# Micro-mouse Project Brief

*EEE3088F 2025*

# 0 General Instructions

This document contains the project description as well as all documentation on the micro-mouse (MM). 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, 28 March 2025 at 5pm on Amathuba)
2. The *Interim Design Report* (Friday, **25** April 2025 at 5pm on Amathuba AND Gradescope)
3. The *Lab demonstration* (can be conducted every lab session until the final due date on Wednesday, 14 May in White Lab)
4. The *Final Report* (Friday, 16 May 2025 at 5pm on Amathuba AND Gradescope)

A special note is that the report assessments are **individual.** You are each expected to submit individual reports as per the assignment descriptions (later in this document).

## Group work

For the remainder of the semester, you will continue to work in your chosen pairs from the breadboard assignment.

In your pairs, you will work together on a **single** PCB solution and will **both** be required to attend the lab demonstrations. The demonstration will be a **group mark.** However, you will **each** be required to submit individual reports and **will receive individual report marks.** In these reports, you will write the design and other sections based on the parts of the system you chose to design (more on this later).

As always, 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. Effective communication is a core engineering skill. Asking the right questions and engaging with clients ensures that you will deliver the correct product. If something isn't clear, research it. If you're still unsure, ask—suffering in silence leads to mistakes and lost time.
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 (these will also be noted in the Errata section).
3. During lab sessions, tutors will be available to help you. However, to make the most of this time, you will need to do additional work outside of these hours.
4. ***All reports must be written in LaTeX.***
5. ***You will need to use GitHub for version control and be required to upload the link in your reports.***

# 1 Project Overview

The course project involves 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, however 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 the possibility of some minor software components). The complexity of this project is in meeting the requirements while still adhering to the STRICT budget.

The micro-mouse has been compartmentalised into four modules: the processor, motherboard, sensing and power. The processor, motherboard and sensing modules have been designed and will be given to you. **Your project involves designing and manufacturing the power module.** More information about each module is provided in the table below.

| Module | Description |
|---|---|
| The motherboard | The motherboard has already been designed is responsible for connecting all the PCBs together. It is the base board that all other modules will slot onto. |
| The processor | The processor board has already been designed. It has an **STM32L476** microcontroller onboard, a significant upgrade in performance compared to the $2^{nd}$ 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 the micro-mouse's supplementary modules. |
| The sensor | This module has already been designed and will be responsible for detecting/sensing objects. |
| 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. |

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

# 2 Project Background

This section contains information about the entire MM. It is important to take note of all subsystems and their requirements when working on the project.

## The Motherboard

| Interfaces | Description |
|---|---|
| Processor board | Two sets of 2x19 (2.54mm pin pitch) pin sockets (*pinouts listed in the processor section*). |
| Motors | Two sets of 2x1 (2.54mm pin pitch) pin headers. |
| Sensor board | 2x16 (2.54mm pin pitch) pin sockets (*pinouts listed in the sensor section*). |
| Power board | 2x16 (2.54mm pin pitch) pin sockets (*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 must design your power PCBs to fit onto the specified power connection. *Take special note the position of the center of rotation of the robot. The center of rotation is under the IMU.*

