Micro-mouse Project Brief

EEE3088F 2024

General Instructions

This document contains the project description for both the power and sensing subsystems. Please read this document carefully and make sure that you understand all the project requirements. There are four assessments linked to the project:

1. The *Gerber Files* submission (Friday, 24 March 2024 at 23:55 on Amathuba)
2. The *Interim Design Report* (Sunday, **21** April 2024 at 23:55 on Gradescope AND Amathuba)
3. The *Lab demonstration* (Wednesday, ~~8~~ **10** May 2024 in ~~White Lab~~ venue to be confirmed)
4. The *Final Report* (Friday, 17 May 2024 at 23:55 on Gradescope AND Amathuba)

A special note is that the assessment is **individual.** You are each expected to submit the Gerber files, reports and attend the lab demonstration for your respective subsystems.

Group work

The project is setup to be a paired group project with completely individual assessments and marking structures. In your pairs, you already decided which subsystem each person is responsible for. This selection will remain the same for the project and all submissions for the remainder of the semester.

Working in groups is a necessary skill you need to develop, and we encourage you to collaborate with other students. However, it is often the case that some team members do not fully participate in the course and often rely on others to do the work for them. To try and deter this behaviour, we have changed the assessment strategy this year. Although you are working in pairs, each **individual student is required to make individual submissions and will receive an individual mark.** To clarify, if you are responsible for the sensing subsystem, you will complete and submit all the sensor subsystem assignments.

We encourage you to communicate and collaborate with your teammate throughout the course, it will take you farther than if you go alone.

General notes

1. One of the core skills of an engineer is the ability to ask more questions and communicate with the client to ensure that you deliver the correct product. If you do not understand something, do some research and if you still do not understand then ask – suffering in silence is a mistake.
2. This is a live document, make sure to regularly check for updates and to download the latest versions so that you do not miss out on key changes.
3. During the lab sessions tutors will be available to help you. However, you will need to do some work outside of these hours if you wish to make good use of the time with tutors.
4. Again, this is all an **individual** assessment. Each person will need to **submit each assignment** for their subsystem.
5. ***All reports must be written in LaTeX.***
6. ***You will need to use GitHub for version control and be required to upload the link in your report.***

Project Overview

The project for 2024 is building some subassemblies for your very own (simplified) micro-mouse.

What is a micro-mouse?

In short, it is a maze-solving robot. Watching the Veritasium video will give you a clearer understanding of what a micro-mouse is. It is important to note that your micro-mouse will likely (almost certainly) be much slower than the ones featured in the video. Nonetheless, let the video serve as inspiration for your project.

What is your project?

In this course you will be focusing on the design of the micro-mouse's hardware (with some minor software components). The complexity of this project is in meeting the requirements while still adhering to the STRICT budget. The project has been compartmentalised into four modules: the processor, motherboard, sensing and power. The processor and motherboard modules have been designed and will be given to you. **Your project involves designing and manufacturing the sensor and power modules.** More information about each module is provided in the table below.

Module	Description
The motherboard	You are given the motherboard which is responsible for connecting all the PCBs together. It is the base board that all other modules will slot onto.
The processor	You are given the processor board which has a **STM32L476**, a significant upgrade in performance compared to the 2nd year STM32F051. It is a 100-pin package and has 78 output pins that are available to use. Most of these have already been dictated by the required interconnections between supplementary modules.
The power	This module will be responsible for powering the entire system. **You will need to design and manufacture this module** to fit the requirements detailed in this document. The basic idea is: ▪ It needs to run the motors and charge a battery. ▪ It will need to fit onto the pin headers on the motherboard. ▪ It will need to be an appropriate size for the robot.
The sensor	This module will be responsible for detecting/sensing objects. **You will need to design and manufacture this module** to fit the requirements detailed in this document. The basic idea is: ▪ It needs to detect objects. ▪ It will need to fit onto the pin headers on the motherboard. ▪ It will need to be an appropriate size for the robot.

To be successful, you will need to understand how your component fits into the greater picture and what you would need to do on your module to meet the requirements.

Project Description

Overview

As with the breadboard assignment, you will continue to work in your groups and on your chosen subsystems. To be clear, in your groups, one person will be responsible for designing the power subsystem and the other will be responsible for designing the sensing subsystem. Although you are working in a group, you will be required to complete all assessments **individually** and will receive an **individual** mark.

The rest of this document will outline the requirements and specifications of your subsystems as well as the assignments that you will need to hand in.

Subsystem: The Motherboard

Interface	Description
Processor board	Two 2x19 (2.54mm pin pitch) pin headers (*pinouts listed in the processor section*).
Motors	Two 2x1 (2.54mm pin pitch) pin headers
Sensor board	2x14 (2.54mm pin pitch) pin headers (*pinouts listed in the sensor section*).
Power board	2x8 (2.54mm pin pitch) pin headers (*pinouts listed in the power section*).

Table 1 Motherboard interfacing connections

The image below features a pin view of the Motherboard PCB. This is the exact layout and shape of the motherboard. You will design your sensor and power PCBs to fit onto the sensor and power connections.

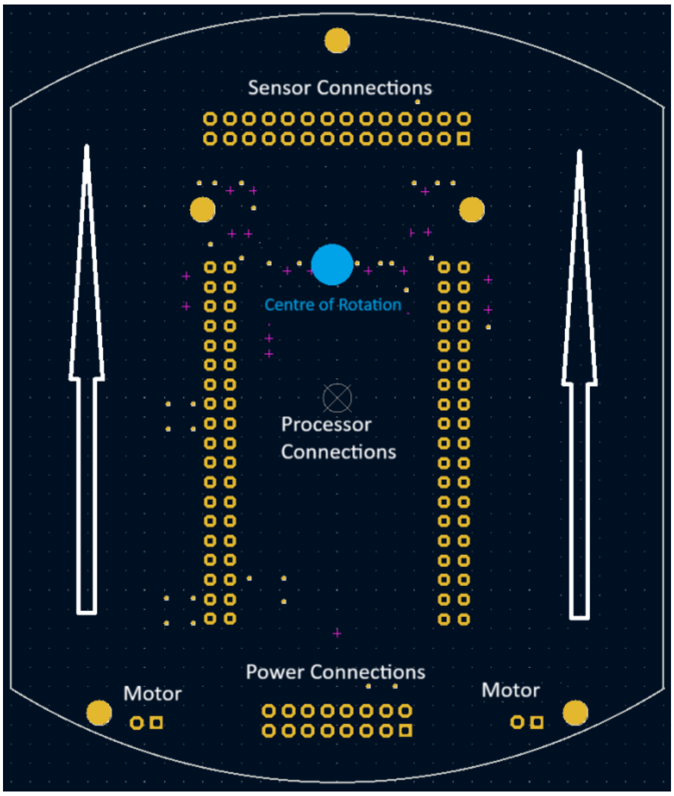


Figure 1 Pin-view of the MM Motherboard PCB.

Note: pay attention to the position of the center of rotation.

