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 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 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.*
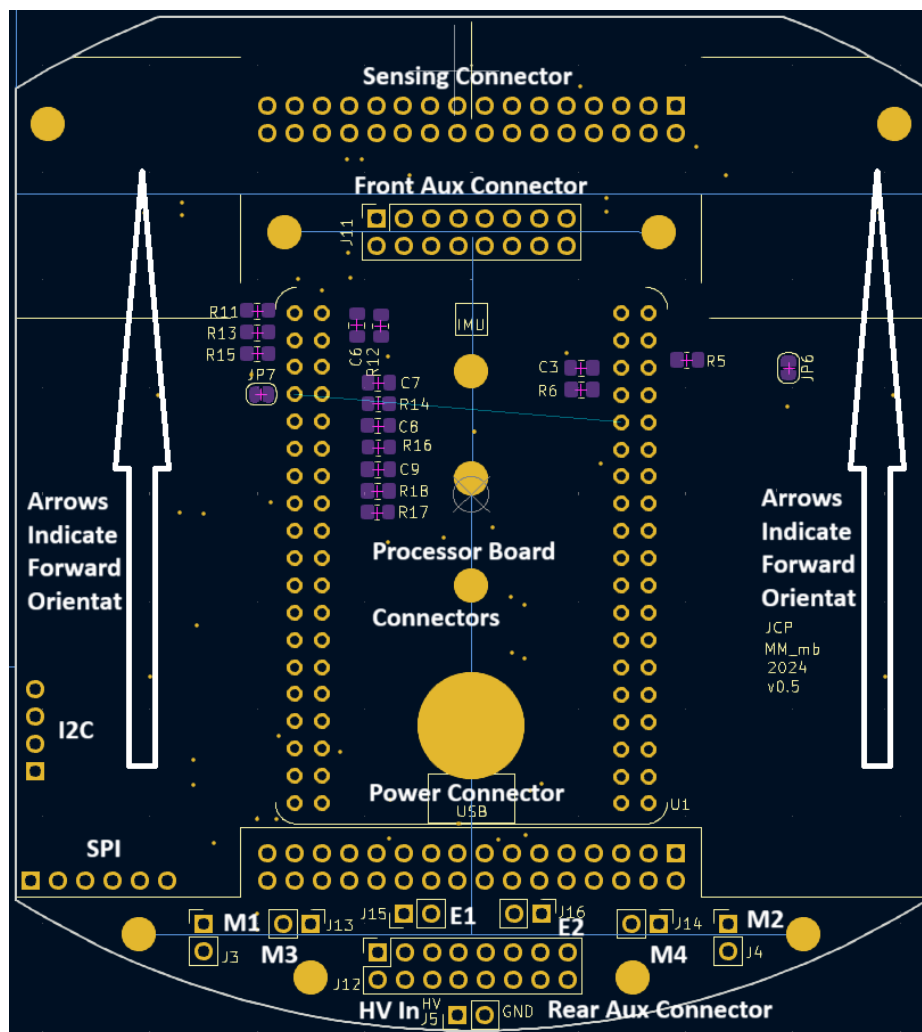

Figure 1 Pin-view of the MM Motherboard PCB.
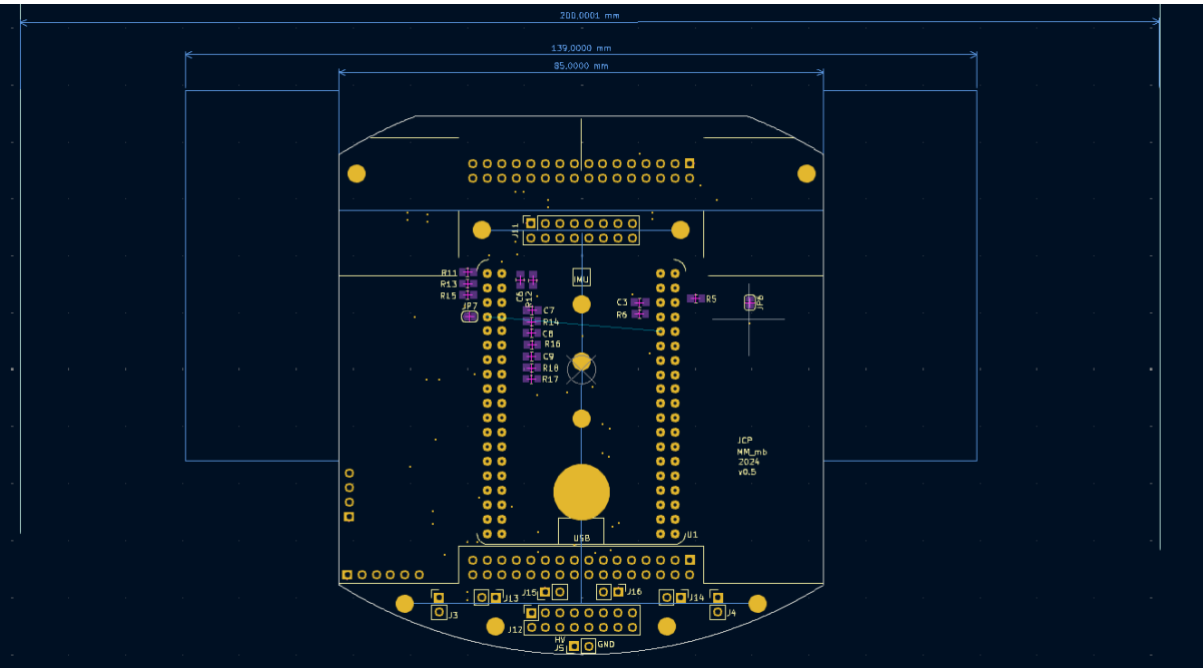
Figure 2 Dimensions of MM

