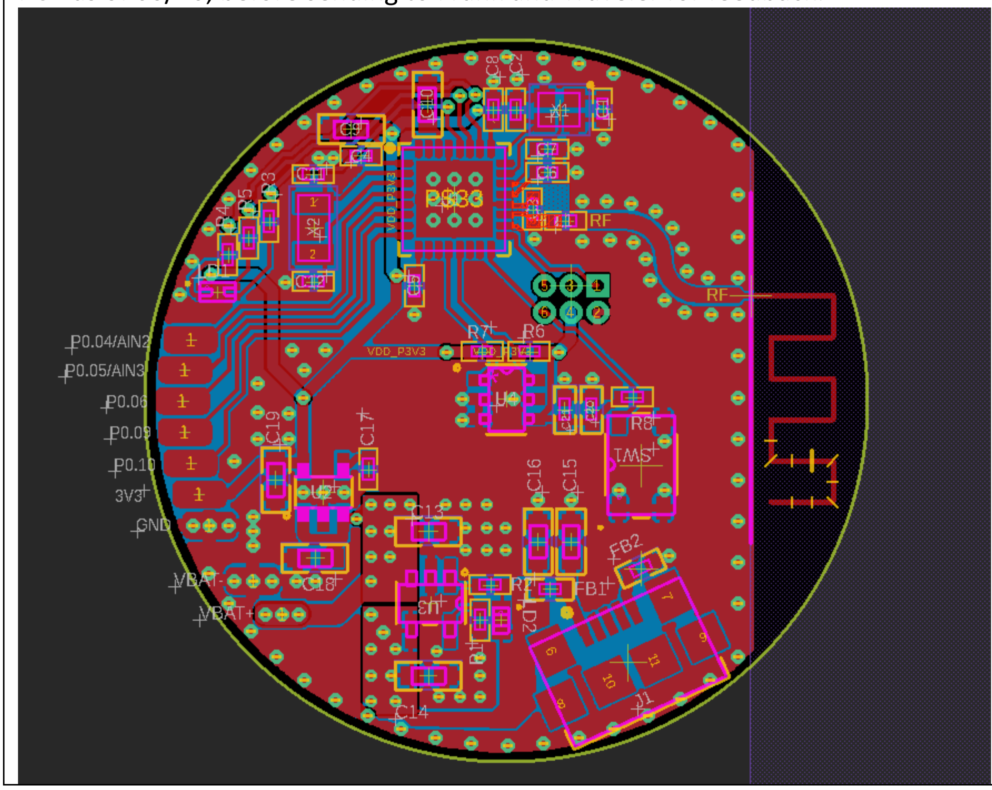
MG 7101 Engineering Development Project – Weekly Diary

Student Name: Flynn Stilwell Supervisor: Frank Beinersdorf

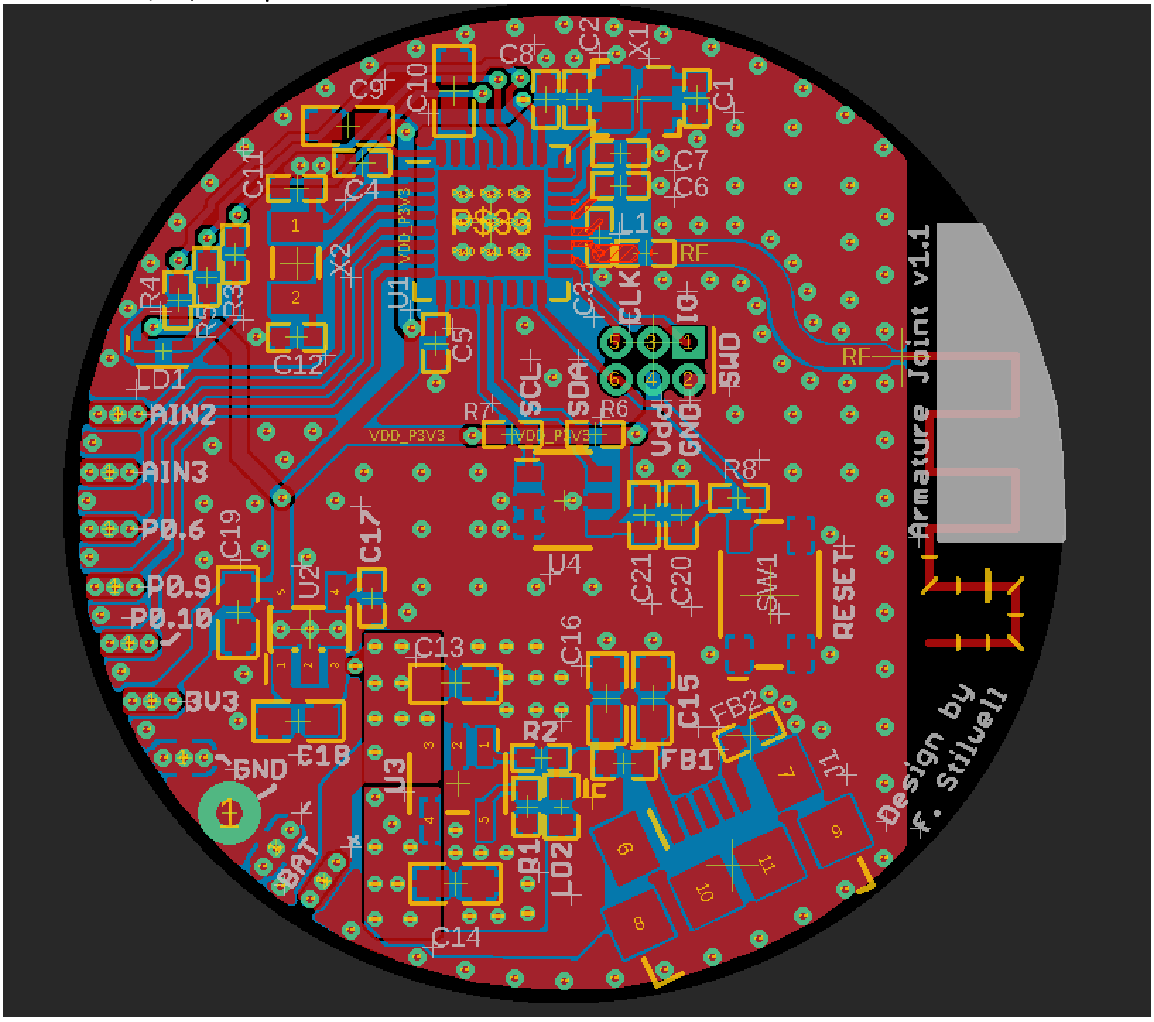
Week Number From: **05/10/2020** To: **11/10/2020**