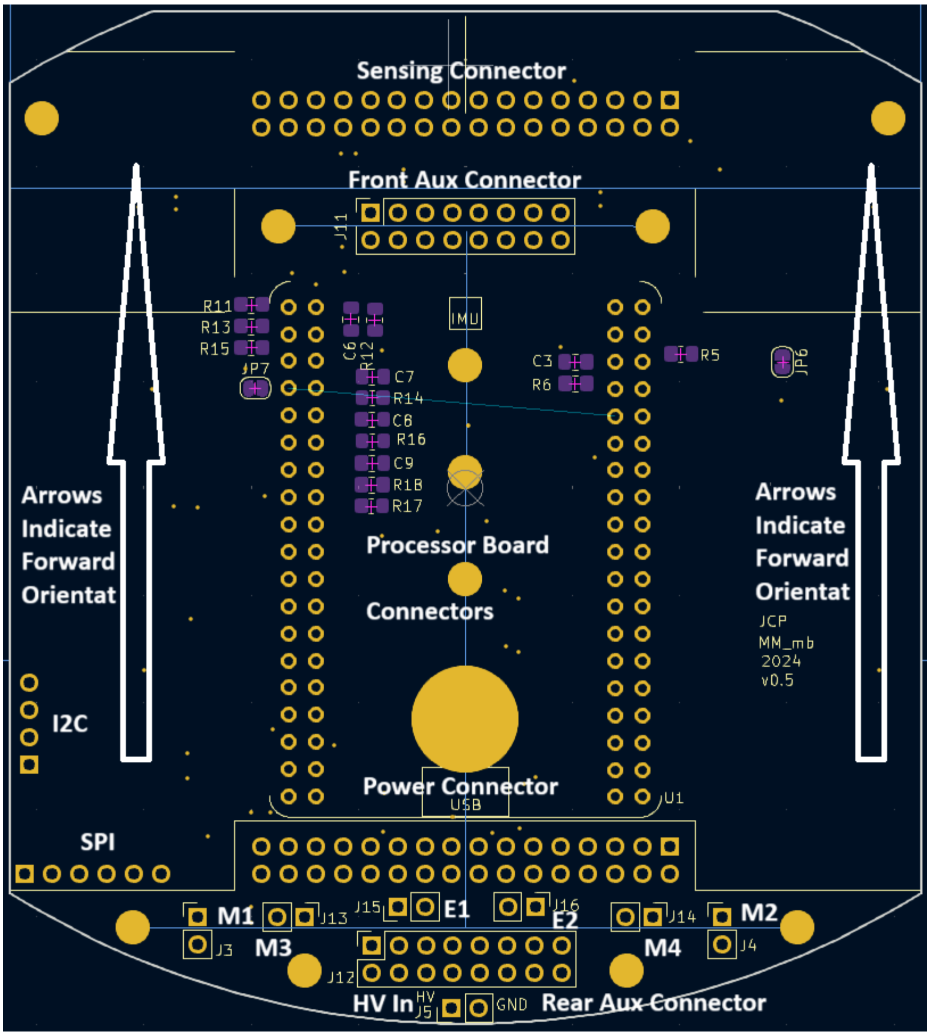



*Figure 1 Pin-view of the MM Motherboard PCB.*

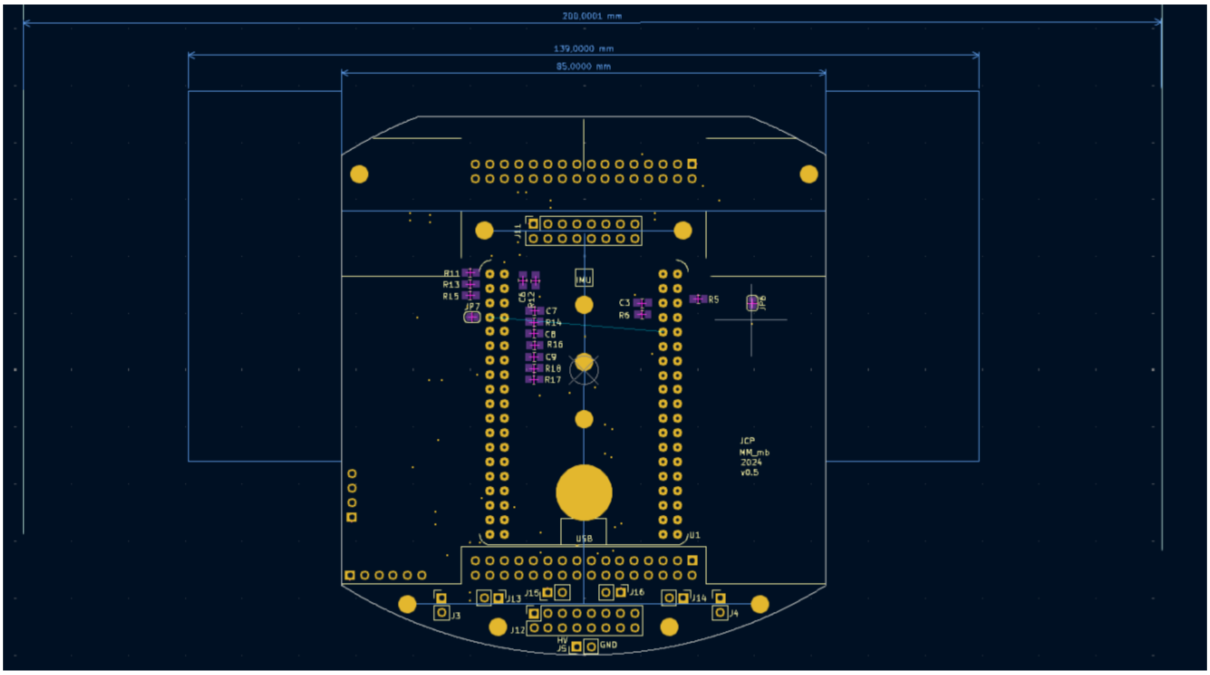



*Figure 2 Dimensions of MM*

## The Processor

The processor board with the STM32L476 microcontroller on it. The images below depict the 3D PCB render and the schematic of the **processor board**.

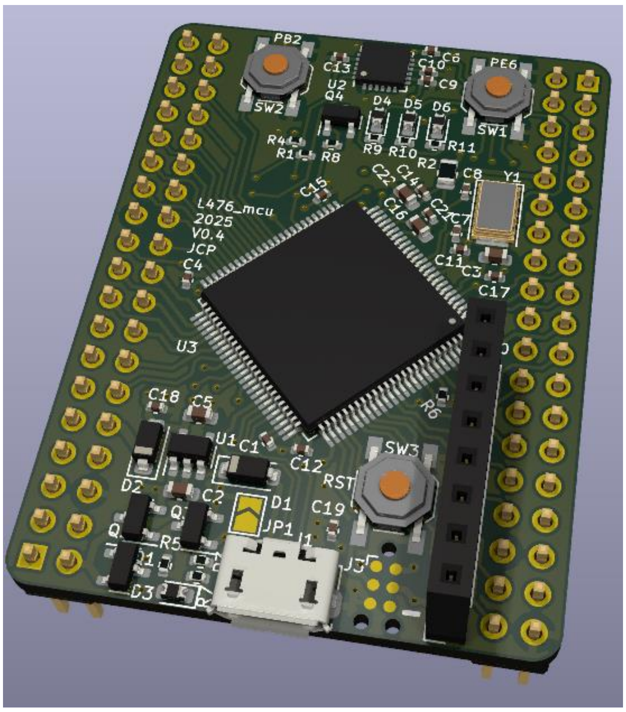



*Figure 3 3D Render of the MM Processor PCB*

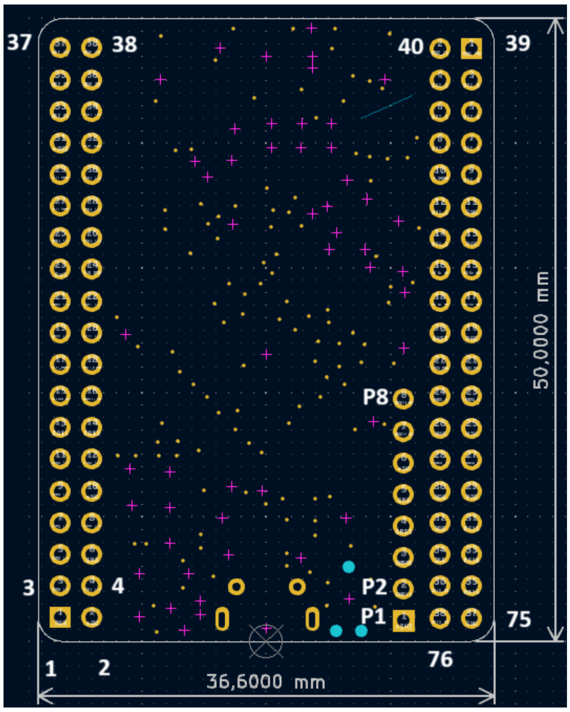



*Figure 4 Pin-view of the MM Processor PCB*

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 (P1) | Target Voltage sense | |
| Prog_ SWCLK (P2) | SWCLK | |
| Prog_ SWDIO (P3) | SWDIO | |
| Prog_ NRST (P4) | NRST | |
| Prog_ GND (P5) | GND | |
| Prog_ 5V (P6) | Programming Power | |
| Prog_VCP_In (P7) | USART1_Tx | |
| Prog_VCP_Out (P8) | USART1_Rx | |

*Table 2 Electrical pinout of the MM Processor PCB*

## Sensing System

The sensor module is effectively the “eyes” of the MM and provides information to the processor to determine whether there is an obstruction in the way of the MM.

