Interim Design Report

Micromouse Power Subsystem

Prepared by:
Nicholas McFarland
MCFNIC002

Prepared for:
EEE3088F
Department of Electrical Engineering
University of Cape Town

April 21, 2024

Declaration

1. I know that plagiarism is wrong. Plagiarism is to use another's work and pretend that it is one's own.
2. I have used the IEEE convention for citation and referencing. Each contribution to, and quotation in, this report from the work(s) of other people has been attributed, and has been cited and referenced.
3. This report is my own work.
4. I have not allowed, and will not allow, anyone to copy my work with the intention of passing it off as their own work or part thereof.

April 21, 2024

Nicholas McFarland

Date

Contents

Chapter 1

Introduction

1.1 Problem Description

The goal of this project is to build a micromouse that is capable of solving and navigating a maze. It consists of a sensing and power subsystem, and this report gives an in-depth analysis of the designed power subsystem, which will eventually be used to drive and power the micromouse. This subsystem must be capable of driving the motors used to control the wheels of the mouse and supplying the power used to drive the rest of the system.

1.2 Scope and Limitations

The designed circuit board uses a singular 1S1P battery to power all parts of its system and most external subsystems of the micro-mouse. It must be able to operate 2 motors at rated current and voltage and provide an analog connection that provides information to the processor regarding the battery's current voltage and charge state. It must be able to charge the battery from an external 5V input voltage and house an ON/OFF switch that can control the power supply to the system.

Besides the connections between the power subsystem's pin headers and motherboard, this project provides no detail into the other subsystems of the overall micromouse. It does not provide any coding required to operate the other subsystem/s.

The shape of the PCB design must feature a tab with dimensions that prevent it from obstructing the motor connections or colliding with any of the motherboard connections. The cost of producing the board may not exceed $30 when ordered, and all components selected for designing the system must be readily available on JLCPCB [1]. Finally, the design must be suitable for testing any potential problem points on the board and feature alternative options to prevent these problems.

1.3 GitHub Link

https://github.com/NichTucker/Micromouse

Chapter 2

Requirements Analysis

2.1 Requirements

The requirements for a micromouse power module are described in Table 2.1.

Table 2.1: User and functional requirements of the power subsystem.

Requirement ID	Description
SW01R	Requires switching between ON/OFF state
SW02R	When OFF, current drawn by battery $<$ 500uA
CHRG01R	Charge the battery from an input pin
CHRG02R	Recommended discharge at 0.5C (Capacity) for battery
BATT01R	Must include JST battery connector
BATT02R	Must be powered by Battery LiPo 800mAh 3.7V
MTR01R	Powers 2 motors
ADC01R	Provides information on the battery's current voltage and sense of state of charge (SoC)
PCB01R	16 available pin headers
PCB02R	Pin connector must be proud of the board
CST01R	Cost of the board must be less than $30

2.2 Specifications

The specifications, refined from the requirements in Table 2.1, for the micro-mouse power module are described in Table 2.2.

Table 2.2: Specifications of the sensing subsystem derived from the requirements in Table 2.1.

Specification ID	**Description**
SW01S	The switch must provide 2 discreet states of output voltage, ranging between 0V - Battery's Current voltage
SW02S	Quiescent currents for various components will be taken from the datasheet/calculated.
CHRG01S	Circuit uses an Li-Po battery charger with a 5V operating input voltage.
CHRG02S	Drawn current < 400mA when charging
BATT01S	JST pin connector with a pin pitch of 2mm
BATT02S	Battery will be provided by EEE3088F course
MTR01S	Circuit will use a dual bridge motor driver to power and operate the motors. Motors operate at 200mA each and 3.7V. Driver selected based on operating voltage.
ADC01S	Voltage divider circuit to provide batter's state of charge (SoC), as well as ensure the maximum input voltage < 3V3
PCB01S	2x8 pin header with 2.54mm pin pitch
PCB02S	Height of tab > 18mm and width of tab < 35mm to prevent collision with motors
CST01S	Starting allowance at $30, with manufacture costs of $4, Assembly cost of $9.50 leaving $8.25 per board for components

2.3 Testing Procedures

A summary of the testing procedures [2] detailed in chapter 4 is given in Table 2.3.