Day	Work Done (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Started and nearly completed PCB routing. Updated packages to provide better silkscreen markings. Added outer ‘fence’ ground vias to the board.
Tuesday	Remapped SWD connector pins to allow routing with no vias. Added more ground vias to board. Altered ‘VBUS’ polygon to allow ground plane to pass under all outer fence vias. Sent the design to Frank and Traveler for feedback. Traveler recommended increasing the solder pad spacing, adding signal and power markings, and a box near board name and version on the silkscreen for marking the board if needed, paying attention to pin 1 marking and showed an example of one of his boards. He uses lines next to the pins which look clearer than the already implemented dots.
Wednesday	Checked and exported the digi-key shopping cart and sent it to Traveler to be ordered. Shifted solder pads for clearance while soldering as recommended by Traveler, updated RGB LED and USB connector footprint silkscreen markings for polarity and clearance with pads/vias.
Thursday	Added and positioned component names, edited pin 1 markings as recommended, added board name and box, added extra ground vias, labelled I2C signals, attempted adding MechAdept logo to eagle. Exported the gerber files to send to Traveler. Had a meeting with Traveler to fix issues with gerbers (solder mask and exposed vias) and ordered 6 boards. A grounded, plated through-hole was added as recommended in the bottom left for a test point to clip an oscilloscope lead to.
Friday	

PCB as of 06/10, before sending to Frank and Traveler for feedback:



PCB as of 07/10, as exported and sent to manufacturer:



Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
14.5	239.5	………………………………	………………………………

MG 7101 Engineering Development Project – Weekly Diary

Student Name: Flynn Stilwell   Supervisor: Frank Beinersdorf

Week Number: 11   From:**12/10/2020**   To: **18/10/2020**

Day	**Work Done** (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Looked at 'getting started' guides for the nRF52 dongle programming.
Tuesday	Reinstalled nRF52v17 SDK for fresh start, found guide for developing for nRF52810, looked at TWI documentation. Attempted uploading example from youtube video – unsuccessful
Wednesday	Working through Bluetooth Low Energy examples for the dongle, trying to understand them. Modelling V1.0 structure link 1 design, almost finished link 2.
Thursday	Had a supervisor meeting with Frank to work through BLE blinky and UART examples for the dongle. Exported 3D circuitboard file from eagle to Fusion360, imported the model into Solidworks, modelled the cylindrical magnet to be used.
Friday	Attempted uploading BLE UART example again, bricked the dongle by hitting 'reset' on the programming app which erased the USB bootloader. Needs a debugger to be programmed again. Started PCB stops and peg design in the V1.0 mechanical structure model test. Looked for USB 3D package in Eagle. Researched debugger connection pinout for Travelers's Atmel-ICE debugger.
Saturday	Made the decision to wait to use Frank's J-link debugger, set to arrive in 5 days. Sourcing debugger cables – found that the 0.05" pitch header pins were rear in NZ, SWD cables were rear but a couple were accessible from NiceGear.co.nz. Modelled pegs for V1.0 structure test print, prepared the test print files to be sent to Frank.
Sunday	

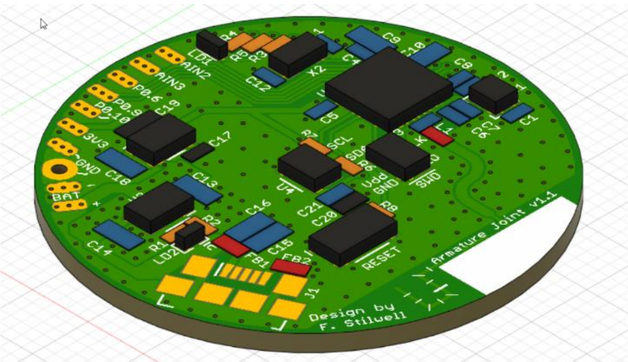


Figure 1: PCB model in Fusion360

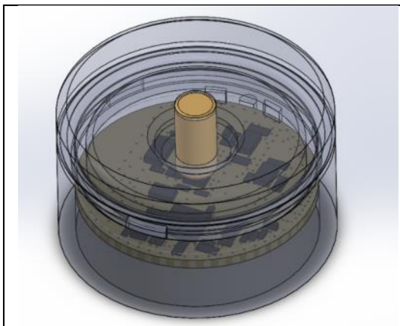


Figure 2: Preview of test print of structure v1.0

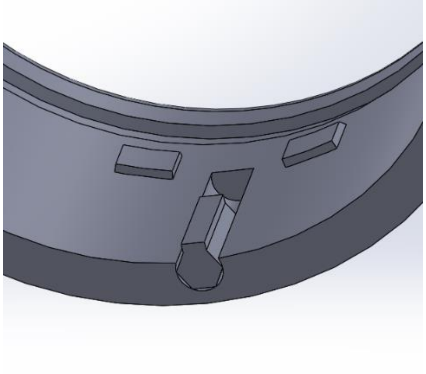


Figure 3: PCB stops and peg design - peg is to hold PCB in place against the stops