Subsystem: The Processor

As discussed before, you will be receiving a processor board with the STM32L476 microcontroller on it. The images below depict the 3D PCB render and the schematic of the **processor board**.

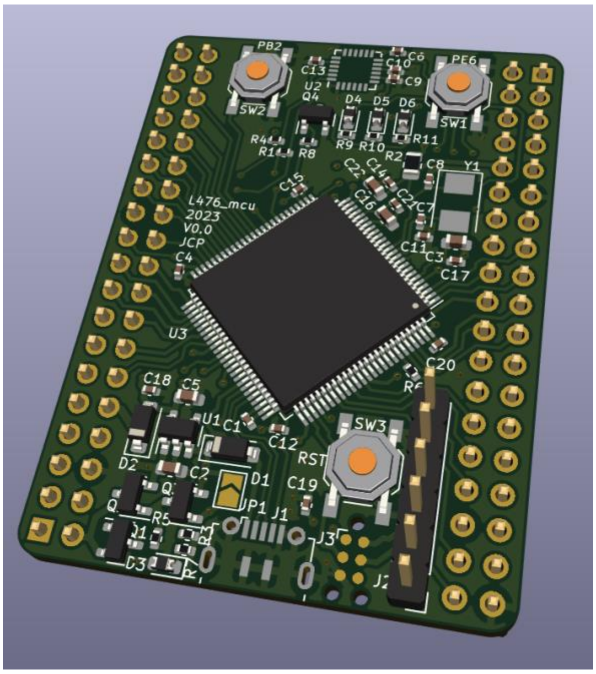


Figure 2 3D Render of the MM Processor PCB

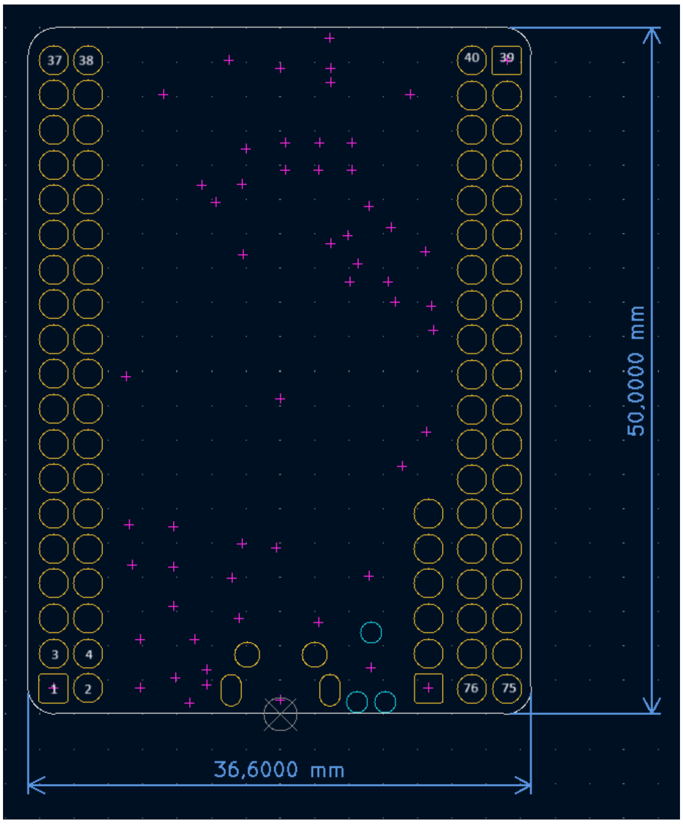


Figure 3 Pin-view of the MM Processor PCB

The following table highlights the pinouts of the processor board:

Pin Number	Associated to	Useful/Function
1	PA8	
2	VBAT	
3	PC9	
4	PC8	
5	PC7	
6	GND	
7	PC6	
8	PD15	
9	PD14	
10	5V	
11	PD13	
12	PD12	
13	PD11	
14	PD10	
15	PD9	
16	PD8	
17	PB15	
18	PB14	
19	PB13	
20	PB12	
21	PB11	
22	PB10	
23	GND	
24	PE15	
25	PE14	
26	PE13	
27	PE12	
28	PE11	
29	PE10	
30	PE9	
31	3V3	TLV740P
32	PE8	
33	PB1	
34	PE7	
35	PB0	
36	PC5	
37	PC4	
38	PA7	
39	PA5	
40	PA6	
41	PA3	
42	PA4	
43	PA1	
44	PA2	
45	PC3	
46	PA0	
47	PC2	
48	3V3_ADC	
49	GND	
50	PC1	
51	PE1	
52	PC0	
53	PE4	
54	PE5	
55	PE2	
56	PE3	
57	PE0	
58	NRST	
59	PB8	
60	PB9	
61	PB7	
62	PB6	
63	PD7	
64	PB4	
65	PD5	
66	PD6	
67	PD3	
68	PD4	
69	PC11	
70	PD2	
71	PC10	
72	GND	
73	PA15	
74	PD1	
75	GND	
76	PD0	
USB Micro Connector	Standard USB Connector	
Prog_3V3	Target Voltage sense	
Prog_ SWCLK	SWCLK	
Prog_ SWDIO	SWDIO	
Prog_ NRST	NRST	
Prog_ GND	GND	
Prog_ X	NC	

Table 2 Electrical pinout of the MM Processor PCB

Subsystem: Power

Your module needs:

- To operate 2 motors with the pins available to you (*listed in Table 3*). You will need to control 2x motors which could each draw 200mA at the highest voltage of a 1S1P battery (the battery is further specified in the battery section).
- Needs to provide an analog connection that provides information on the battery's voltage for the processor to sense battery state of charge (SoC).
- Needs to charge the battery from the 5V input pin (*listed in Table 3*). **Hint: What happens when 5V removed?**
- Needs an ON/OFF switch. **OFF state: battery draw <500uA. ON state: your robot peak current.**

Additionally, the following connectors need to be included:

1. **A JST PH 2mm pin pitch** connector for the battery. The battery will either be tucked away between the motherboard PCB and the processor PCB or secured to the bottom of the motherboard.
2. A **2x8 (2.54mm pin pitch) pin header** as shown in the image below:

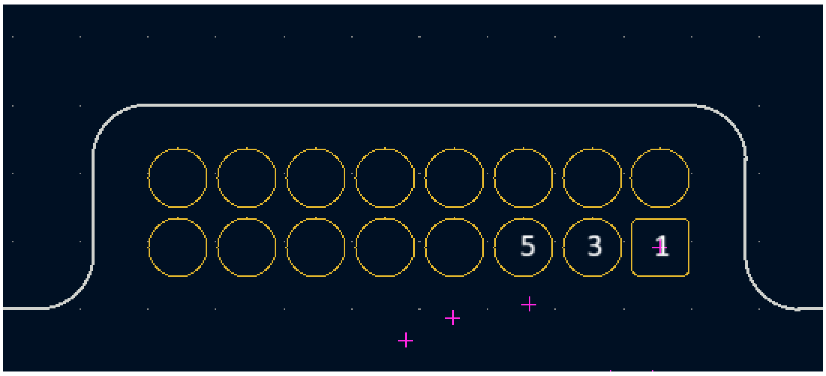


Figure 4 MM Power PCB pinouts that plug onto the MM Motherboard PCB.