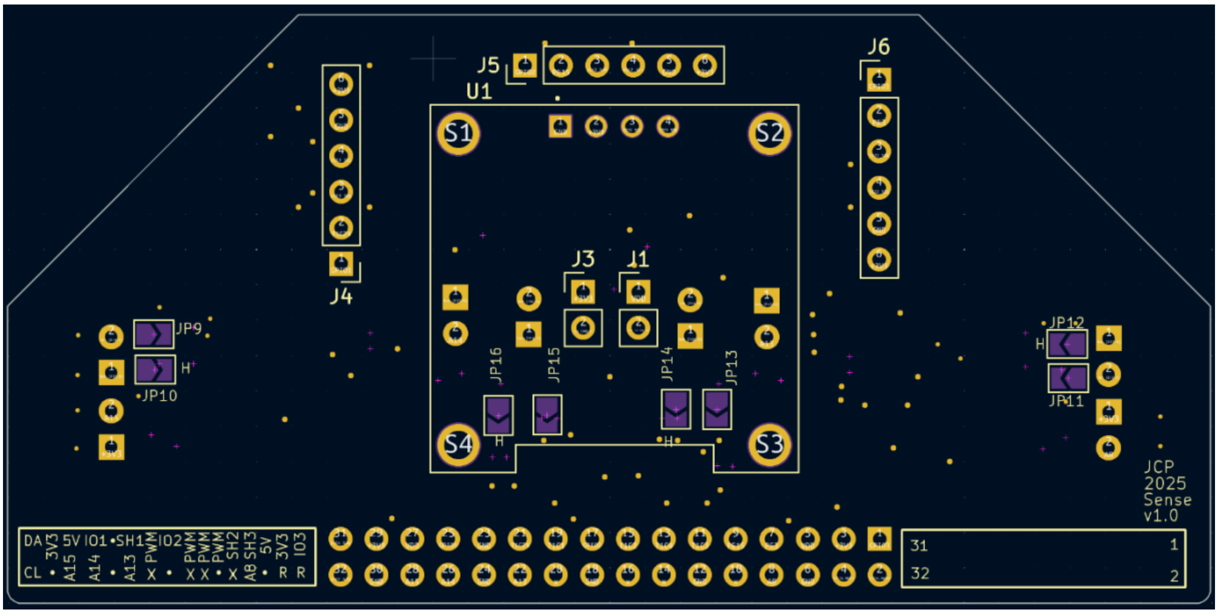



*Figure 5 MM Sensing pinouts that plug in the MM Motherboard PCB*

| Pin Number | Associated to | Useful |
|---|---|---|
| 1 | GPIO3 | PE2 |
| 2 | RESV | PE1 |
| 3 | 3V3 | |
| 4 | RESV | PC2 |
| 5 | 5V | |
| 6 | GND | |
| 7 | XSHUT3 | PE3 |
| 8 | A8 | PA3 |
| 9 | XSHUT2 | PE15 |
| 10 | NC | |
| 11 | T1C3 | PE13 |
| 12 | GND | |
| 13 | T1C4 | PE14 |
| 14 | NC | |
| 15 | T1C2 | PE11 |
| 16 | NC | |
| 17 | GPIO2 | PE12 |
| 18 | GND | |
| 19 | T1C1 | PE9 |
| 20 | NC | |
| 21 | XSHUT1 | PE10 |
| 22 | A13 | PC4 |
| 23 | GND | GND |
| 24 | GND | PB11 |
| 25 | GPIO1 | PE8 |
| 26 | A14 | PC5 |
| 27 | 5V | |
| 28 | A15 | PB0 |
| 29 | 3V3 | |
| 30 | GND | GND |
| 31 | SDA | PB11 |
| 32 | SCL | PB10 |

*Table 3 Electrical pinout of the Sensor PCB*

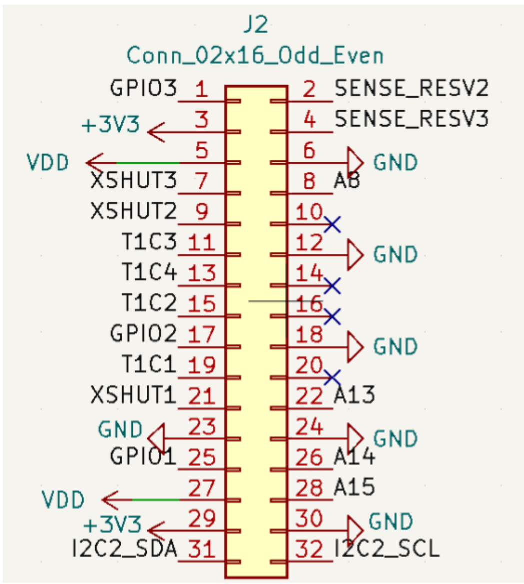



*Figure 6 Sensor PCB Connector and pin layout*

### *Micro-sensing Board*

The primary improvement between sensing boards in 2024 and 2025 is that the previous only had proximity sensors which provided information on whether there was a wall next to the robot. In 2025, there are 3 distance sensors for left/right and forward sensing. This should provide significantly more information for the MM.

To achieve this, STM Time of Flight sensors (ToF) were implemented. The sensors operate on I2C and were built onto a smaller daughter board called the MicroSensing board (for now).

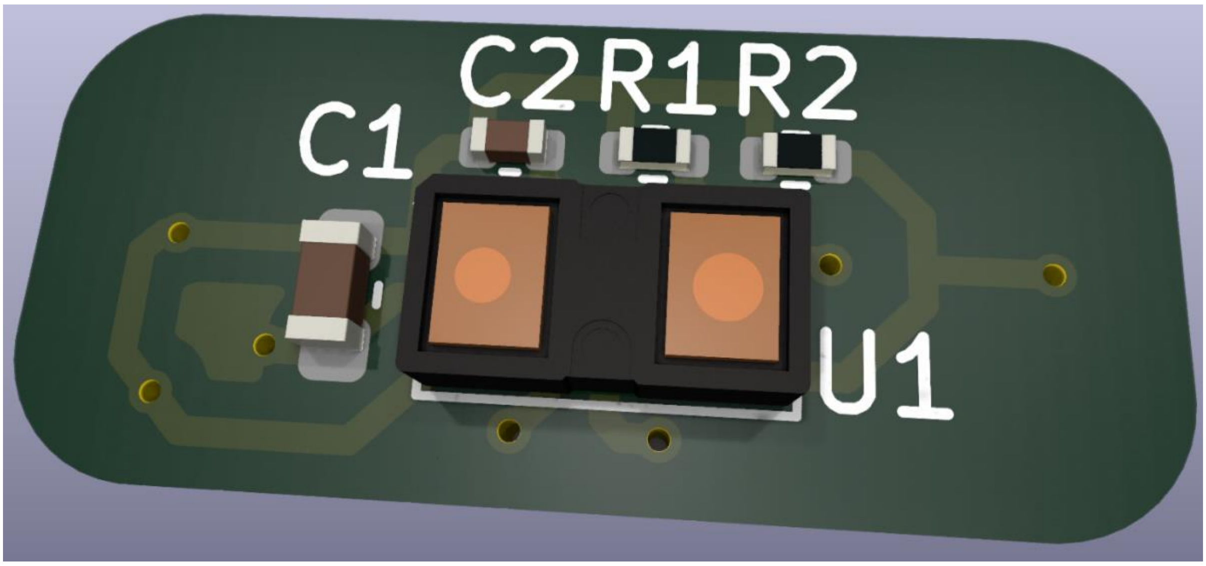



*Figure 7 MM micro-sensing daughter board PCB*

### *The Maze*

The MM will ultimately need to navigate through a maze (this will be the second semester design goal). The maze will have dead ends and **multiple paths to the finishing area** with each pixel being a 200mm square. An example is provided below:

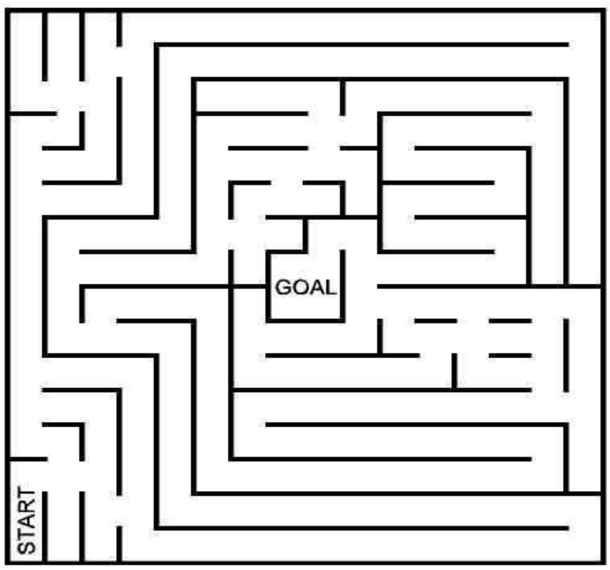



*Figure 8 Example of a MM maze.*

## Power System

In the first semester, you will be working in your groups to design and manufacture this subsystem.

## 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. The better you control the charging and discharging conditions the better the overall battery operational lifetime. This isn’t a requirement for our project, but something worth noting.

