Micro-mouse Project Brief

EE3088F 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, 22 March 2024 at 23:55 on Amathuba)
2. The *Interim Design Report* (Sunday, 14 April 2024 at 23:55 on Gradescope)
3. The *Lab demonstration* (Wednesday, 8 May 2024 during lab session in White Lab)
4. The *Final Report* (Friday, 17 May 2024 at 23:55 on Gradescope)

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.
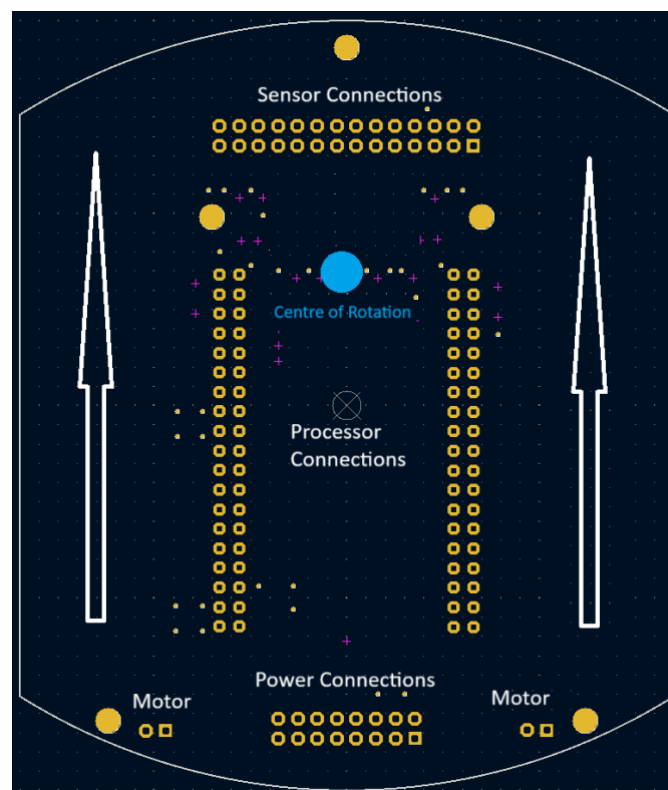

Figure 1 Pin-view of the MM Motherboard PCB.

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**.
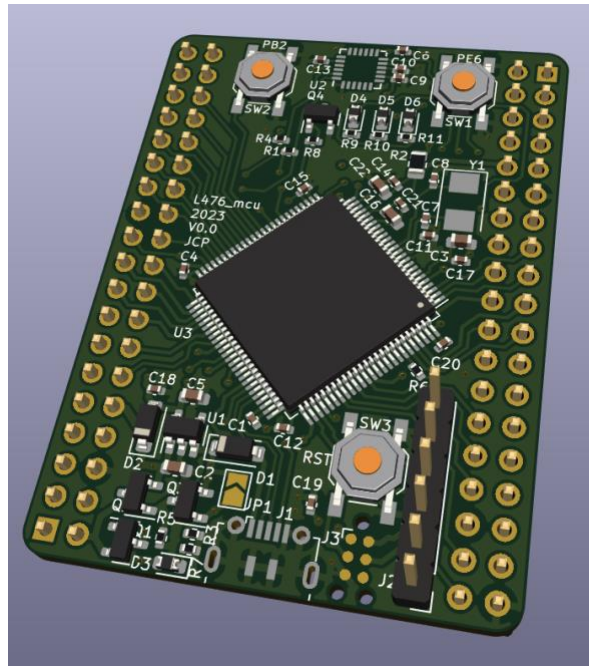

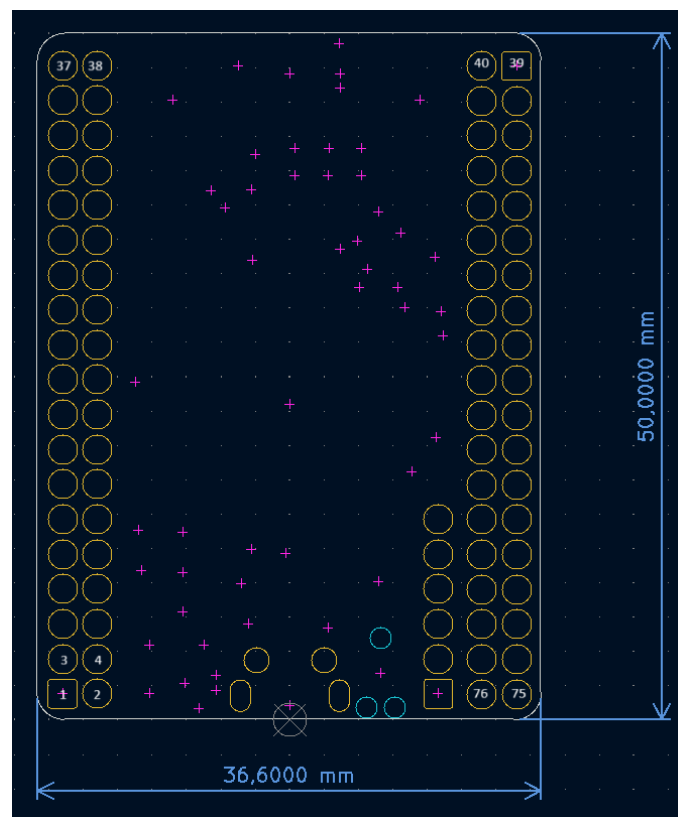*Figure 2 3D Render of the MM Processor PCB*