Note: the shape and size of the PCB board is an important consideration:

1. The Connector does NEED to be proud of the resulting board (i.e. needs to have a tab) as the motors and connections would collide otherwise. **The height of the tab can be 18mm or greater, the width can be no greater than 35mm and the connector should be centered in this tab.**
2. Additionally, you want to minimize the maximum distance of your robot from the center of rotation (indicated on the MM motherboard PCB). Essentially this means that you want to make your PCB as small as possible.

The table below surmises the connections:

Connector	Pin Number	Associated to	Useful
	1	Motor2_A	
	2	PC6-9	PWM1
	3	Motor2_B	
	4	PC6-9	PWM2
	5	GND	
	6	GND	

2x8 pin (2.54mm pin pitch) header	7	BATT	
	8	BATT	
	9	PD7	RESV
	10	BATT_ADC	
	11	5V	
	12	5V	
	13	Motor1_A	
	14	PC6-9	PWM3
	15	Motor1_B	
	16	PC6-9	PWM4
JST PH 2mm pin pitch	Battery Connection		

Table 3 Electrical pinout of the MM Power PCB

Subsystem: Sensing

The sensor PCB is effectively the "eyes" of your robot. Your subsystem will provide information to the processor to determine whether there is an obstruction in the way of your MM. Your submodule needs to:

- Detect whether there is an obstacle in front of the robot (probably on the sides too).
- Have switching means to save power when not in operation.

Additionally, you need to:

- Design for reliability such that you can prove your system works.
- Consider how much power and current your sensing board is using/drawing so that you do not drain the battery before the maze is solved.
- Write some code to interface your sensor with the rest of the system and prove it senses the wall. **The processor board has 3 LEDs (so you do not need to add LEDs for this purpose on your sensing board)**. Use 1 to indicate a wall is sensed on the right, 1 to indicate that there is a wall on the left and 1 to indicate a wall in front of the robot. If no wall is sensed, then no LEDs will be on.

The following image is a template of the board shape, but the arc and design are not necessarily an optimal solution.

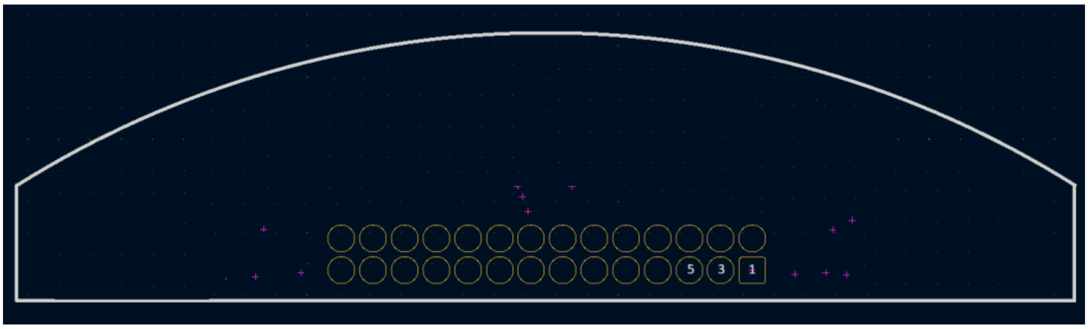


Figure 5 MM Sensing pinouts that plug in the MM Motherboard PCB

It will need to fit onto the 2x14 connection standard and be an appropriate size for the robot.

Pin Number	Associated to	Useful
1	RESV3	PC2
2	3V3	
3	GND	
4	BATT	
5	PA3	Analog8
6	RESV2	PE1
7	PA4	Analog9
8	PE15	PWM
9	GND	
10	PE14	PWM
11	PA5	Analog10
12	PE13	PWM
13	PA6	Analog11
14	PE12	PWM

Pin Number	Associated to	Useful
15	GND	
16	PE11	PWM
17	PA7	Analog12
18	PE10	PWM
19	PC4	Analog13
20	PE9	PWM
21	GND	
22	PE8	PWM
23	PC5	Analog14
24	RESV1	PB11
25	AB0	Analog15
26	BATT	
27	GND	
28	3V3	

To be successful, you will need to understand how your component fits into the greater picture and what you would need to do on your module to meet the requirements.

Battery

The battery will be: [Battery LiPo 800mAh 3.7V - Micro Robotics]

Typically, try keep the max discharge at 0.5C (Capacity) for this type of battery. This implies 400mA max draw and from full to fully discharged in 2 hours. Obviously the lower you can get this number the more stable your system and longer lasting your battery would be.

Budget

Before we break this down, please note that when ordering the PCBs you will NOT POPULATE the connectors. To be clear, we will provide you with the pin headers and JST connectors described in the sensing and power subsystem sections. You will still need to provide the footprint on your PCB but will not populate them in your BOM.

Now to truly understand the difficulty of each task, you each have a total of R360 to have your board manufactured. That is 5x PCBs made @ $4, 2x PCBs populated @ $9.50 and then your component costs.

R600 / 20 = $30. $13.5 is being used to manufacture a standard sized simple board. So, you have $16.5 worth of components on the sum of your board section. $8.25 each provided you have no extended parts.

Just explaining it one more time:

Item	**Student 1 (Sensing)**	**Student 2 (Power)**
Starting Allowance	$30	$30
PCB Manufacture (5 units)	$4	$4
PCB Assembly (2 units)	$9.50	$9.50
Remaining Budget for components	30 – 13.5 = **$16.5 for 2 boards**	30 – 13.5 = **$16.5 for 2 boards**
OR alternatively	$8.25 per board	$8.25 per board

Table 4 Budget Explanation per Student

If for some reason your board fails to meet the basic requirements or is too expensive, it will not be able to be ordered. I know this budget is super tight, but while I am optimizing the upcoming costs – this is the provisional budget.

Note:

1. For this project, your design challenge is to meet the budget requirements WITHOUT the use of additional resources. Using components from the bread board assignment or elsewhere is NOT permissible in this project.
2. The extended component fee is paid PER board order but not per individual PCB of that order. For example, I am designing the sensing PCB and require an extended component which costs R1 but has an extended fee of R10. When I place my order, I will be populating 2 boards with this component BUT I will only pay the extended fee **once**. This means that to populate both boards with this component I will only pay 2xR1 + R10 = R12 NOT 2x(R1+R10) =R22.
3. Do not choose to not populate resistors and components with the expectation that you will receive them from White Lab. White Lab components will be give out on a case-by-case basis – but you certainly **will not receive WL components if you did not populate them in your order**.
4. When ordering the PCBs you will NOT POPULATE the connectors. To be clear, we will provide you with the pin headers and JST connectors described in the sensing and power subsystem

sections. You will still need to provide the footprint on your PCB but will not populate them in your BOM.