# 3 Project Description

## Overview

As with the breadboard assignment, you will continue to work in your groups. To be clear, you will work together on designing a single PCB to fulfill the **power module** requirements.

Although you are working in a group, you will each be required to take responsibility for certain parts of the power module's design. The **demonstration will be assessed as a group** whilst both the Interim and Final Design Reports are **individual assessments**. In the design sections of the reports you will focus in on writing about the parts of the solution that you designed.

## Power Subsystem Requirements

For the course project your group is required to design the power subsystem for the MM. To achieve this, your module will need to:

- Operate up to 4 motors bidirectionally with the pins available to you (*listed in the power pinout table*). You will need to control 2x brushed DC motors which could each draw 200mA at the highest voltage of a 1S1P battery (the battery is further specified in the battery section). The other 2x motors are for the auxiliary connection and need to operate 500mA each.
- Place an INA219 for monitoring the battery on the I2C Bus and configure it correctly with respect to the hardware (cannot have BOTH A0 AND A1 on GND)
- Charge the battery from the 9V input pin (*listed in the power pinout table*).
- Have two charging modes for a higher and lower charging current for the battery (200mA, and approximately 600mA ±100mA from the battery perspective.
- Integrate USB C and get 9V out of the USB Host
- Provide 2x External Load Switching at 1A each (High Side connected to your 5V)
- Provide a 3V3 5% accuracy (300mA max) and 5V Out 5% accuracy (1.5A max)
- Provide an ON/OFF switch. **OFF state: battery draw <30uA. ON state: can provide your robot peak current of 2A.** The switch needs to shut down 5V and 3V3.

Spend a little time looking through these requirements, ranking them in difficulty and share the tasks evenly between you and your colleague. You will write about your chosen parts your individual report.

Additionally, the following connectors need to be included:

1. **A JST PH 2mm pin pitch** connector for the battery. The battery will be tucked away between the motherboard PCB and the processor PCB.
2. A **2x16 (2.54mm pin pitch) pin header**: It needs to fit onto the MM's motherboard 2x16 2.54mm pitch connector with the correct pinout. **Hint: be sure to pay attention to the alignment of the pins**
3. The maximum dimensions for your board are 82 x 60 for it to fit under the motherboard PCB. Please note this even though JLC allows larger.
4. The shape of the board is an important consideration: If your board extends to any of the drill holes on the MB, your power

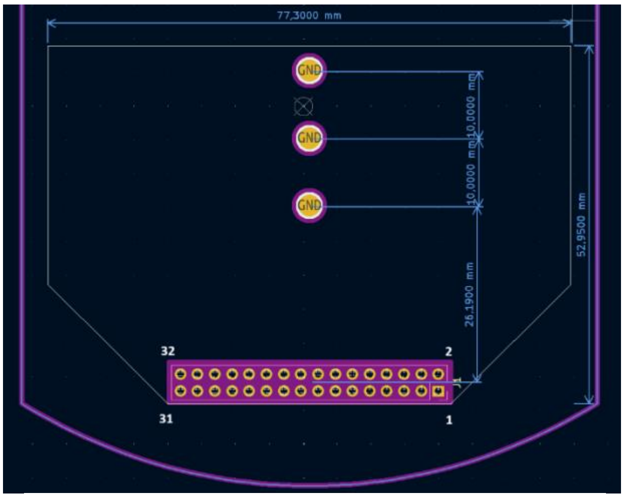



*Figure 9 MM Power PCB pinouts that plug onto the MM Motherboard PCB (Example shape).*

board needs to use them for securing your PCB as the weight of the PCB will be too much for the connector alone.

5. The Motherboard dimensions arc radius is 80mm and the connectors centre is 15.3mm from the apex of the arc.

The table below surmises the connections for the board

| Connector | Pin Number | Associated to | Useful Info |
|---|---|---|---|
| | 1 | | Motor2_B_OUT |
| | 2 | | Motor2_A_OUT |
| | 3 | | Motor4_A_OUT |
| | 4 | PD12/ TIM4C1 | MOTOR2_CTRL1 |
| | 5 | | Motor4_B_OUT |
| | 6 | PD13/ TIM4C2 | MOTOR2_CTRL2 |
| | 7 | PD2 | CTRL_EXT_LOAD2 |
| | 8 | PD14/ TIM4C3 | MOTOR4_CTRL1 |
| | 9 | PD6/RESV | USART2_RX |
| | 10 | PD15/ TIM4C4 | MOTOR4_CTRL2 |
| | 11 | | EXT_LOAD2_OUT |
| | 12 | PD5/RESV | USART2_TX |
| | 13 | PB4 | FAST_CHARGE_CTRL |
| | 14 | | Battery |
| | 15 | PD7/ RESV | |
| | 16 | PB8 | I2C1_SCL |
| | 17 | PB9 | I2C1_SDA |
| | 18 | | 3V3 Out |
| | 19 | | GND |
| 2x16 pin (2.54mm pin pitch) header | 20 | | 5V Out |
| | 21 | | EXT_LOAD1_OUT |
| | 22 | HV | High Voltage (9V) |
| | 23 | | GND |
| | 24 | PC6/ TIM3C1 | MOTOR3_CTRL1 |
| | 25 | PD11 | CTRL_EXT_LOAD1 |
| | 26 | PC7/ TIM3C2 | MOTOR3_CTRL2 |
| | 27 | | MOTOR3_A_OUT |
| | 28 | PC8/ TIM3C3 | MOTOR1_CTRL1 |
| | 29 | | MOTOR3_B_OUT |
| | 30 | PC9/ TIM3C4 | MOTOR1_CTRL2 |
| | 31 | | MOTOR1_B_OUT |
| | 32 | | MOTOR1_A_OUT |
| Battery Connection | JST PH 2mm pin pitch | Your board | |
| USB Connection | Type C | Your board | |

*Table 4 Electrical pinout of the MM Power PCB*

## Budget

*Before we break this down, please note that when ordering the PCBs you will ONLY NEED TO POPULATE the USB-C connectors. To be clear,* ***we will provide you with the pin headers and JST connectors*** *described in the power subsystem section. 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 $35 to have your board manufactured. That is 5x PCBs made @ $4, 2x PCBs populated @ $9.50 and then your component costs.

$13.5 is being used to manufacture a standard sized simple board. So, you have $(35*2) – $13.5 worth of components on the sum of your board section. $56.5 on components. Pay attention to extended parts…

Just explaining it one more time:

| Item | Student 1 | Student 2 |
|---|---|---|
| Starting Allowance | $35 | $35 |
| PCB Manufacture (5 units) | $4 | |
| PCB Assembly (2 units) | $9.50 | |
| Remaining Budget for components | $56.5 | |
| **OR** alternatively | Approx. $28.25 for components for your section | Approx. $28.25 for components for your section |
| **NOTE:** | If your board is ANYTHING over $70, it cannot be ordered. | |

*Table 5 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

### *General Notes*

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 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 DO NOT POPULATE the 2x16 connector nor the battery connector. 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.

## What is expected

1. In your pairs, you are required to evenly divide up the 8 tasks/requirements. Take some time to understand each task and rank them amongst yourselves based on difficulty. Note that these are the tasks that you will discuss in the design sections of your Interim Design Report.
2. You will need to submit
    a. Production files (Gerber, BOM and PnP files for a single PCB) in your pairs by the due date.
    b. an Interim Design Report in which you will document all the design decisions that you have made and your solution to the problem.
    c. Demonstrate your board
    d. A Final Design Report
3. You will need to check this document frequently and redownload it when more information becomes available.