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	

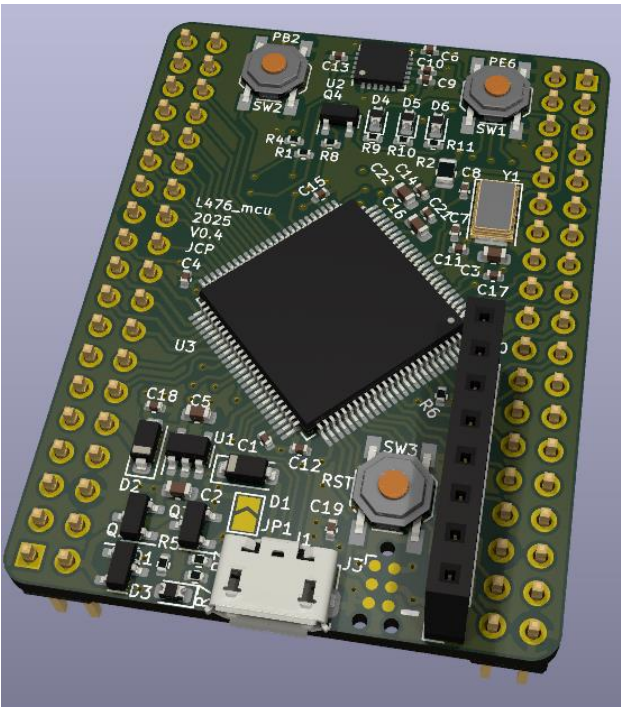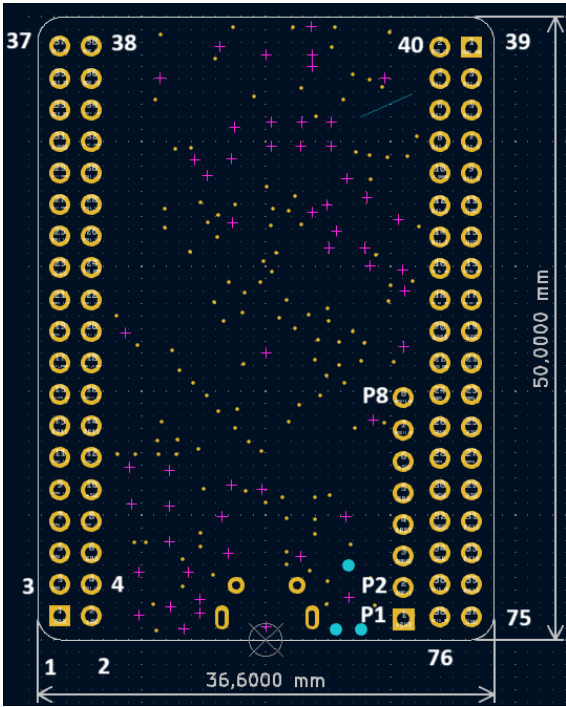|					
6	GND		48	3V3_ADC	
7	PC6		49	GND	
8	PD15		50	PC1	
9	PD14		51	PE1	
10	5V		52	PC0	
11	PD13		53	PE4	
12	PD12		54	PE5	
13	PD11		55	PE2	
14	PD10		56	PE3	
15	PD9		57	PE0	
16	PD8		58	NRST	
17	PB15		59	PB8	
18	PB14		60	PB9	
19	PB13		61	PB7	
20	PB12		62	PB6	
21	PB11		63	PD7	
22	PB10		64	PB4	
23	GND		65	PD5	
24	PE15		66	PD6	
25	PE14		67	PD3	
26	PE13		68	PD4	
27	PE12		69	PC11	
28	PE11		70	PD2	
29	PE10		71	PC10	
30	PE9		72	GND	
31	3V3	TLV740P	73	PA15	
32	PE8		74	PD1	
33	PB1		75	GND	
34	PE7		76	PD0	
35	PB0		USB Micro Connector	Standard USB Connector	
36	PC5		Prog_3V3 (P1)	Target Voltage sense	
37	PC4				
38	PA7		Prog_ SWCLK (P2)	SWCLK	
39	PA5		Prog_ SWDIO (P3)	SWDIO	
40	PA6		Prog_ NRST (P4)	NRST	
41	PA3		Prog_ GND (P5)	GND	
42	PA4		Prog_ 5V (P6)	Programming Power	
43	PA1				
44	PA2		Prog_VCP_In (P7)	USART1_Tx	
45	PC3		Prog_VCP_Out (P8)	USART1_Rx	
46	PA0				
47	PC2				