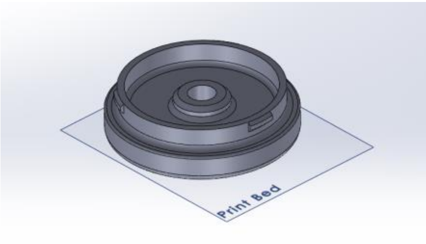


Figure 4: Link 2 test print

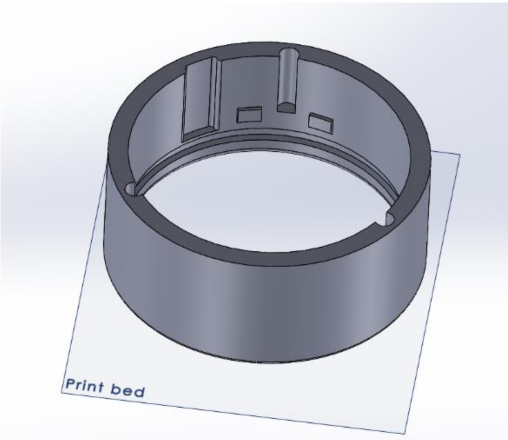


Figure 5: Link 1 test print. Includes a notch to test clearance of battery wires looping under the pcb

Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
21	260.5	JtStilwell …………………………….	……………………………..

MG 7101 Engineering Development Project – Weekly Diary

Student Name: Flynn Stilwell   Supervisor: Frank Beinersdorf

Week Number 12   From: **19/10/2020**   To: **25/10/2020**

Day	**Work Done** (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Collected 3D prints from Frank, reading through BLE examples
Tuesday	Watched videos on PCB assembly process for better understanding before my own assembly. Analyzing V1.0 structure test print, found pegs method was not going to work with 3D printed parts. Made changes for V1.1 test print. Read and signed confidentiality agreement with MechAdept. Watched another PCB assembly video. Prepared for PCB assembly at MechAdept.
Wednesday	Looked into antenna tuning considerations, checked BOM and datasheets. Looked into custom nRF board programming. Assembled 6 V1.1 PCB's at MechAdept. Examined and fixed obvious reflow errors on the boards, electrically tested one of them.
Thursday	
Friday	
Saturday	Comparing BLE central and Peripheral example programs. Looked through UART examples. Installed Meld software to compare code between examples.
Sunday	