Maze

The maze will have dead ends, multiple paths to the finishing area with each pixel being a 200mm square.

An example is provided below:

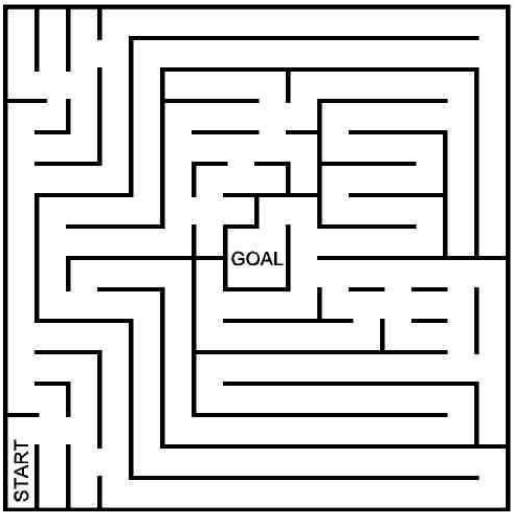


Figure 6 Example of a MM maze.

There are multiple paths to get to the goal.

Other

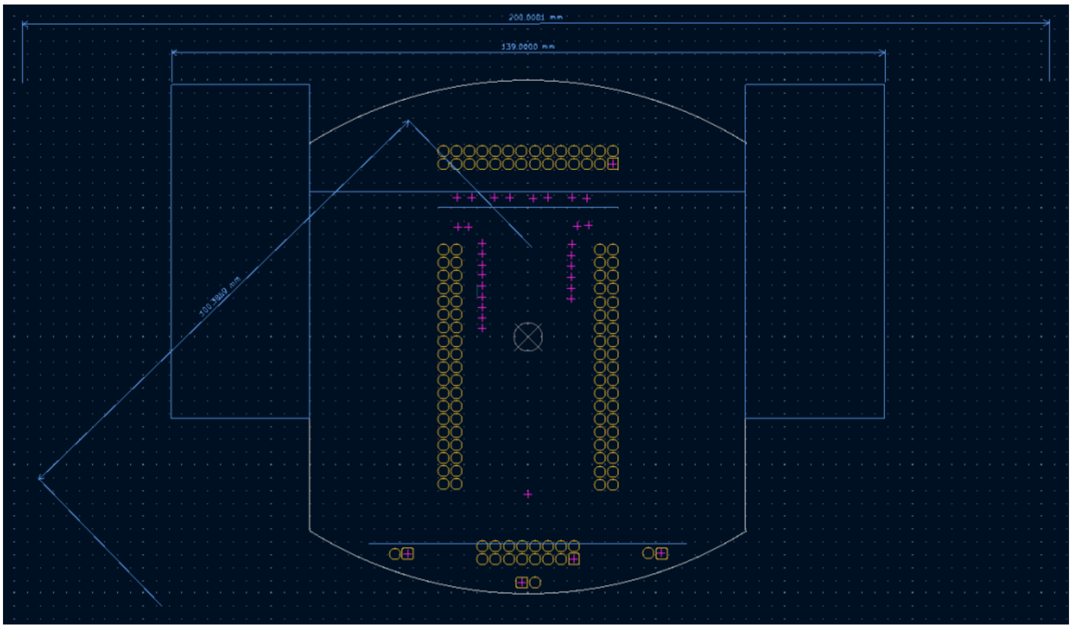


Figure 7 Dimensions of MM

PCB Production Files

You will need to submit all the production files for your PCB on Amathuba on the **24th of March 2024 at 23:55**.

Use JLCPCB tools to generate the production files and then provide:

1. A zipped file containing your Gerbers
2. The Bill of Materials (BOM) .csv file
3. The POS/CPL .csv file

Before submitting, you need to upload these files to JLPCB as if you were going to order the boards. During this activity, you will need to take a screenshot of the:

4. BOM page
5. JCP add to cart page.

These screenshots will also need to be uploaded to Amathuba with the following naming conventions***:*** ***EEE3088F_Project_JLCPCBBOM_studentnumber*** and ***EEE3088F_Project_JLCPCBacceptance_studentnumber.***

Notes

1. PLEASE WRITE YOUR STUDENT NUMBER AND GROUP NUMBER ON THE SILKSCREEN
2. The JLCPCB tool generates these files for you. If your board is not JLCPCB compliant (you do not provide the files generated by JLCPCB tools) your board will not be ordered.
3. Please make sure to choose components that are well stocked – if your components are not in stock when we order then your PCB will return incomplete.
4. Do not choose to not populate resistors and components with the expectation that you will receive them from White Lab. White Lab components will be give out on a case-by-case basis – **you will not receive WL components if you did not populate them in your order**.

Interim Design Report

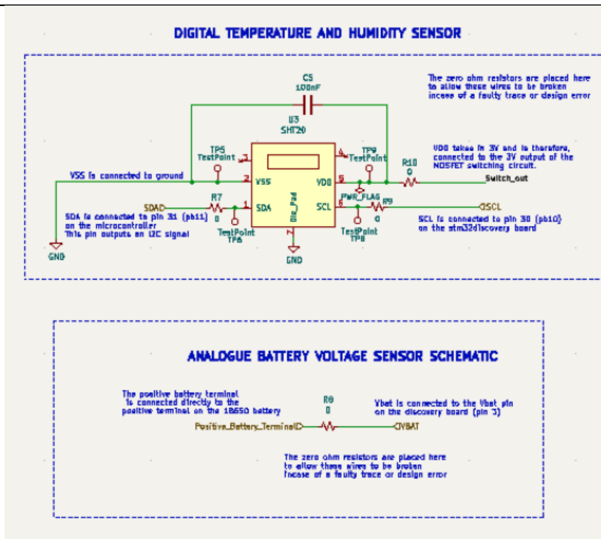
Using the **NEW LaTeX report** template available on Amathuba, you are required to write a report about the subsystem you have designed.

Notes:

- ***Do not write in first person i.e. using I, we, us, they, them, him, her, etc.***
- Marks will be allocated to report presentation (grammar, formatting, use of space, etc.).
- Feel free to add in subsection headings when writing the report, but do not make changes to the chapter names and section names in the template. When uploading to Gradescope, the rubric will expect you to have the section names described below.
- You will be penalised for not assigning pages on Gradescope.
- You will be penalised for exceeding the page limit, this could be in the form of an overall report penalty, or markers could stop marking each section after the set number of pages has been exceeded. **The 12-page page limit includes the pages used to add in figures and tables.**

Report structure		Description	Page limit
Chapter	Section		
Introduction	Problem Description	Holistically describe the project problem: - What is the context? - What is the greater project? - What is your specific problem in the context of the greater problem?	1
	Scope and limitations	- Highlight exactly what your project entails and what it does not entail. - Highlight what limits the project designs, testing, development, etc.	
	GitHub Link	Provide the link to your group's shared repository. The files should be organised logically and there should be a repo README. *Please make sure that the link is correct, and that the repository is publicly visible at the time of submission.*	
Requirements Analysis	Requirements	Fill in the requirements table in the template, give it a meaningful heading. Make sure to give each requirement a unique ID.	3
	Specifications	Fill in the specifications table in the template, give it a meaningful heading. Make sure to give each specification a unique ID.	
	Testing Procedures	Fill in the testing procedures table in the template, give it a meaningful heading. Make sure to give each testing procedure a unique ID. *This should be a simplified version of what you have in the Acceptance Testing chapter – that means that the IDs you use here need to match up in that chapter and there should be no new information here. This is just a summary of your ATP (a simple copy and paste and remove the extra columns).*	
	Traceability Analysis	- Complete the Traceability matrix which shows how the requirements, specifications and testing procedures all link. Use the IDs to show this.	