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	
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*).
- Needs an ON/OFF switch.

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.
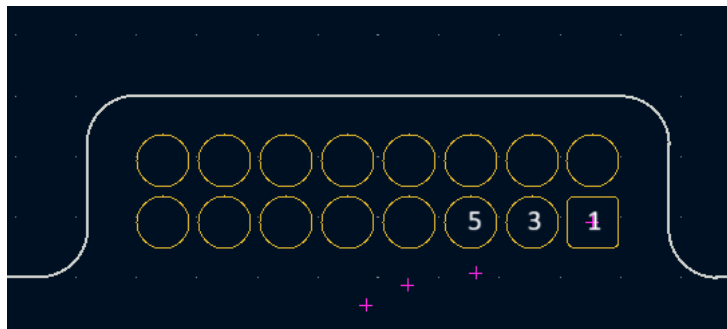2. A **2x8 (2.54mm pin pitch) pin header** as shown in the image below:

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
2x8 pin (2.54mm pin pitch) header	1	Motor2_A	
	2	PC6-9	PWM1
	3	Motor2_B	
	4	PC6-9	PWM2
	5	GND	
	6	GND	
	7	BATT	
	8	BATT	
	9	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.
- 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. 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.
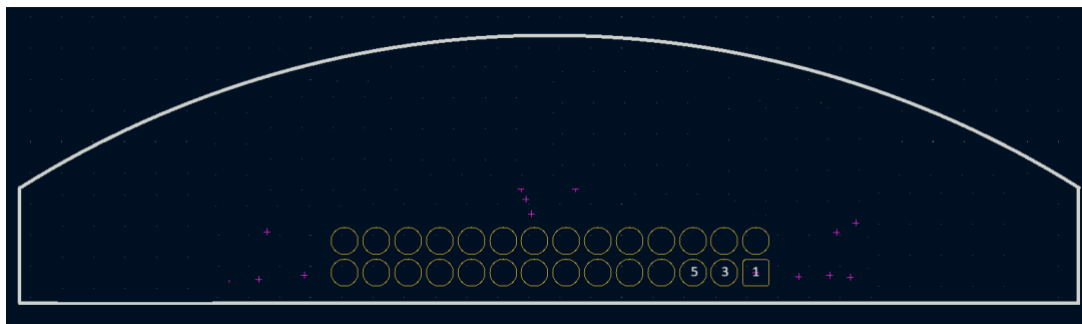
It will need to fit onto the 2x14 connection standard and be an appropriate size for the robot.

Pin Number	Associated to	Useful
1	RESV3	PC2
2	3V3	
3	GND	
4	BATT	PWM2
5	Analog8	
6	RESV2	PE1
7	Analog9	
8	PE15	PWM
9	GND	
10	PE14	PWM
11	Analog10	
12	PE13	PWM
13	Analog11	
14	PE12	PWM

Pin Number	Associated to	Useful
15	GND	
16	PE11	PWM
17	Analog12	
18	PE10	PWM
19	Analog13	
20	PE9	PWM
21	GND	
22	PE8	PWM
23	Analog14	
24	RESV1	PB11
25	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.

R360 / 20 = $18. $13.5 is being used to manufacture a standard sized simple board. So you have $4.5 worth of components on the sum of your board section. $2.25 each provided you have no extended parts.