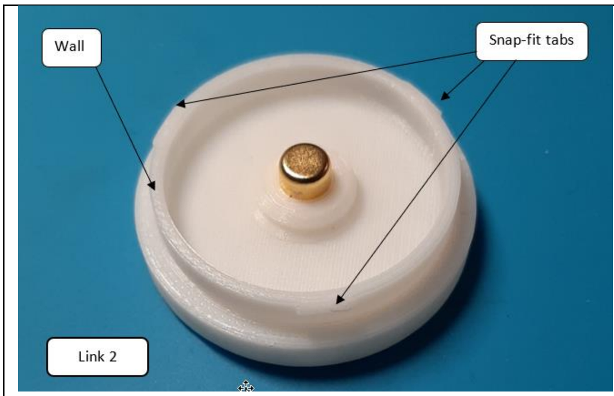


Figure 1: V1.0 Structure Link 2 with magnet installed

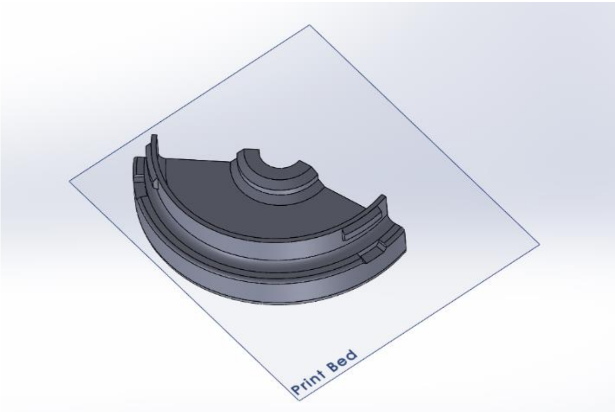


Figure 2: V1.1 Structure test print

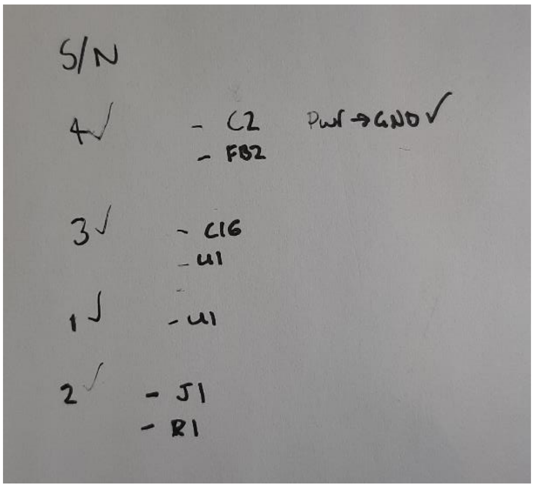


Figure 3: Brief Notes on components to fix on the boards

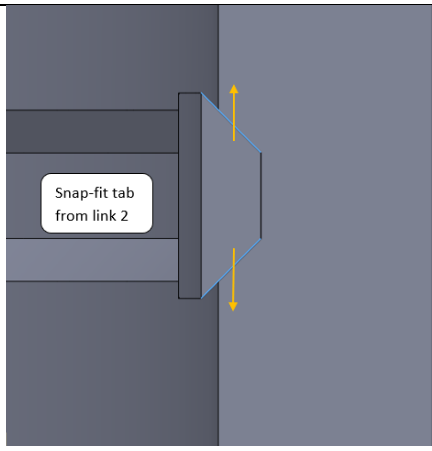


Figure 4: V1.0 Structure modification for greater friction

Figure 5: All 6 V1.1 Boards after reflow

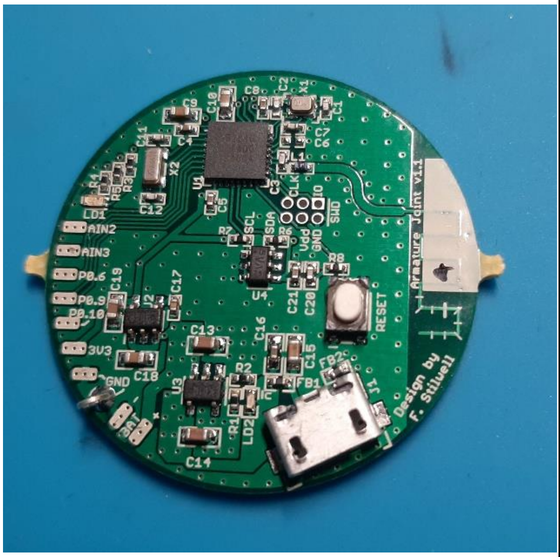


Figure 6: Complete V1.1 Armature Joint PCB, serial number 4

Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
20	280.5	F Stilwell ………………………………	………………………………

MG 7101 Engineering Development Project – Weekly Diary

Student Name: Flynn Stilwell   Supervisor: Frank Beinersdorf

Week Number 13   From: **26/10/2020**   To: **01/11/2020**

Day	**Work Done** (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	Week 12 entries diary entries completed. 3D modelled PCB USB connector, enclosure USB cutout according to standard, tabs and stops for holding the PCB and sent mechanical structure v1.1 test print to Frank.
Tuesday	Reading through SDK examples, modified BLE Blink application to continuously send notifications to phone. Estimated times for finishing project, emailed coordinator with concerns. Collected J-link debugger and mechanical structure v1.1 test print from Frank. Sanded PCB tabs to fit into test print, tested battery fit, prepared power test sheets for the PCBs.
Wednesday	Tested power/ground resistances on all boards. Found Error in design of button. Made up SWD connector cable for programming the boards, removed button from PCB number 6.
Thursday	Installed nRF52 v14.2 SDK and softdevice s132 v5.1 from Nordic semiconductor, started custom BLE service example tutorial. Followed a tutorial on a short blink program, successfully connected the J-link to the board.
Friday	Started and finished first draft of product synopsis, sent to coordinator.
Saturday	Modelled link 1 cap, adjusted magnet position, exported model and sent to Frank for virtual robot illustration. Exported files to print. Sent print models to Frank. While attempting to upload a simple program to the board, a memory placement overflow occurred. Tried to solve error. Spoke to Traveler and made progress on finding the cause of the error. Segger embedded studio was not producing a sufficient linker script file.
Sunday	Finished and submitted synopsis.

To Do:

- ~~1-page synopsis of project – due by end week~~
- A1 size poster for showcase evening
- ~~Get structure v1.1 test print to Frank by end of day 26/10~~
- Laser cut a custom protractor for joint position measurement
- ~~Program dongle to stream Bluetooth notifications~~
- ~~Test serial number 4 board further – attach battery, program the board~~
- ~~Acquire debugger and cable from Frank~~

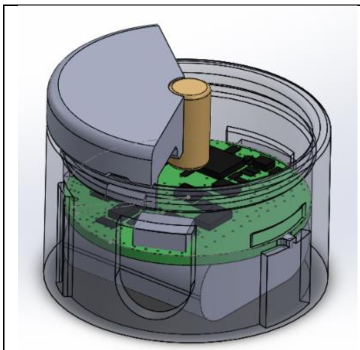


Figure 1: Structure v1.1 test print sent to Frank

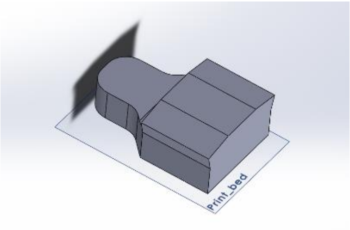


Figure 2: New design on pegs to hold PCB in place (for the above test design)

Figure 3: Test print PCB and battery fit check

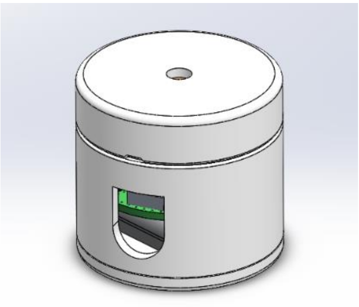


Figure 4: Full test prototype model

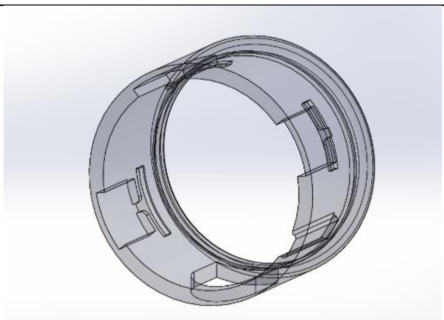


Figure 5: Structure v1.1 test print model showing PCB tab cut-outs and stops

```c
int main(void)
{
    // Initialize.
    log_init();
    leds_init();
    timers_init();
    buttons_init();
    power_management_init();
    ble_stack_init();
    gap_params_init();
    gatt_init();
    services_init();
    advertising_init();
    conn_params_init();

    // Start execution.
    NRF_LOG_INFO("Blinky example started.");
    advertising_start();

    // Enter main loop.
    for (;;)
    {
        uint8_t testing = 10;
        ble_lbs_on_button_change(m_conn_handle, &m_lbs, testing);
        idle_state_handle();
        nrf_delay_ms(100);

        testing = 8;
        ble_lbs_on_button_change(m_conn_handle, &m_lbs, testing);
        idle_state_handle();
        nrf_delay_ms(100);
    }
}
```

Figure 6: Modified BLE example for streaming a notification every 100ms

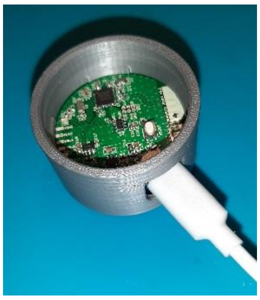


Figure 7: Test print PCB and USB fit check

Figure 8: Assembled full test prototype

Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
40.75	321.25	JStilwell …………………………….	…………………………….