<table>
<tr><td></td><td></td><td>Then, write a description for each row in the matrix. Make sure to describe/motivate/highlight how everything links and why. These descriptions should clearly convey your thought process behind the specifications and testing procedures you have come up with. You may present this information however you see fit – it must be clear and easy to follow.</td><td></td></tr>
<tr><td>Subsystem Design</td><td>Design Decisions</td><td>This is a very important section. Here you will discuss all the design decisions you have made, for instance:
Decisions involving component selection: which components did you consider for each part of the subsystem? What makes them suitable for this project? What does not? Pros and cons of multiple options for components – if you are choosing a voltage regulator consider more than one option.
Evaluate possible solutions: did you consider multiple designs? Why did you discard some of them? What are the pros and cons of each?
Provide all calculations you used to design the subsystem
You need to <u>clearly</u> show at least 3 unique design decisions that you made:
For example, use a table to help you convey the specifications, pros and cons, etc., of each OR use images to help describe things.
Compare, compare, compare.
Provide any calculations you used. If you change any passive components to E24 values, discuss the effect this might have on the system.
Motivate your final solution by summarising all the design decisions made that lead to your final solution.
Describe your final solution.
Provide screenshots of your final schematic. Make sure to:
Include the title block with version number, author name, date, schematic title.
Include comments explaining what different parts of the schematic are for and do.
Neaten up the schematic so that it is easy to follow.
All labels used are clear and describes the connection well e.g. PA7_VADC</td><td>5</td></tr>
</table>

<table>
<tr>
<td></td>
<td></td>
<td>


Provide screenshots of your PCB (front and back and then one 3D viewer screenshot).

</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Failure Management</td>
<td>

list/Tabulate the failure management processes taken and why (<u>concisely</u>)
Make sure that you give each a clear descriptive name that describes what it is.

This is specifically with regards to the PCB design. This is where you discuss how you designed the PCB to mitigate component failures/trace issues/errors in your circuit design/etc.
</td>
<td></td>
</tr>
<tr>
<td></td>
<td>System Integration and Interfacing</td>
<td>

Replace the table with your own table that clearly shows which pins connect to and interact with other parts of the system.
Include a simple, high level block diagram showing how your subsystem fits into the larger system. This should include information about expected inputs/outputs or what the interactions are (brief and to the point).

</td>
<td></td>
</tr>
<tr>
<td>Acceptance Testing</td>
<td>Tests</td>
<td>
This is specifically with regards to HOW you will test your PCB to verify that every part of it works as expected. This is different to the Failure management section in that you are not describing ERCs, DRCs, or general measures you took to prevent failure on the board. Rather, you now have the board and must describe how you will prove that you met the requirements and specifications.

Complete the table in the template:

Provide each test with a unique ID
What is the test? Why is it being tested?
Concisely explain how you will test (using jumpers, test points, equipment, etc.). Explain the procedure.
What is the pass/fail criteria.

</td>
<td>2</td>
</tr>
</table>

		Please note that each requirement and specification could have more than one ATP associated with it.	
	Critical Analysis of Testing	This will only be completed for the final report.	0
Conclusion		This will only be completed for the final report.	0
	Recommendations	This will only be completed for the final report.	0
Extra page for formatting.			**1**

Final Demonstration

Notes

- You will need to sign up for an individual demonstration slot. You do not need to be in the same session as your group member as this is an individual assessment.
- You are required to bring a printed copy of your schematic – you will not be able to give justification for marks without this.

Sensing Subsystem Demonstration

The Code

As described in other sections of this project brief, you are expected to write code to prove that your sensing board can detect obstacles/walls in front, to the left and to the right of the micromouse. For the demonstration, your code will need to meet the following requirements:

1. Your code needs to be compatible with the 2nd year STM32F0 devkit. To do this, you should use a combination of STM32CubeMX with VSCode – Justin has put some sample code and getting started instructions together (available on Amathuba). Then look at the STM32 Getting Started guides, these contain a lot of tips and tricks on using the STM32Cube software as well as HAL. Using HAL means that you can toggle an LED on and off as simply as HAL_GPIO_TogglePin(…) rather than having to assign the IDR registers correctly. *Of course, you may do things on the register level if you would like. Using HAL might just prove simpler for this application.*
2. You specifically need to toggle PB7=left, PB6=front and PB5=right as shown in the image below:

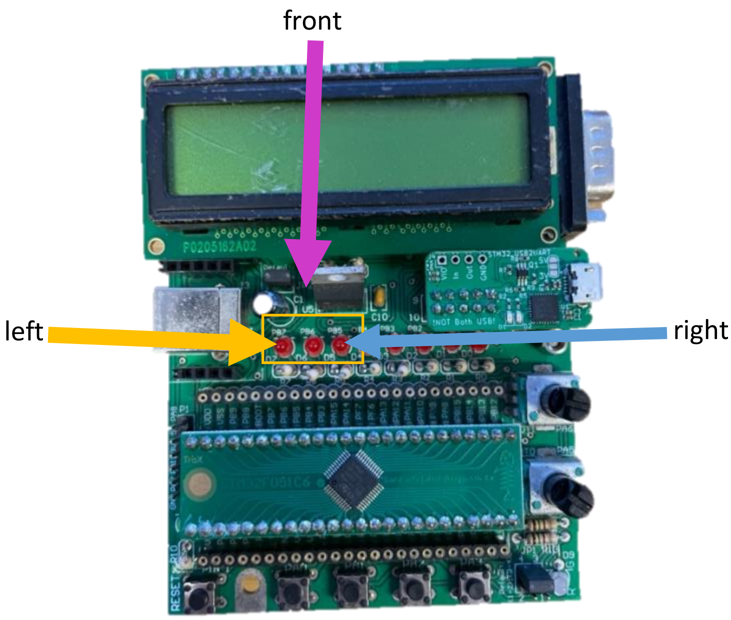


Figure 8 Wall detected indication LEDs

We will be attaching scopes to these lines and **will not be changing the equipment**. If you do not follow these guidelines your board will not be marked during your session.