Just explaining it one more time:

Item	Student 1 (Sensing)	Student 2 (Power)
Starting Allowance	$18	$18
PCB Manufacture (5 units)	$4	$4
PCB Assembly (2 units)	$9.50	$9.50
Remaining Budget for components	18 – 13.5 = **$4.5 for 2 boards**	18 – 13.5 = **$4.5 for 2 boards**
OR alternatively	$2.25 per board	$2.25 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. The extended component fee is not 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.
2. 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**.
3. 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.
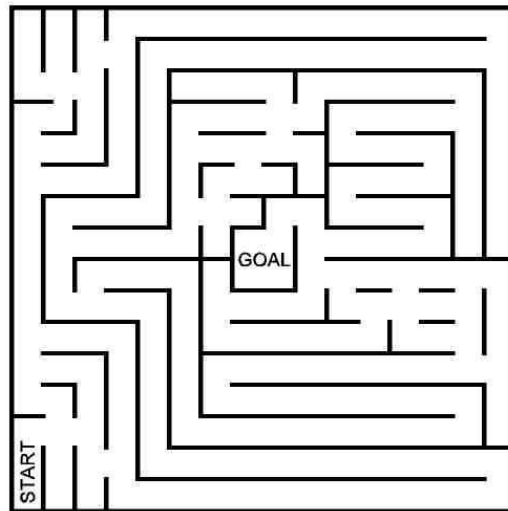
An example is provided below:

Figure 5 Example of a MM maze.

It can be seen that there are multiple paths to get to the goal.
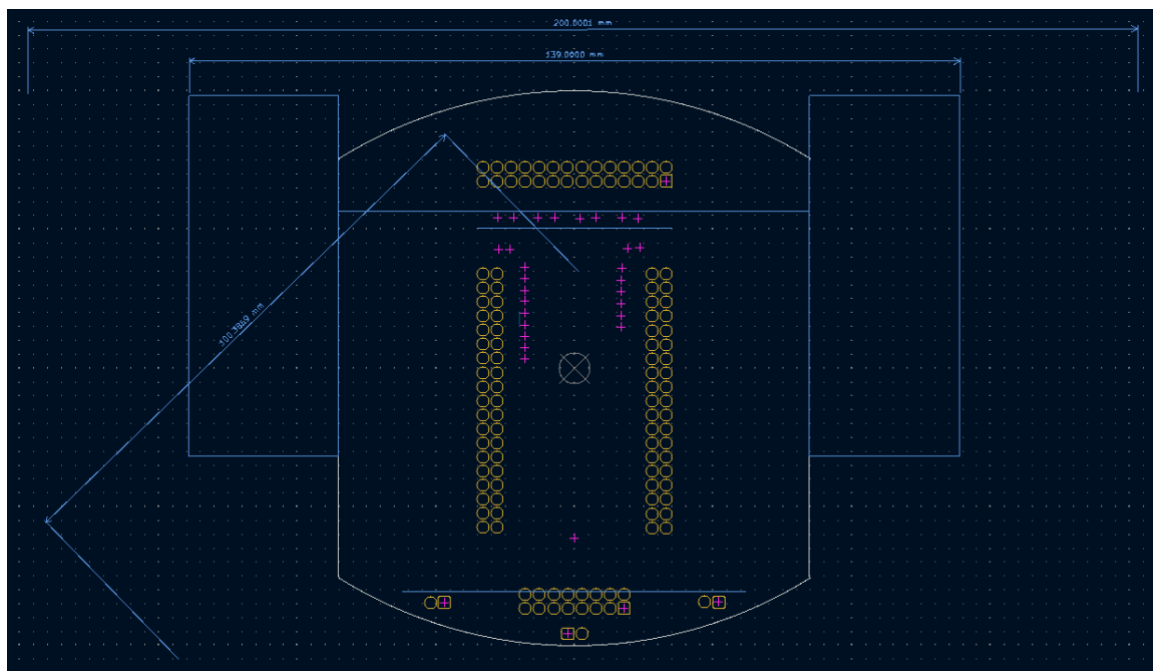
Other

Figure 6 Dimensions of MM

Gerber Files

You will need to submit all the production files for your PCB on Amathuba on the **22nd 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
1. BOM
2. final JCP acceptance pages.

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

<u>Notes</u>
1. 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.
2. 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.
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 – **you will not receive WL components if you did not populate them in your order**.

Interim Design Report

This section is under-construction…

But think about what you are trying to do for the project:

- **Your group's GitHub link where all your PCB files, datasheets, etc are stored.**
- What is the problem you are trying to solve?
- Specifications and requirements?
- How will you test your system? What have you added to your board to aid this? (things like jumpers and testing points)
- Design decisions: comparison of components for use in your circuit, why did you choose certain components over others?
- Your circuit diagrams: make sure to label the different parts and explain what each part of the circuit is for.
- Interfacing diagrams: How does your subsystem work with other subsystems?
- Interfacing table: How does your subsystem connect to other subsystems?
- Bill of materials – how much has your PCB costed.

Final Demonstration

This section is under-construction…

Final Report

This section is under-construction…