MG 7101 Engineering Development Project – Weekly Diary

Student Name: Flynn Stilwell Supervisor: Frank Beinersdorf

Week Number 14 From: **02/11/2020** To: **08/11/2020**

Day	Work Done (If you have not worked on the Project on any particular day, just leave the space blank)
Monday	3D modelled and laser cut a custom protractor to fit over a testing version of the joint. Modelled the testing version. Encountered 'unplaced sections' build errors. Tried installing a different version of Segger embedded studio for resolving issue. These versions were not supported (licensing methods outdated).
Tuesday	Encountered and resolved compiling errors. Used different example code and the program built successfully. Successfully programmed the board but there was no activity. While debugging, the program was being forced into a hardfault mode. Could only escape this type of error by resetting the board. Was not sure how to debug very well.
Wednesday	Researched how to debug and installed Ozone (debugging application). Found that the memory settings were not changed. After changing these to the correct values, the program worked, blinking an LED. However, the LED only blinked when I had placed the voltmeter probe onto its resistor. This highlighted issue of absent solder on the resistor pads. Resolved the 'unplaced sections' error. Some memory sections were named incorrectly which resulted in the error. Added solder to the LED's resistors and successfully blinked all LED colours. Modified and uploaded BLE blink application successfully. Tested connection range of >7m.
Thursday	Exporting v1.4 3D prints and sent to Frank. Testing on board Serial Number 1: Tested power, found button was wired incorrectly in two areas – one was already known. Modified the button connections by sliding electrical tape underneath the button pads. Soldered battery on and tested voltages. Uploaded BLE UART example to the board and attempted modifying it. Started working on presentation poster.
Friday	Rendered image of model in Solidworks. Poster draft mostly finished.
Saturday	More attempts at editing BLE UART example. Learning about software development with nRF52 sofdevice API. Found that the level of knowledge required for the software was much beyond that which has been taught in the programme. Raised these concerns with Frank.
Sunday	Finished poster draft and sent to Frank and James.

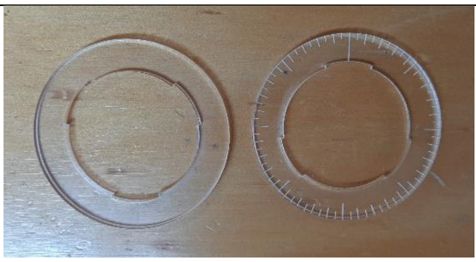


Figure 1 Custom protractor for testing joint angle against calculated angle

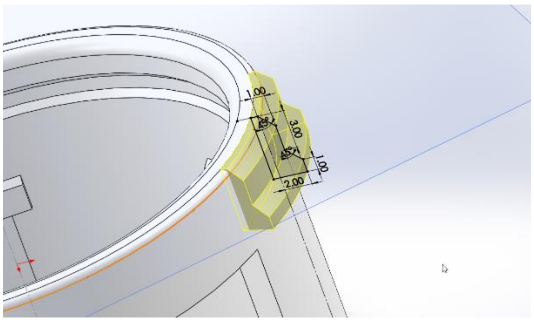


Figure 2: Testing version modification to link 1. This is to hold the custom protractor in place

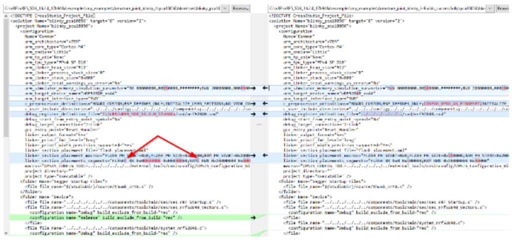


Figure 3: Found the cause of 'Unplaced sections' error by comparing a project that did work with one that wasn't with Meld, a file comparing software

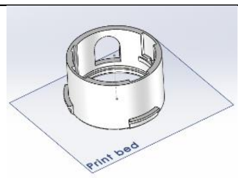


Figure 4: link 1 test version as exported

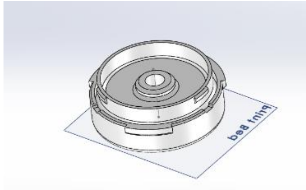


Figure 5: Link 2 test version as exported

Figure 6: Board serial number 1 with battery soldered on

Number of Project Hours This week	Cumulative Project Hours	Student's Signature	Supervisor's Signature
39	360.25	FStilwell …………………………………	…………………………………