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.
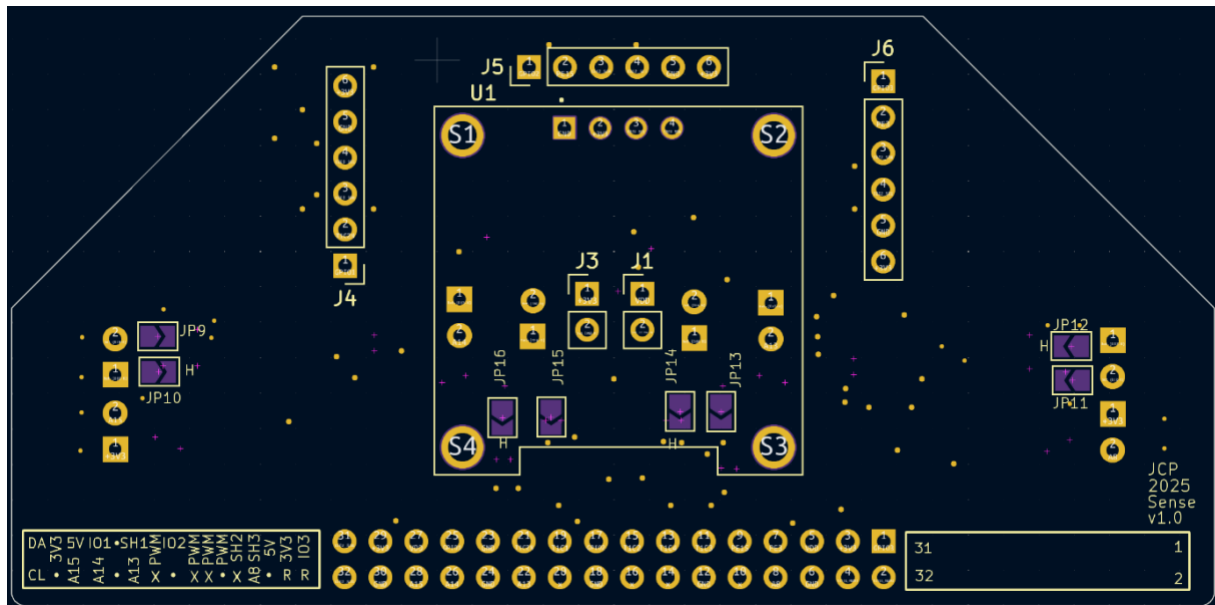

Figure 5 MM Sensing pinouts that plug in the MM Motherboard PCB

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

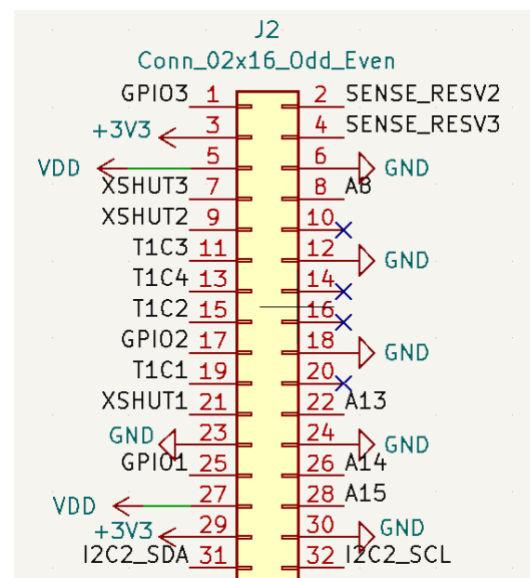| 24	GND	PB11
25	GPIO1	PE8
26	A14	PC5
27	5V	
28	A15	PB0
29	3V3	
30	GND	GND
31	SDA	PB11
32	SCL	PB10