# 4a Project Assignment: PCB Production Files

## Instructions

You will need to submit all the production files for your PCB on Amathuba by the **28th of March 2025 at 5pm**. Late submissions will not be accepted as the boards will not arrive on time for you to complete the course.

You need to submit a single zipped folder with the following naming convention: ***EEE3088F 2025 Project PCB Files Group xx STUDENTNUMBER1 STUDENTNUMBER2.zip*** e.g. EEE3088F Project PCB Files Group 10 FRRTRI001 XYZMNO009.zip. The zipped folder must contain:

1. A zipped file containing your Gerbers. **Please write your student numbers and group number on the silkscreen of your PCB**
2. The Bill of Materials (BOM) .csv file
3. The POS/CPL .csv file
4. A screenshot of JLCPCB BOM page after you have successfully uploaded all your files.
5. A screenshot of the JLCPCB add to cart page after you have successfully uploaded your files. With the naming convention

## More Information

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 JLCPCB 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. JLC add to cart page.

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

- ***EEE3088F_Project_JLCPCBBOM_studentnumber***
- ***EEE3088F_Project_JLCPCBacceptance_studentnumber***

These screenshots will serve as proof of your ability to submit the files as if you were going to place the order on JLCPCB yourself.

**Notes**

1. Please write your student numbers and group number on the silkscreen of your PCB
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**.

# 4b Project Assignment: 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:

- This report is an individual submission.
- 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. This is important for the markers.
- Marks will be allocated to report presentation (grammar, formatting, use of space, etc.). General presentation remarks:
    - ***Do not write in first person i.e. using I, we, us, they, them, him, her, etc.***
    - Include a title page, the plagiarism declaration and table of contents. If you have citations, include a Bibliography (remove this where it is not used).
    - Ensure that table headings are above the table and that figure headings are below the figure (IEEE standard).
- You will be penalised for not assigning pages on Gradescope.
- You will be penalised for exceeding the page limit. **The 12-page page limit includes the pages used to add in figures and tables.** The limit does not include the cover page, plagiarism declaration, table of contents or the bibliography.
- You will be penalised for overlapping discussions with your group member in certain sections. Pay attention to the expectations of the report.

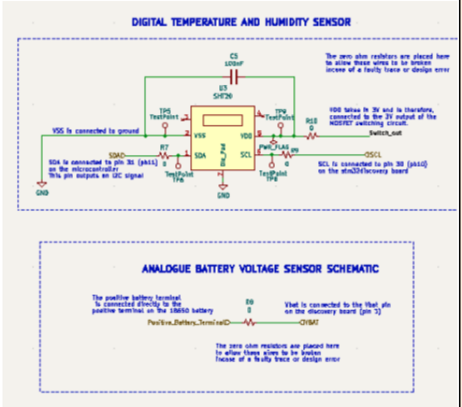
## Report Expectations

You are expected to write this report individually. In some of the sections you are specifically required to highlight what work you have done on the design of the solution. Please pay special attention to where these sections are.

The table below highlights the expectations for the Interim Design Report.

| Report structure | | Description | Notes | Page limit |
| --- | --- | --- | --- | --- |
| Chapter | Section | | | |
| Introduction | Problem Description | Holistically summarise the project problem:<br>- What is the context?<br>- What is the greater project?<br>- What is your specific problem in the context of the greater problem? | Written about overall power subsystem. | 1 |
| | Scope and limitations | - Highlight exactly what your project entails and what it does not entail.<br>- Highlight what limits the project designs, testing, development, etc. | Written about overall power subsystem. | |
| | GitHub Link | Provide the link to your group's shared repository. The files should be organised logically and there should be a repo README.<br>*Please make sure that the link is correct, and that the repository is publicly visible at the time of submission.* | Written about overall power subsystem. | |
| Requirements Analysis | Traceability Analysis | - Create your own version / complete the table in the template to show how the requirements, specifications and testing procedures all link. Use the IDs to show this. **You are only expected to fill in the ATP column for the 5 ATPs that you have completed in your report but must still show all requirements and specifications.**<br><br>- Write a description for each completed row in the matrix **(the rows where you have assigned an ATP)**:<br>  ○ Briefly 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.*<br>  ○ The idea here is to see your critical thinking about why and how you derived the specifications and tests in this section. | Each group member should cover different parts of the design. No overlapping tests will be accepted. | 5 |
| | Requirements | Fill the template /create a requirements table, give it a meaningful heading. Make sure to give each requirement a unique ID. | Written about overall power subsystem. | |

| | | | | |
|---|---|---|---|---|
| | Specifications | Fill the template /create a specifications table, give it a meaningful heading. Make sure to give each specification a unique ID. | Each group member should only cover the specifications for their parts design. This is to save space. | |
| | Acceptance Testing Procedures | Give 5 tests that you will conduct on your board to verify that your system meets some of the requirements and specifications. 5 tests are not comprehensive enough to test for all reqs and specs, rather you are tasked to show thinking around testing some parts of your system. To do this, create your own version / complete the table in the template:<br>- Provide each test with a unique ID<br>- What is the test? Why is it being tested?<br>- Concisely explain how you will test (using jumpers, test points, what equipment, etc.). Explain the procedure.<br>- What is the pass/fail criteria be specific, general statements such as "turns on" are incorrect.<br><br>*Notes:*<br>1. *Please note that each requirement and specification could have more than one ATP associated with it* ***BUT one ATP CANNOT be associated with multiple specifications or requirements.***<br>2. *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 measures you took to prevent failure on the board. Rather, imagine that you now have the board, describe how you will prove that you met the requirements and specifications.* | Each group member should cover different parts of the design. No overlapping tests will be accepted. | |
| Subsystem Design | Design Decisions | Discuss 3 unique design decisions you have made:<br>- *To ensure that your report flows, these need to link back to the specifications and requirements that you chose ATPs for.*<br>- Comparison of passive component values alone will not be considered as design decisions.<br>- For each design decision: | Each group member should cover different parts of the design. No overlapping design decisions will be accepted. These decisions should link to | 5 |

| | | | | |
|---|---|---|---|---|
| | | ○ Briefly make it clear why this decision is necessary (i.e. link to the reqs/specs)<br>○ Evidence of options considered as well as thoughtful and relevant comparisons (using tables, diagrams, calculations, etc.)<br>○ **Clearly state the final decision and why.** It is good practice to link back to the specifications/requirements. | the ATPs you have previously described. | |
| | Final Design Description Summary | 1. Write a summary of your final design: briefly describe your solution to the problem. Make sure to highlight key features of your design that help solve the problem. Do not be verbose, it is only a summary.<br><br>2. Provide a screenshot of your final schematic (this should only include the requirements and specifications that you designed solutions for). Make sure to:<br>- Include the title block with version number, author name, date, schematic title.<br>- Include comments explaining what different parts of the schematic are for and do.<br>- Neaten up the schematic so that it is easy to follow.<br>- All labels used should be clear and describe the connection well e.g. PA7_VADC<br><br><br>3. **Provide screenshots of your group's PCB** (front and back and then one 3D viewer screenshot). | Each group member should cover different parts of the design. No overlapping will be accepted. | |
| | Failure Management | - List/Tabulate at least 3 plausible and sensible failure management processes taken and why (<u>**concisely**</u>)<br>- Make sure that you give each a clear descriptive name that describes what it is. | Each group member should cover only the parts they applied to their section of the design. No overlapping will be accepted. | |