3. Your PCB needs to fit onto the testing jig PCB. If you have designed your PCB with the incorrect headers, then you will need to find a solution to connecting to our rig (there are several options so just put some thought into it)

The Testing Jig

To test that you have designed a PCB that meets the project requirements, we have designed a testing jig. The jig has two parts: the maze pixel and the testing PCB. The testing PCB will be placed in the centre of the maze pixel and will not be moved throughout the test. The walls of the maze pixel will be moved around the PCB to simulate a micromouse moving through a maze.

<u>Testing PCB</u>

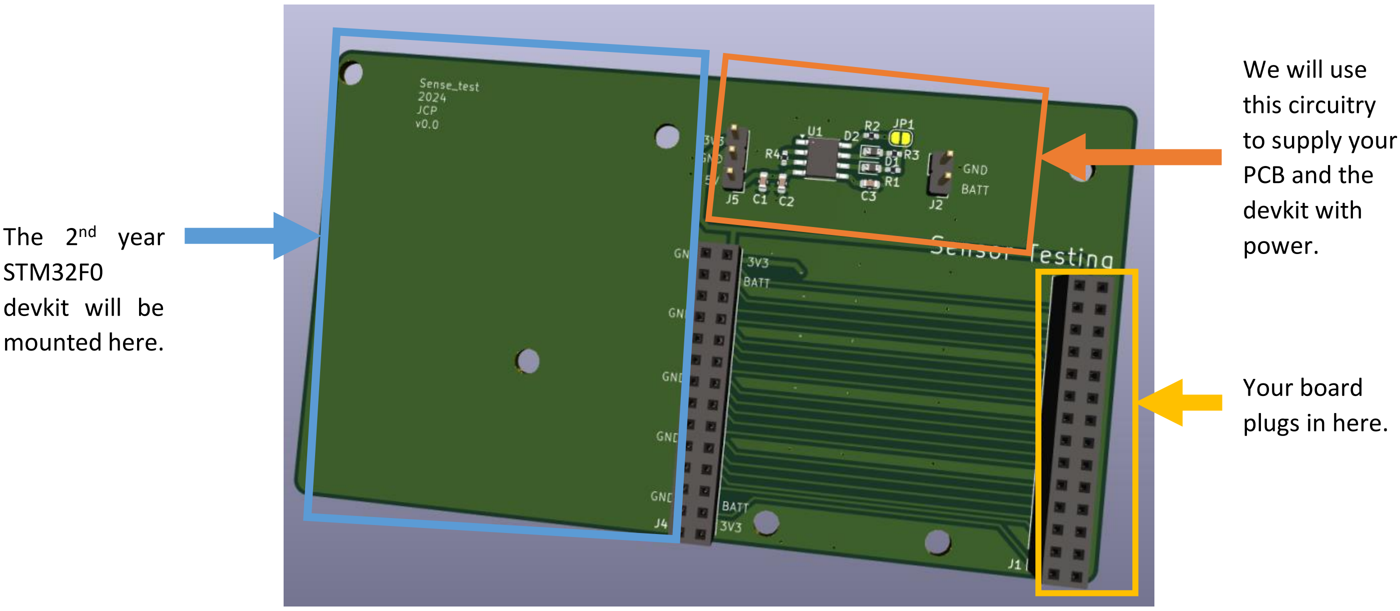


Figure 9 PCB for the testing jig.

<u>Maze Pixel</u>

This is a single pixel/block of the maze (which you will be working with in the 2nd semester design course). It is made of wood and designed to be changeable – all the walls can be removed.

Figure 10 Top view of maze pixel

Figure 11 Side view of maze pixel

The testing PCB will be secured in the middle of the maze pixel and will remain there for all demonstrations. The walls will be removed one by one to test whether your sensor is able to sense the changes (we will monitor the 3 LEDs mentioned in the coding section). The possible permutations are shown below:

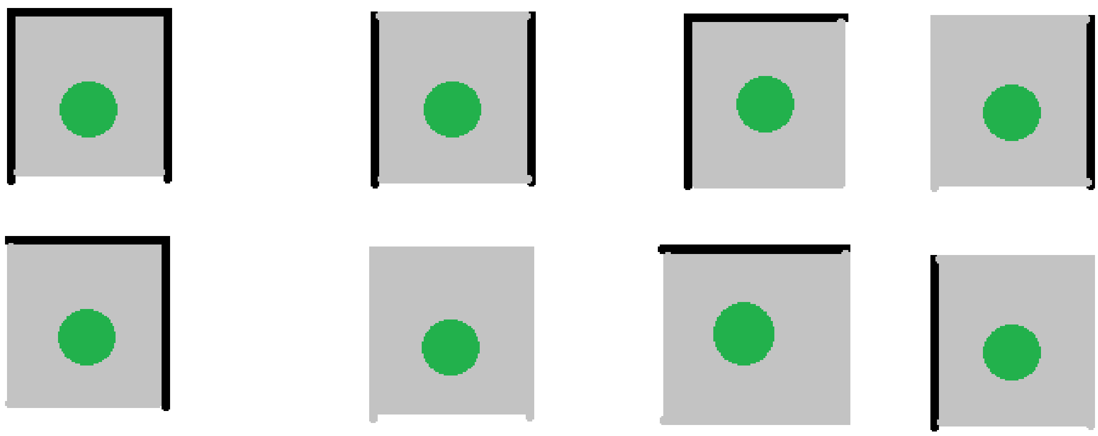


Figure 12 Testing jig: maze pixel wall state permutations (black lines = walls, grey = base plate, green dot = testing PCB)

The Procedure

The following is expected of you at the demonstration:

1. Arrive to your booked demonstration slot **on time** with:
    a. A printed copy of your schematic
    b. Your PCB
    c. Your code, ready to flash onto the STM
    d. Both populated PCB boards (the boards that have components on)
2. Flash your code onto the STM32F0
3. Connect the wires from the STM to the testing PCB in the configuration that you need (it is a good idea to bring a list of the connections you need so that you can do this wiring quickly).
4. Plug your sensing PCB into the testing PCB
5. The marker will supply power to the testing PCB
6. The walls will be moved in the configurations shown in *Figure 12* and you will be awarded marks accordingly.
7. You will need to complete all of this within the time slot.

The Rubric

<table>
<tr><td colspan="2">12</td><td colspan="3">Total Marks</td></tr>
<tr><td rowspan="4">2</td><td>+0.5</td><td rowspan="4">Appearance</td><td colspan="2">No visible rework done to the PCB (cut traces, extra wires, different components, etc.)</td></tr>
<tr><td>+0.5</td><td colspan="2">All ICs on board</td></tr>
<tr><td>+0.5</td><td colspan="2">Board shape would be practical for the micromouse (size, shape, etc.)</td></tr>
<tr><td>+0.5</td><td colspan="2">Correct connectors have been used - board would fit onto the micromouse</td></tr>
<tr><td rowspan="3">10</td><td>+1</td><td rowspan="3">Function: Sensing</td><td colspan="2">A low current mode exists</td></tr>
<tr><td>+1</td><td colspan="2">Hysteresis is used - single transition event when the LED turns on
[Probe an LED]</td></tr>
<tr><td>+1</td><td>Sensing</td><td>Right</td></tr>
</table>

	+1			Right, Front
	+1			Right, Front, Left
	+1			Front, Left
	+1			Front
	+1			Left
	+1			Left, Right
	+1			No walls (should not sense anything)

Table 5 Final Demonstration Rubric: Sensing

Power Subsystem Demonstration

The Testing Jig

The jig is an all-in-one PCB designed to test that your PCB meets all the project requirements. If facilitates separate and repeatable testing of different functionality of your PCB, namely battery charging, motor control, switching ON and OFF and SOC measurements.