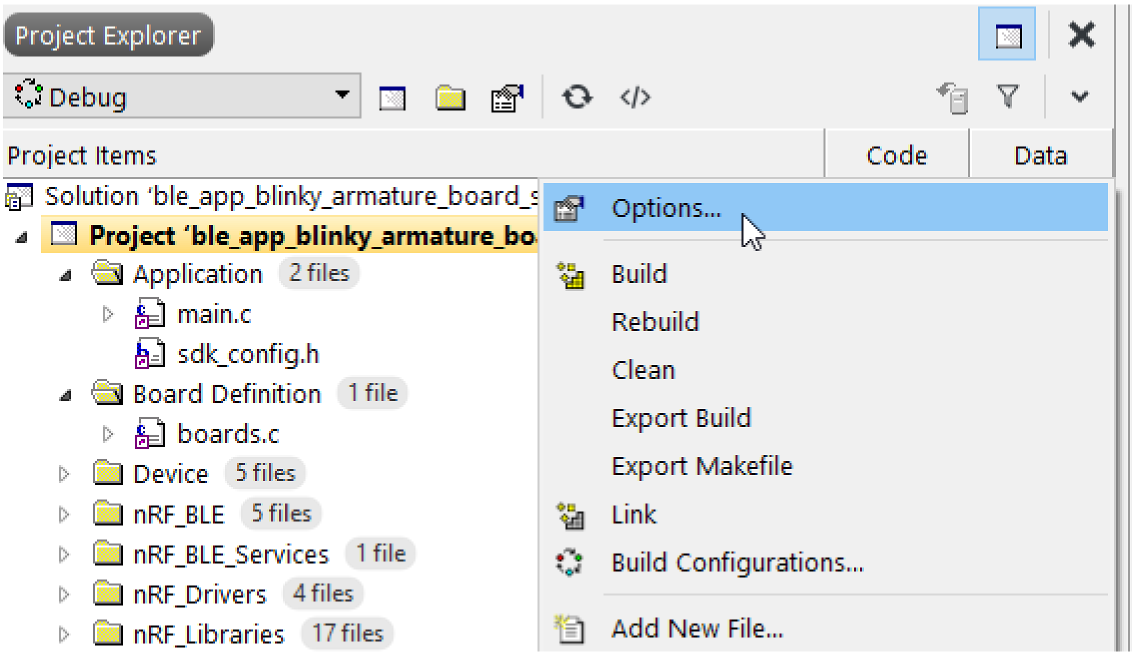
Appendix H – Guide to Configuring SDK14.2 Examples for the Armature Board

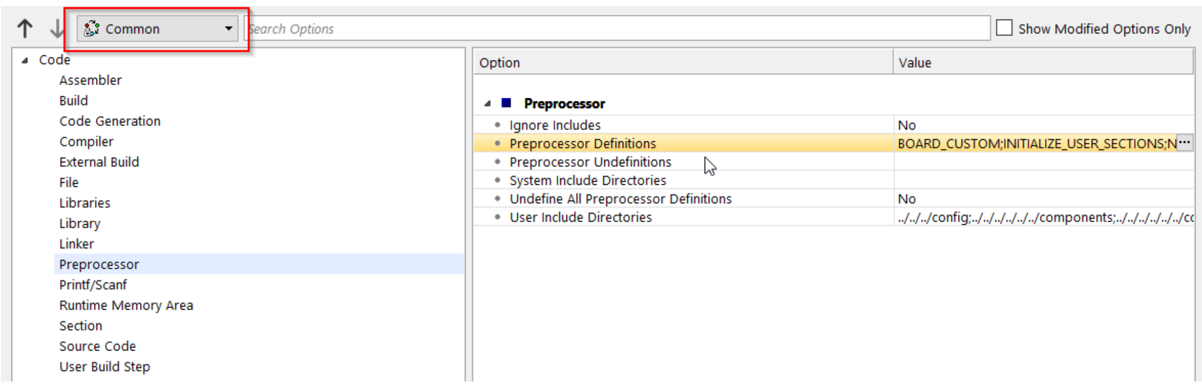
Steps to configure SDK examples for Armature Board

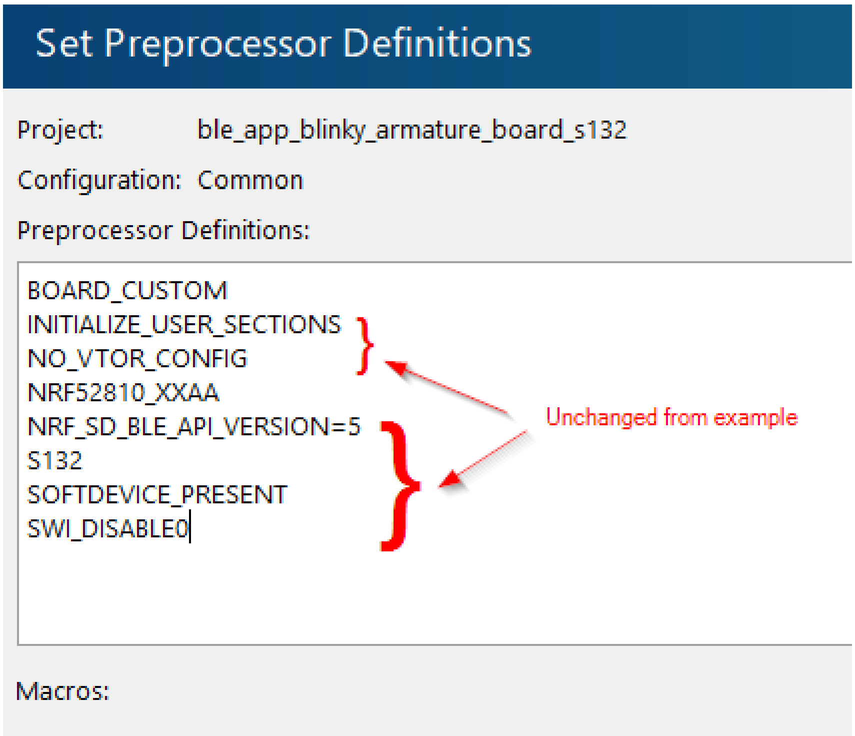
These steps refer to Segger Embedded Studio.

1. Example selection
   After an example has been copied into a user directory in the /examples folder of the SDK, the "pca10040" folder
   should be duplicated and renamed. Use the .emproject file in the /s132/ses folder.