<table>
<tr><td></td><td></td><td>This is specifically with regards to post-componen- selectio- design phase (how you add measures to fix something on your PCB when it has arrived). These are pre-emptive steps you have taken to ensure that if something in your circuitry is broken/incorrectly connected/etc, you are still able to fix it once the board arrives. 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><td></td></tr>
<tr><td></td><td>Micro-mouse System Interfacing</td><td>
<ul>
<li>Replace the template table with your own table that clearly shows which pins connect to and interact with other parts of the system (specifically the motherboard interface). An example is given below.</li>
<li>Include a simple, high level (do not draw a circuit diagram or go into detail) interfacing diagram showing how your subsystem fits into the larger micro-mouse system. This should include information about expected inputs/outputs and/or what the interactions are and/or pin connections.</li>
</ul>
The table and diagram should describe and display the same information. The idea here is to show you two different ways in which interfacing information can be conveyed.<br>
Table 3.2: Interfacing specifications
<table>
<tr><th>Interface</th><th>Description</th><th>Pins/Output</th></tr>
<tr><td>I001</td><td>SparkFun 9DoF IMU Breakout board to STM for data transfer (SPI)</td><td>• MISO: Breakout MISO* to STM PB14<br>• MOSI: Breakout MOSI* to STM PB15<br>• SCLK: Breakout SCLK* to STM PB13<br>• CS: Breakout CS* to STM PB12 (GPIO)<br>• Power: Breakout 1V8-5V5* to STM 3V<br>• GND: Breakout GND* to STM GND</td></tr>
<tr><td>I002</td><td>STM to FTDI for data transfer to computer (UART)</td><td>• STM PA2(RX) to FTDI RXD<br>• STM PA3(TX) to FTDI TXD</td></tr>
<tr><td>I003</td><td>FTDI to computer for data transfer from STM</td><td>• Assuming the FTDI is connected to a Micro B USB port (which can be used to connect to a USB cable to connect to a computer), output via cable</td></tr>
</table>
</td><td>Written about overall power subsystem.</td><td></td></tr>
<tr><td>Results and Discussion</td><td></td><td>This will only be completed for the final report.</td><td></td><td>0</td></tr>
<tr><td>Recommendations</td><td></td><td>This will only be completed for the final report.</td><td></td><td>0</td></tr>
<tr><td colspan="3">Extra page for formatting.</td><td></td><td>1</td></tr>
</table>

# 4c Project Assignment: Final Demonstration

## Demonstration Instructions

- You are required to bring a **printed copy of your schematic** – you will not be able to give justification for marks without this.
- You are required to bring all 5 of your PCB boards to your final demonstration
- An automarker will be used to assess the functionality of your PCB (more on this later in the section). If you are unable to comply with the automarker, you will need to demonstrate all the marks to your marker using one of the benches in White Lab – think of this like a practical demonstration in one of your other courses. You will have 3 attempts to demonstrate and only the marks awarded on your *final demonstration* will be recorded for the demonstration.
- Your *final demonstration* is your 3rd demonstration or any time before that (you must indicate this to the marker).
- **On your final demonstration, you will need to place both group member's student cards on the testing jig (or on the White Lab bench next to your PCB), take a picture and upload it to the Google Form using the QR code at the demonstration table. Failing to do this will result in an automatic 0, as this your only proof of demonstration.**

## 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. It facilitates separate and repeatable testing of different functionality of your PCB, namely battery charging, motor control, switching ON and OFF, USB C and SOC measurements.

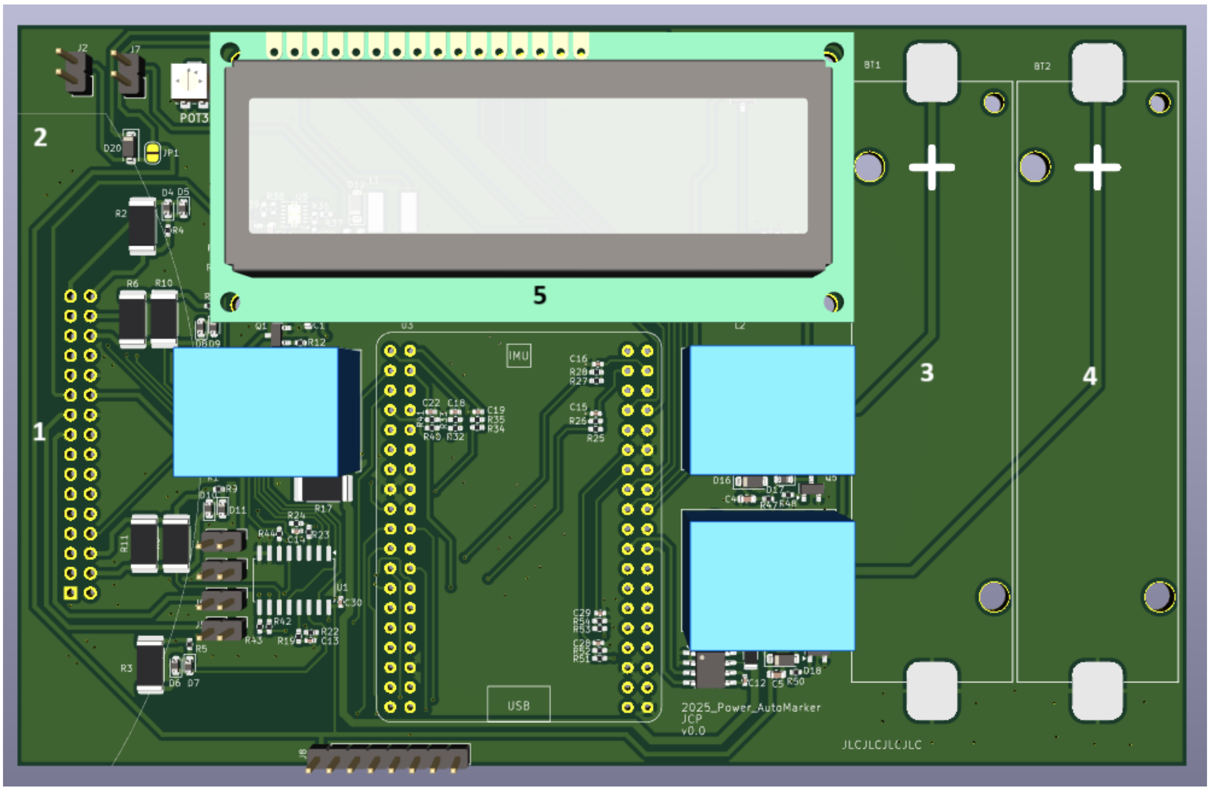



*Figure 10 The power subsystem testing jig*

5 key noteworthy elements have been highlighted in the image above:

1. This is the connection point for your board. Please note that your board will be placed upside down on the testing jig.
2. The silkscreen curve is the outline of the micro-mouse platform, and your design should not exceed the mouse dimensions.
3. Battery 1 holder
4. Battery 2 holder – The batteries are located off your PCB to facilitate speed of connection and allow for monitoring on the testing PCB.
5. LCD screen displaying the progress of the overall rig status

### *The Procedure*

The following is expected of you at the demonstration:

1. Plug your power PCB into the testing jig.
2. Each component of your board will be tested (as per the rubric)
3. The auto-marking procedure will be:

   0. Attempt Connection to I2C sensor

   1. With the device OFF, measure input current from our Battery input to the board.

   2. With the switch set to on, confirm 5V and 3V3 present

   3. Load Both 5V and 3V3 and confirm regulation

   4. Plug in USB C and confirm 9V present

   5. Unplug USB C

   6. Provide our own 9V

   7. Confirm slow charge on battery

   8. Confirm fast charge on battery

   9. Test Ext Load drives

   10. Test H Bridges

### *The Rubric (PROVISIONAL)*

Please note that this rubric might change before the demonstrations take place. This is a guideline for now to help you understand how the assessment will take place.