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).
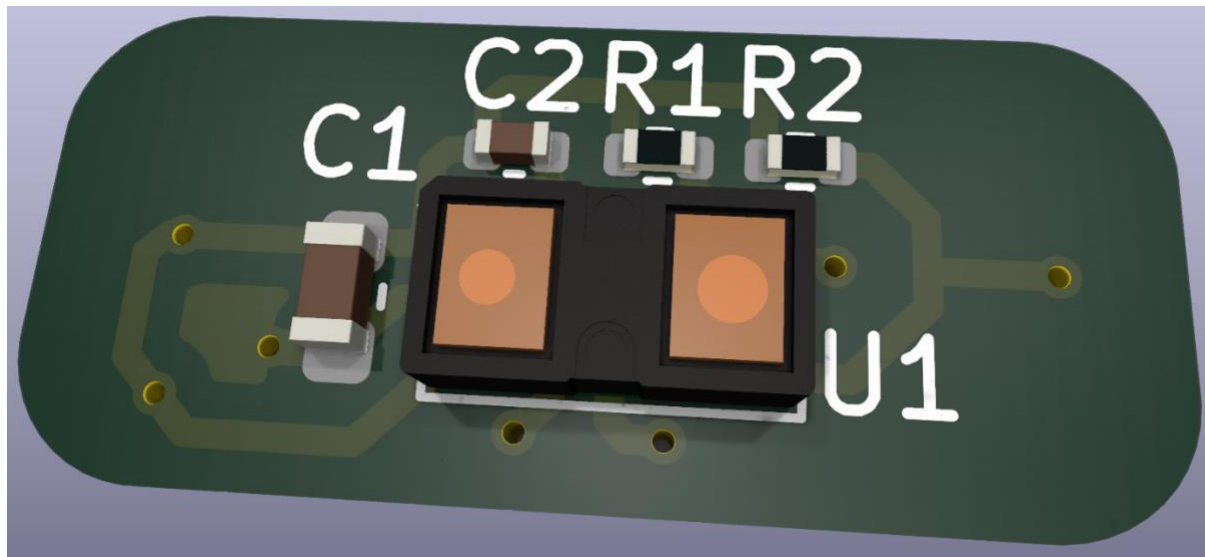
Figure 7 MM micro-sensing daughter board PCB

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:
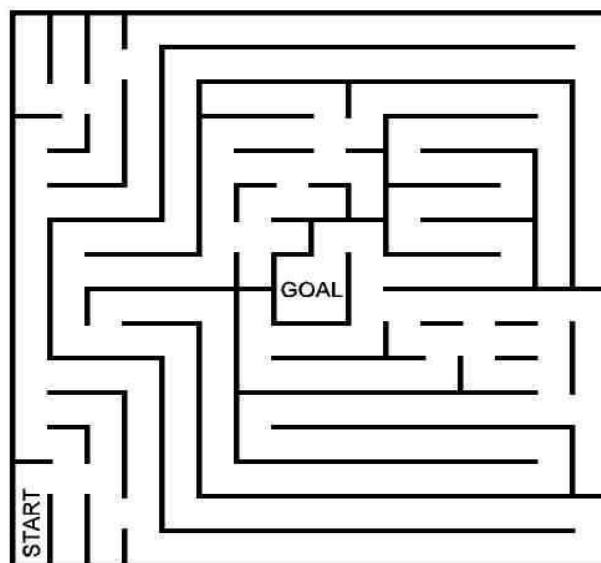
Figure 8 Example of a MM maze.

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 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 on the I2C Bus and configure it correctly with respect to the hardware
- 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
- Integrate USB C and get 9V out of the USB Host
- Provide 2x External Load Switching at 1A each
- Provide a 3V3 and 5V Out
- Provide an ON/OFF switch. **OFF state: battery draw <30uA. ON state: can provide your robot peak current.** 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 board needs to use them for securing your

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

PCB as the weight of the PCB will be too much for the connector alone.

The table below surmises the connections for the board

Connector	Pin Number	Associated to	Useful Info
	1		Motor2_B
	2		Motor2_A
	3		Motor4_A
	4	PD12	TIM4C1
	5		Motor4_B
	6	PD13	TIM4C2
	7	PD2	CTRL_EXT_LOAD2
	8	PD14	TIM4C3
	9	PD6	USART2_RX
	10	PD15	TIM4C4
	11		EXT_LOAD2
	12	5V	USART2_TX
	13	PB4	FAST_CHARGE
	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
	22	HV	HV
	23		GND
	24	PC6	TIM3C1
	25	PD11	CTRL_EXT_LOAD1
	26	PC7	TIM3C2
	27		MOTOR3_A
	28	PC8	TIM3C3
	29		MOTOR3_B
	30	PC9	TIM3C4
	31		MOTOR1_B
	32		MOTOR1_A
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

In your groups, you will need to submit all the production files for your single 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 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

Coming soon…

4c Project Assignment: Final Demonstration

Coming soon…

4d Project Assignment: Final Report

Coming soon…

5 Appendix: Errata

Version 2 –