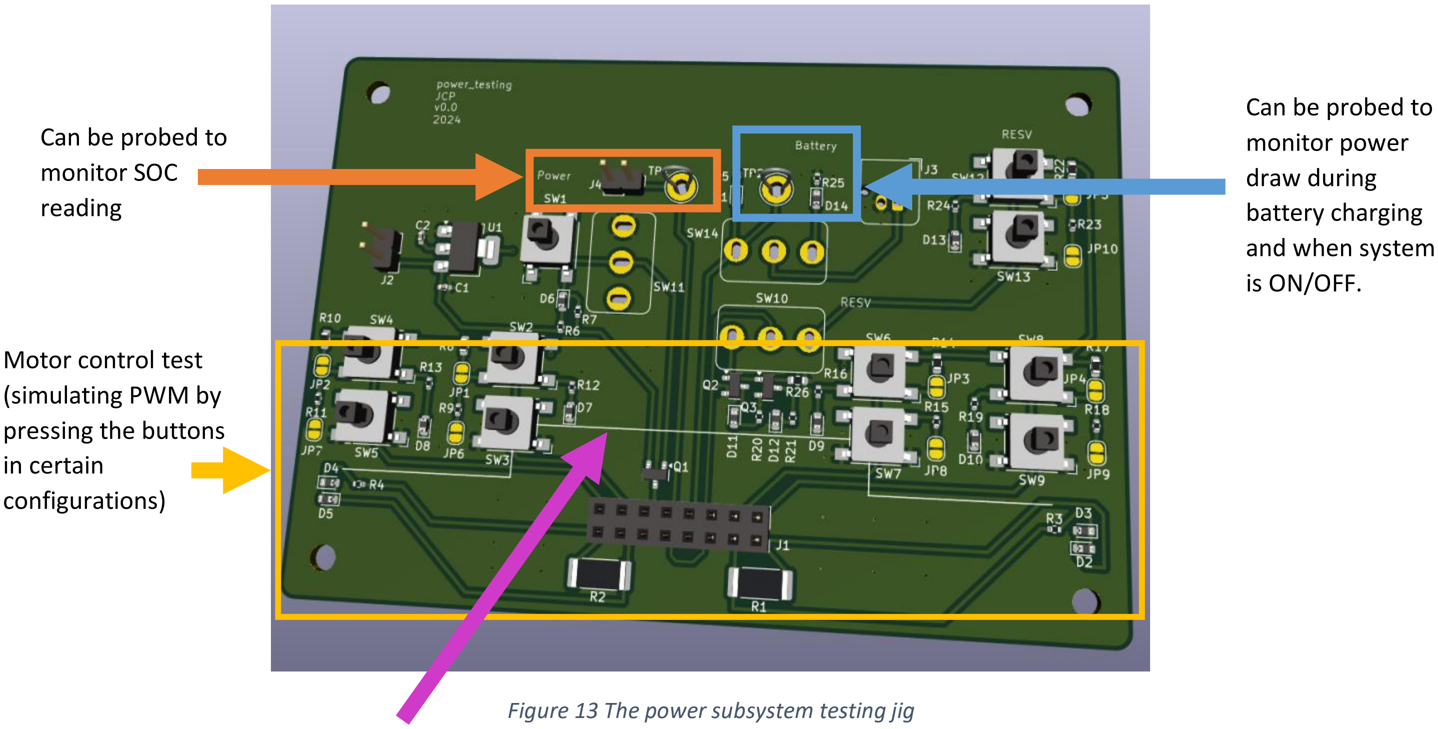


Figure 13 The power subsystem testing jig

Line to test if your board shape and size would fit the micromouse.

Note: the default state for PWM is LOW meaning OFF. Please notify us if this is not what you expected.

The Procedure

The following is expected of you at the demonstration:

1. Arrive to your booked demonstration slot **on time** with:
    a. A printed copy of your schematic
    b. Your PCB
    c. Your code, ready to flash onto the STM
2. Plug your power PCB into the testing jig
3. Each component of your board will be tested (as per the rubric)
4. You will need to complete all of this within the time slot.

The Rubric

12		Total Marks	
2	+0.5	Appearance	No visible rework done to the PCB (cut traces, extra wires, different components, etc.)
	+0.5		All ICs on board

	+0.5		Board shape would be practical for the micromouse (size, shape, etc.)	
	+0.5		Correct connectors have been used - board would fit onto the micromouse	
10	+1	Function: Power	H-bridge	Off-State: Current <1mA *[Marking note: Make sure Diodes are off, Vres = low as possible, push buttons to achieve this]*
	+1			On-State: Voltage > 95% Supply Voltage.
	+1			The design for both H-bridges is symmetric
	+1			No shoot through possible (should not have a state that damages the system.) *[Look at schematic or battery/PSU droop on the scope]*
	+1		Switch	Switch can handle 1A or student has a solution to avoid needing this. *[Look at schematic or Justification- show calculations]*
	+1			Current in off state <1mA *[Look at schematic or Justification- show calculations]*
	+1		Analog Voltage	Scales with battery voltage
	+1			Highest value less than 3.3V *[Look at schematic or Justification- show calculations]*
	+1		Charge Battery	Does the Batt pin go to 4.2V
	+1			Battery could be connected to the board (correct header)

Table 6 Final Demonstration Rubric: Power

Final Report

Using the **NEW LaTeX report** template available on Amathuba, you are required to write a report about the subsystem you have designed.