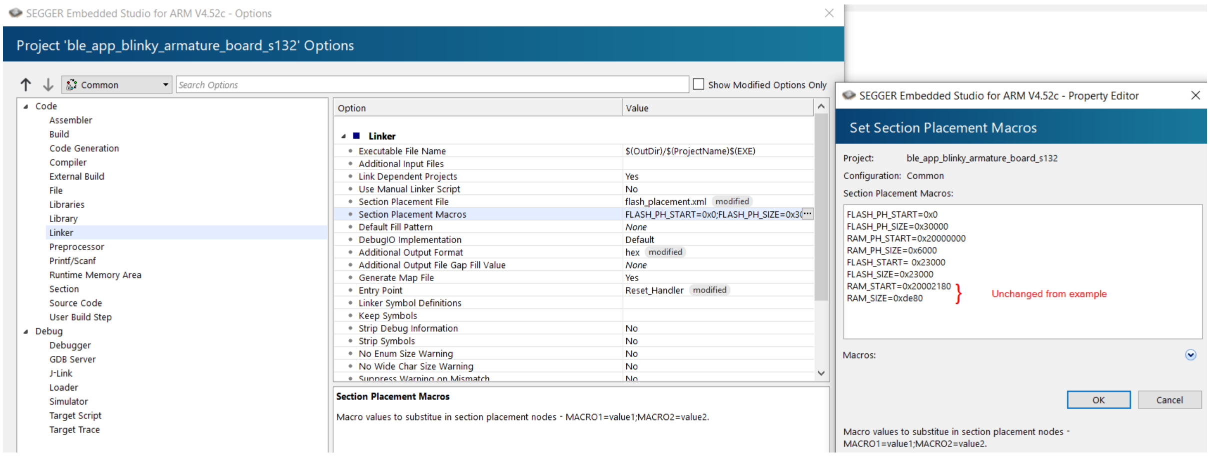
2. Preprocessor definitions
   Follow clicks below for to get to the preprocessor definitions



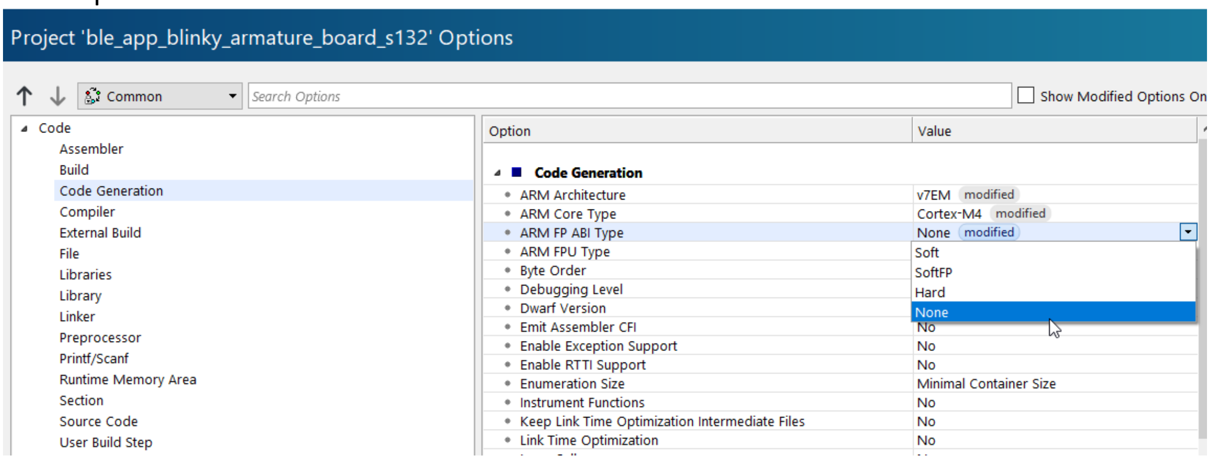




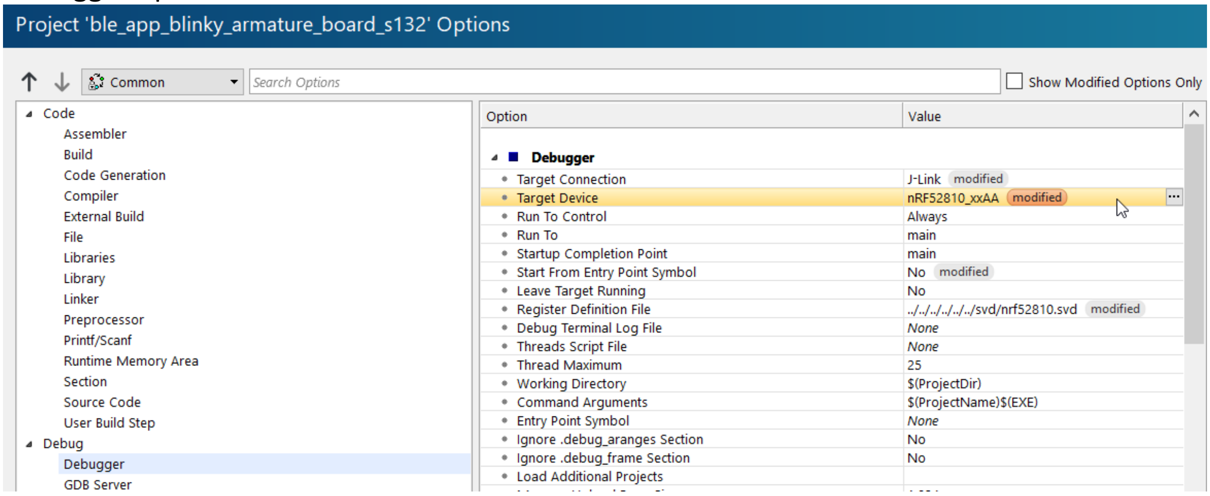
3. Linker options
   Section Placement macros