<table>
<tr><td colspan="2">31</td><td colspan="3">Total Marks</td></tr>
<tr><td rowspan="5">3</td><td>+0.5</td><td rowspan="5">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 micro-mouse (size, shape, etc.)</td></tr>
<tr><td>+0.5</td><td colspan="2">Correct connectors have been used - board would fit onto the micro-mouse</td></tr>
<tr><td>+1</td><td colspan="2">Group Identifier on the board</td></tr>
<tr><td rowspan="2">28</td><td>+1</td><td rowspan="2">Function: Power</td><td rowspan="2">H-bridge</td><td>Off-State: Current <1mA</td></tr>
<tr><td>+1</td><td>On-State: Voltage > 95% Supply Voltage.</td></tr>
</table>

| | | | | |
|---|---|---|---|---|
| | +1 | | | Low current H Bridges operate correctly |
| | +1 | | | Higher current H Bridges operate correctly |
| | +1 | | Switch | Switch can handle 2A or student has a solution to avoid needing this. *[Look at schematic or Justification- show calculations – Tutor Assessed]* |
| | +1 | | | Current in off state <30uA |
| | +1 | | Monitoring - on I2C | I2C operational |
| | +1 | | | Battery voltage correctly measured |
| | +1 | | | Sense Resistor appropriately chosen and current measurement possible *– Tutor Assessed*<br>Size >= 0603<br>Value <= X |
| | +1 | | Charge Battery | Does the Batt pin go to 4.2V |
| | +1 | | | Battery could be connected to the board (correct header) *– Tutor Assessed* |
| | +1 | | | Low Current Charging - approximately 200mA. |
| | +1 | | | High Current Charging - approximately 600mA. |
| | +1 | | External Load Capabilities | External Load can be switched (High Side) |
| | +1 | | | Current Requirements can be achieved at 90% rated voltage |
| | +1 | | 5V Out | 5V present when switch in ON state |
| | +1 | | | 5V NOT present when switch in OFF state |
| | +3 | | | 5V to rated power 90% Voltage (30, 60, 100% I) |
| | +1 | | 3V3 Out | 3V3 present when switch in ON state |
| | +1 | | | 3V3 NOT present when switch in OFF state |
| | +3 | | | 3V3 to rated power 90% Voltage (30, 60, 100% I) |
| | +1 | | USB C | Connector on the Board – *Tutor Assessed* |
| | +1 | | | 9V present when plugged in |
| | +1 | | | Battery Charging when power supplied |

*Table 6 Final Demonstration Rubric: Power*

# 4d Project Assignment: Final Report

For this assignment, use of the LaTeX report template that was used in the Interim Design Report. You will have already completed some of the Final Report sections in the Interim Design Report and received some feedback on them but, please note that the Final Report has some updates, additional chapters and sections to be completed.

## Notes:

- This report is an individual submission.
- You must submit to both Amathuba and Gradescope by the deadline. You will be penalised for not submitting to both platforms.
- Please use the following naming convention: ***EEE3088F 2025 Final Project Report STUDNUM.pdf*** **e.g.** ***EEE3088F 2025 Project Report FRRTRI001.pdf***
- Please make sure that your chapter names and section names match the ones in the table below. When uploading to Gradescope, the rubric will expect you to have the section names described below. This is important for the markers. Feel free to add in subsection headings when writing the report.
- Marks will be allocated to report presentation (grammar, formatting, use of space, etc.). General presentation remarks:
    - ***Do not write in first person i.e. using I, we, us, they, them, him, her, etc.***
    - Include a title page, the plagiarism declaration and table of contents. If you have citations, include a Bibliography (remove this where it is not used).
    - Ensure that table headings are above the table and that figure headings are below the figure (IEEE standard).
- You will be penalised for not assigning pages on Gradescope.
- You will be penalised for exceeding the page limit. **The 16-page page limit includes the pages used to add in figures and tables.** The limit does not include the cover page, plagiarism declaration, table of contents or the bibliography.
- **You will be penalised for overlapping discussions with your group member in certain sections. Pay attention to the expectations of the report.**

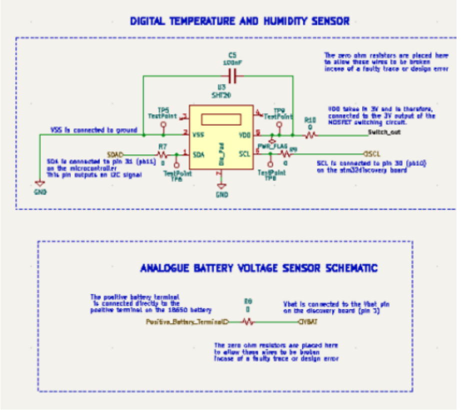
## Report Expectations

You are expected to write this report individually. In some of the sections you are specifically required to highlight what work you have done on the design of the solution. Please pay special attention to where these sections are.

The table below highlights the expectations for the Interim Design Report.

| Report structure | | Description | Notes | Page limit |
|---|---|---|---|---|
| Chapter | Section | | | |
| Introduction | Problem Description | Holistically summarise the project problem:<br>- What is the context?<br>- What is the greater project?<br>- What is your specific problem in the context of the greater problem? | Written about overall power subsystem. | 1 |
| | Scope and limitations | - Highlight exactly what your project entails and what it does not entail.<br>- Highlight what limits the project designs, testing, development, etc. | Written about overall power subsystem. / Each group member's scope and limitations. | |
| | GitHub Link | Provide the link to your group's shared repository. The files should be organised logically and there should be a repo README.<br>*Please make sure that the link is correct, and that the repository is publicly visible at the time of submission.* | Written about overall power subsystem. | |
| Requirements Analysis | Traceability Analysis | - Create your own version / complete the table in the template to show how the requirements, specifications and testing procedures all link. Use the IDs to show this. **You are only expected to fill in the ATP column for the 5 ATPs that you have completed in your report but must still show all requirements and specifications.**<br><br>- Write a description for each completed row in the matrix **(the rows where you have assigned an ATP)**:<br>o Briefly 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.*<br>o The idea here is to see your critical thinking about why and how you derived the specifications and tests in this section. | Each group member should cover different parts of the design. No overlapping tests will be accepted. | 3 |
| | Requirements | Fill the template /create a requirements table, give it a meaningful heading. Make sure to give each requirement a unique ID. | Written about overall power subsystem. | |