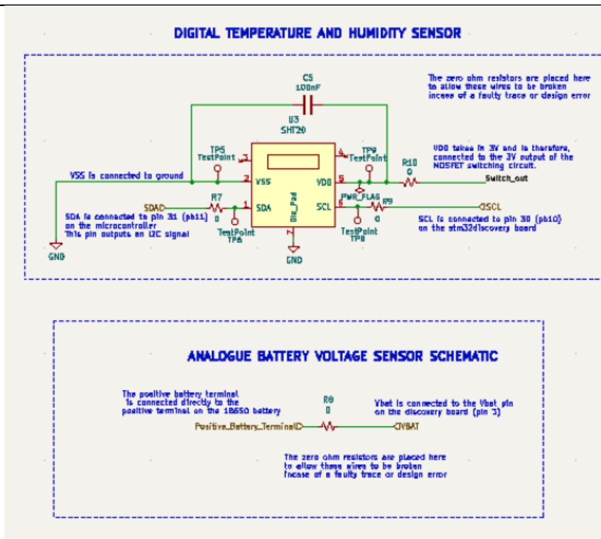
Notes:

- ***Do not write in first person i.e. using I, we, us, they, them, him, her, etc.***
- Marks will be allocated to report presentation (grammar, formatting, use of space, etc.).
- Feel free to add in subsection headings when writing the report, but do not make changes to the chapter names and section names in the template. When uploading to Gradescope, the rubric will expect you to have the section names described below.
- You will be penalised for not assigning pages on Gradescope.
- You will be penalised for exceeding the page limit, this could be in the form of an overall report penalty, or markers could stop marking each section after the set number of pages has been exceeded. **The 18-page page limit includes the pages used to add in figures and tables.**

Report structure		Description	Page limit
Chapter	Section		
Introduction	Problem Description	Holistically describe the project problem: - What is the context? - What is the greater project? - What is your specific problem in the context of the greater problem?	1
	Scope and limitations	- Highlight exactly what your project entails and what it does not entail. - Highlight what limits the project designs, testing, development, etc.	
	GitHub Link	Provide the link to your group's shared repository. The files should be organised logically and there should be a repo README. *Please make sure that the link is correct, and that the repository is publicly visible at the time of submission.*	
Requirements Analysis	Requirements	Fill in the requirements table in the template, give it a meaningful heading. Make sure to give each requirement a unique ID.	3
	Specifications	Fill in the specifications table in the template, give it a meaningful heading. Make sure to give each specification a unique ID.	
	Testing Procedures	Fill in the testing procedures table in the template, give it a meaningful heading. Make sure to give each testing procedure a unique ID. *This should be a simplified version of what you have in the Acceptance Testing chapter – that means that the IDs you use here need to match up in that chapter and there should be no new information here. This is just a summary of your ATP (a simple copy and paste and remove the extra columns).*	
	Traceability Analysis	- Complete the Traceability matrix which shows how the requirements, specifications and testing procedures all link. Use the IDs to show this.	

		- Then, **DISCUSS** each row in the matrix. Make sure to **CRITICALLY** analyse/describe/motivate/highlight how everything links and why. These descriptions should clearly convey your thought process behind the specifications and testing procedures you have come up with. *You may present this information however you see fit – it must be clear and easy to follow.*	

Subsystem Design	Design Decisions	This is a very important section. Here you will discuss all the design decisions you have made, for instance:
		- Decisions involving component selection: which components did you consider for each part of the subsystem? What makes them suitable for this project? What does not? Pros and cons of multiple options for components – if you are choosing a voltage regulator consider more than one option.
		- Evaluate possible solutions: did you consider multiple designs? Why did you discard some of them? What are the pros and cons of each?
		- Provide all calculations you used to design the subsystem
		
		You need to *clearly* show at least 3 unique design decisions that you made:
		- **For example, use a table to help you convey the specifications, pros and cons, etc., of each OR use images to help describe things.**
		- **Compare, compare, compare.**
		- **Provide any calculations you used. If you change any passive components to E24 values, discuss the effect this might have on the system.**
		
		Motivate your final solution by summarising all the design decisions made that lead to your final solution.
		- Describe your final solution.
		- Provide screenshots of your final schematic. Make sure to:
		    o Include the title block with version number, author name, date, schematic title.
		    o Include comments explaining what different parts of the schematic are for and do.
		    o Neaten up the schematic so that it is easy to follow.
		    o All labels used are clear and describes the connection well e.g. PA7_VADC
			5

<table>
<tr><td rowspan="3"></td><td></td><td>


- Provide screenshots of your PCB (front and back and then one 3D viewer screenshot).
</td><td rowspan="3"></td></tr>
<tr><td>Failure Management</td><td>
- list/Tabulate the failure management processes taken and why (**<u>concisely</u>**)
- Make sure that you give each a clear descriptive name that describes what it is.

This is specifically with regards to the PCB design. This is where you discuss how you designed the PCB to mitigate component failures/trace issues/errors in your circuit design/etc.
</td></tr>
<tr><td>System Integration and Interfacing</td><td>
- Replace the table with your own table that clearly shows which pins connect to and interact with other parts of the system.
- Include a simple, high level block diagram showing how your subsystem fits into the larger system. This should include information about expected inputs/outputs or what the interactions are (brief and to the point).
</td></tr>
<tr><td>Acceptance Testing</td><td>Tests</td><td>
This is specifically with regards to HOW you will test your PCB to verify that every part of it works as expected. This is different to the Failure management section in that you are not describing ERCs, DRCs, or general measures you took to prevent failure on the board. Rather, you now have the board and must describe how you will prove that you met the requirements and specifications.

Complete the table in the template:
- Provide each test with a unique ID
- What is the test? Why is it being tested?
- Concisely explain how you will test (using jumpers, test points, equipment, etc.). Explain the procedure.
- What is the pass/fail criteria.
</td><td>2</td></tr>
</table>

		Please note that each requirement and specification could have more than one ATP associated with it.	
	Critical Analysis of Testing	Discuss your **5 most significant ATPs**; amongst other things, this includes what went wrong and what went right. Remember to provide evidence of your results. This section requires a critical analysis, this is more than just a “AT01 was passed or AT03 was failed” statement. *Note: we will only mark the first 5 you discuss.*	5
Conclusion	You will be marked based on the following criteria and how well you structure it: • Are the results interpreted clearly? • Are the conclusions reached clearly supported in the results and linked to the research question/engineering problem? • Are suggestions made for further work? (emphasis is placed on the quality of these suggestions)		1
Extra page for formatting.			**1**

Errata

Version 2 – full brief released.

Version 3 – updated budget and sensing. Added JST connector link and more information to the report expectations.

Version 4 – fixed one number in the budget.

Version 5 – corrected the Gerber submission date to 24 March 2024.

Version 6 – Added a requirement on the switch states for power, Added a hint to the battery charger and thoughts. (20 March 2024)

Version 7 – Added in the Interim Design Report details (28 March 2024), Added Production Files submission requirement/note. (25 March 2024)

Version 9 – Updated the Interim Design Report - System Integration and Interfacing requirements. (30 March 2024).

Version 10 – Updated the Interim Design Report: (8 April 2024)

- No need for differentiation between functional and user requirements (students will not be penalised if they do decide to do the differentiation – as long as it is logical)
- The design section has been updated: no more 'final design subsection' it is now a part of 'design decisions'.

Version 11 - Updated the Interim Design Report: (9 April 2024)

- Clarified the difference between failure management and the acceptance tests.
- Added an update to the traceability analysis.

Version 12 – Interim Design Report submission date changed

Version 13 – Added the requirement to submit the reports to Amathuba as well as Gradescope.

Version 14 – Added the demonstration procedure for both power and sensing.

Version 15 – Added demonstration rubrics and the final report breakdown. (3 May 2024)