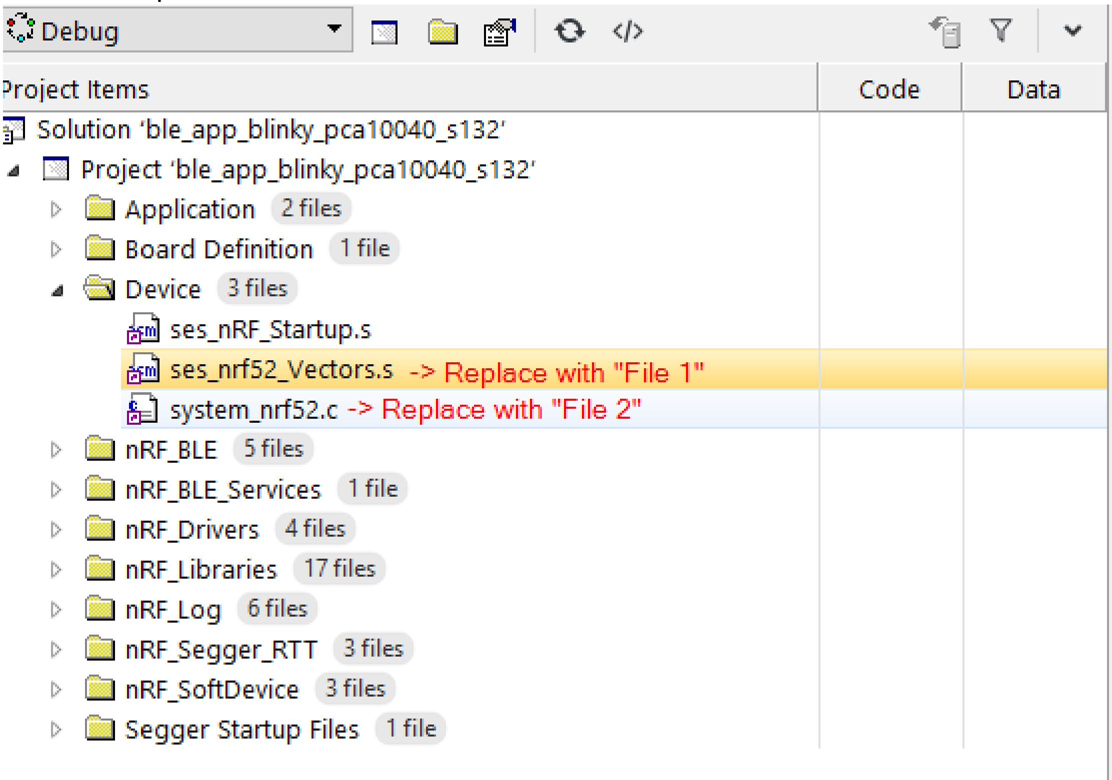
4. Build options



5. Debugger options



6. Device Setup

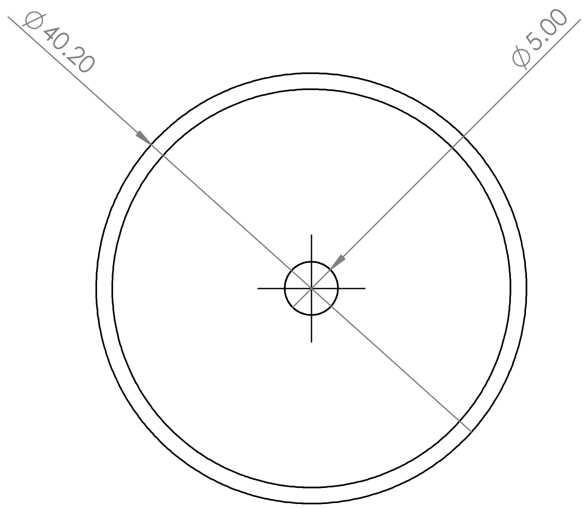


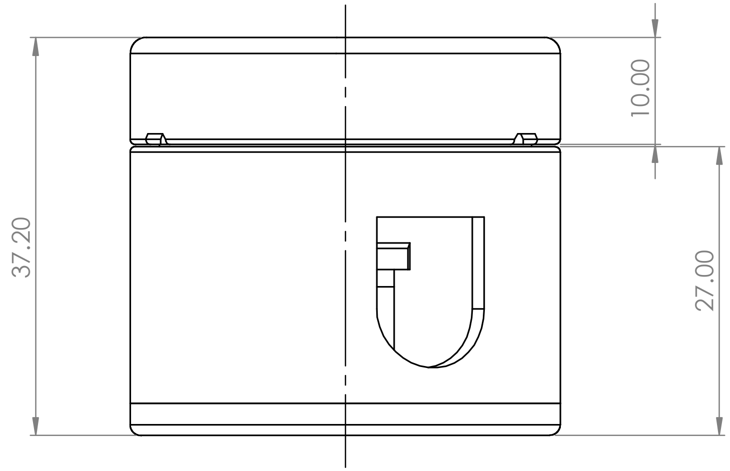
File 1: (SDK)/components/toolchain/ses/ses_nrf52810_Vectors.s

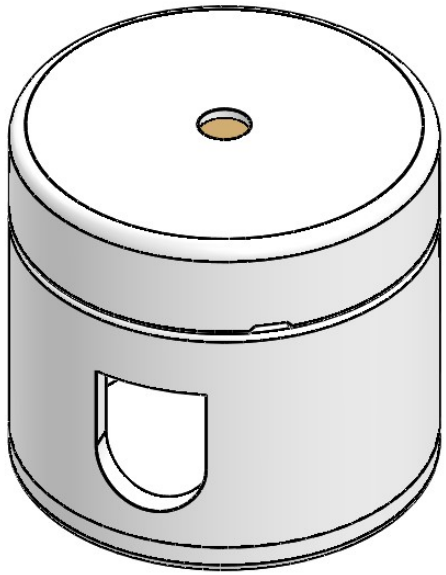
File 2: (SDK)/components/toolchain/system_nrf52810.c

Appendix I – V1.0 Prototype Assembly Drawing

See the next page for the prototype assembly drawing







WelTec	TITLE: ARMATURE JOINT V1.0 PROTOTYPE ASSEMBLY DRAWING	SCALE: 2:1	DRAWN: F. STILWELL
		DRG #: 1	DATE: 27/11/2020