<table>
<tr><td></td><td>Specifications</td><td>Fill the template /create a specifications table, give it a meaningful heading. Make sure to give each specification a unique ID.</td><td>Each group member should only cover the specifications for their parts design. This is to save space.</td><td></td></tr>
<tr><td></td><td>Acceptance Testing Procedures</td><td>Give 5 tests that you will conduct on your board to verify that your system meets some of the requirements and specifications. 5 tests are not comprehensive enough to test for all reqs and specs, rather you are tasked to show thinking around testing some parts of your system. To do this, create your own version / complete the table in the template:<br>- Provide each test with a unique ID<br>- What is the test? Why is it being tested?<br>- Concisely explain how you will test (using jumpers, test points, what equipment, etc.). Explain the procedure.<br>- What is the pass/fail criteria be specific, general statements such as “turns on” are incorrect.<br><br>Notes:<br>1. Please note that each requirement and specification could have more than one ATP associated with it BUT one ATP CANNOT be associated with multiple specifications or requirements.<br>2. 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 measures you took to prevent failure on the board. Rather, imagine that you now have the board, describe how you will prove that you met the requirements and specifications.</td><td>Each group member should cover different parts of the design. No overlapping tests will be accepted.</td><td></td></tr>
<tr><td>Subsystem Design</td><td>Design Decisions</td><td>Discuss 3 unique design decisions you have made:<br>- To ensure that your report flows, these need to link back to the specifications and requirements that you chose ATPs for.<br>- Comparison of passive component values alone will not be considered as design decisions.<br>- For each design decision:</td><td>Each group member should cover different parts of the design. No overlapping design decisions will be accepted. These decisions should link</td><td>5</td></tr>
</table>

| | | | | |
|---|---|---|---|---|
| | | ○ Briefly make it clear why this decision is necessary (i.e. link to the reqs/specs)<br>○ Evidence of options considered as well as thoughtful and relevant comparisons (using tables, diagrams, calculations, etc.)<br>○ **Clearly state the final decision and why.** It is good practice to link back to the specifications/requirements. | to the ATPs you have previously described. | |
| | Final Design Description Summary | 1. Write a summary of your final design: briefly describe your solution to the problem. Make sure to highlight key features of your design that help solve the problem. Do not be verbose, it is only a summary.<br><br>2. Provide a screenshot of your final schematic (this should only include the requirements and specifications that you designed solutions for). Make sure to:<br>- Include the title block with version number, author name, date, schematic title.<br>- Include comments explaining what different parts of the schematic are for and do.<br>- Neaten up the schematic so that it is easy to follow.<br>- All labels used should be clear and describe the connection well e.g. PA7_VADC<br><br><br>3. Bill of Materials<br>4. **Provide screenshots of your group's PCB** (front and back and then one 3D viewer screenshot). | Each group member should cover different parts of the design. No overlapping will be accepted. | |
| | Failure Management | - List/Tabulate at least 3 plausible and sensible failure management processes taken and why (**<u>concisely</u>**)<br>- Make sure that you give each a clear descriptive name that describes what it is. | Each group member should cover only the parts they applied to their section of the | |

<table>
<tr><td></td><td></td><td>This is specifically with regards to post-design phase (how you add measures to fix something on your PCB when it has arrived). These are pre-emptive steps you have taken to ensure that if something in your circuitry is broken/incorrectly connected/etc, you are still able to fix it once the board arrives. This is where you discuss how you designed the PCB to mitigate component failures/trace issues/errors in your circuit design/etc.</td><td>design. No overlapping will be accepted.</td><td></td></tr>
<tr><td></td><td>Micro-mouse System Interfacing</td><td>- Replace the template table with your own table that clearly shows which pins connect to and interact with other parts of the system (specifically the motherboard interface). An example is given below.<br>- Include a simple, high level (do not draw a circuit diagram or go into detail) interfacing diagram showing how your subsystem fits into the larger micro-mouse system. This should include information about expected inputs/outputs and/or what the interactions are and/or pin connections.<br><br>The table and diagram should describe and display the same information. The idea here is to show you two different ways in which interfacing information can be conveyed.<br><br>Table 3.2: Interfacing specifications<br>
<table>
<tr><th>Interface</th><th>Description</th><th>Pins/Output</th></tr>
<tr><td>I001</td><td>SparkFun 9DoF IMU Breakout board to STM for data transfer (SPI)</td><td>• MISO: Breakout MISO* to STM PB14<br>• MOSI: Breakout MOSI* to STM PB15<br>• SCLK: Breakout SCLK* to STM PB13<br>• CS: Breakout CS* to STM PB12 (GPIO)<br>• Power: Breakout 1V8-5V5* to STM 3V<br>• GND: Breakout GND* to STM GND</td></tr>
<tr><td>I002</td><td>STM to FTDI for data transfer to computer (UART)</td><td>• STM PA2(RX) to FTDI RXD<br>• STM PA3(TX) to FTDI TXD</td></tr>
<tr><td>I003</td><td>FTDI to computer for data transfer from STM</td><td>• Assuming the FTDI is connected to a Micro B USB port (which can be used to connect to a USB cable to connect to a computer), output via cable</td></tr>
</table>
</td><td>Written about overall power subsystem.</td><td></td></tr>
<tr><td colspan="2">Results and Discussion</td><td>Critical Analysis of Testing (discuss your 5 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 “AT01 was passed or AT03 was failed”.<br><br>You will be marked based on how thorough you are and how well you structure this chapter.</td><td>Each group member should cover only their tests and the parts of the design that are theirs. No overlapping will be accepted.</td><td>3</td></tr>
</table>

| | | | |
|---|---|---|---|
| Conclusion | You will be marked based on the following criteria and how well you structure it:<br>• Are the results interpreted clearly?<br>• Are the conclusions reached clearly supported in the results and linked to the research question/engineering problem/specifications and requirements?<br>• Are suggestions made for further work? (emphasis is placed on the quality of these suggestions) | **Each group member should highlight their own conclusion within the greater group project. Report only on content covered in your own report.**<br><br>**Note: there still needs to be a clear link back to the problem you solved as a group.** | 1 |
| Reflection | This chapter does not appear in this form in a conventional academic report; however, it is an important part of any process. In this chapter, please reflect on the project by answering the following questions:<br>1. What would you do differently in each phase of the design (design, board layout, JLC design files, testing, etc)?<br>2. Provide a post-facto Gantt chart of time spent on each task and task inter-dependencies that would give another student insight into how to manage time on a similar design project.<br>3. How would you bring down the cost of your design?<br>4. How would you deal with trying to go in a straight line if the two motors have different parameters?<br>5. What happens if the motor terminals are short circuited (or if the motor stalls)?<br><br>*Please pay important attention to structuring this so that it still coherently fits into the rest of your report (this is important for presentation marks).* | Each group member should answer this individually. | 1 |
| **Extra page for formatting.** | | | **2** |

# 5 Appendix: Errata

Most recent changes are in Orange.

Version 2 –

Modified the Requirements for the PCB to be more specific.

Highlighted more mechanical information on the PCB connection to the motherboard on note 5.

Modified the pinout table to be more specific on which pins are responsible for functioning on the Power board.

Version 3 –

Changed the HV on the table to High Voltage (9V)

Version 4 –

Updated the Interim Design Report: Specifications.

**

Version 6 –

Added in the Final Report break down.

Added in PROVISIONAL Final Demonstration rubric