Table 2.3: Testing Procedures

Testing ID	Description
AT01	All ground connections correctly grounded
AT02	All resistor values match the design values
AT03	Switch turns system ON when pushed DOWN and OFF when RELEASED (personal preference)
AT04	Switch provides discreet ON/OFF states
AT05	Quiescent current less than maximum rated
AT06	Voltage provided by 5V pin is indeed 5V
AT07	Charger charges battery
AT08	Battery draws < 400mA when charging
AT09	Motors reach operating current
AT10	Motors reach operating voltage
AT11	Analog sub-circuit produces calculated voltage

2.4 Traceability Analysis

The show how the requirements, specifications and testing procedures all link, Table 2.4 is provided.

Table 2.4: Requirements Traceability Matrix

<table>
<tr><th>#</th><th>Requirements</th><th>Specifications</th><th>Acceptance Test</th></tr>
<tr><td>1</td><td>BATT01R,BATT02R</td><td>BATT01S,BATT02S</td><td>-</td></tr>
<tr><td colspan="4">Design must use the Battery LiPo 800mAh 3.7V provided by the course to power the circuit. It must be connected to the circuit with the JST battery connector.</td></tr>
<tr><td>2</td><td>PCB01R,PCB02R,CST01R</td><td>PCB01S,PCB02S,CST01S</td><td>-</td></tr>
<tr><td colspan="4">Design's input/output connection to the rest of the system consists of a 2x8 pin header with a 2.54mm pin pitch. In order to clear the motor, the PCB must have the headers on a tab with height $>$ 18mm and width $<$ 35mm, and the components on this board when combined cannot cost more than $8.25.</td></tr>
<tr><td>3</td><td>-</td><td>-</td><td>AT01</td></tr>
<tr><td colspan="4">Measure all ground connections as well as the pours to ensure the design is correctly grounded.</td></tr>
<tr><td>4</td><td>-</td><td>-</td><td>AT02</td></tr>
<tr><td colspan="4">Test all resistor values to see how they vary when compared to their calculated values. Ensure they are within their rated tolerances and note how the tolerances changed their values.</td></tr>
<tr><td>5</td><td>SW01R,SW02R</td><td>SW01S,SW02S</td><td>AT03,AT04,AT05</td></tr>
<tr><td colspan="4">Design provides a discreet ON/OFF switching between the battery's voltage and 0V, that draws $<$ 500uA when in the 0V state. Test this by measuring the voltages at the BATT pins for both states of the button and measuring the current being fed by the battery when the switch is off.</td></tr>
<tr><td>6</td><td>CHRG01R,CHRG02R</td><td>CHRG01S,CHRG02S</td><td>AT06,AT07,AT08</td></tr>
<tr><td colspan="4">Design must be capable of charging the battery. Test this by providing and measuring the 5V input voltage to the 5V pins, connecting the battery, and turning the switch ON. Use a multimeter to measure the current drawn by the battery, then let it charge for $>$ 2 hours and measure its voltage to test its charge time.</td></tr>
<tr><td>7</td><td>MTR01R</td><td>MTR01S</td><td>AT09,AT010</td></tr>
<tr><td colspan="4">Design must drive 2 motors with an input of 3.7V, providing the motors with 200mA each. Ensure this by using a multimeter to measure the voltages and currents of the motors.</td></tr>
<tr><td>8</td><td>ADC01R</td><td>ADC01S</td><td>AT11</td></tr>
<tr><td colspan="4">Design uses a voltage divider to provide a 3.3V input, which gives the state of charge of the battery. Test this with a multimeter.</td></tr>
</table>

Chapter 3

Subsystem Design

3.1 Design Decisions

3.1.1 Final Design

From component browsing on JLCPCB it was found that the 0603 package resistors and capacitors were abundantly stocked, so unless explicitly stated otherwise all resistors and capacitors were selected from these packages.

Motor Driver

Due to their incredibly detailed datasheets (ratings, pin connections etc), the DRV8833 motor drivers are the best option for driving the micro-mouse. The selection choice was made by tabulating the following potential choices from JLCPCB and choosing the most viable option:

Table 3.1: Dual bridge motor driver ICs

Possible solution	**Rated Supply Voltage (V)**	**Rated Supply Current (A)**	**Price per IC($)**	**Availability**	**Dimensions (mm x mm)**
DRV8833PWP	2.7 - 10.8	1.5	1.1790	27	5 x 6
DRV8833PWPR	2.7 - 10.8	1.5	0.6660	9631	5 x 6
DRV8833RTYR	2.7 - 10.8	1.5	1.185	1315	4 x 4

from Table 3.1, the DRV8833PWPR was chosen, because of the following:

- Meets the required rated voltage and current
- Much cheaper option than option 1 and option 3
- Readily available (higher stock than 1 and 3)
- The datasheet provides a typical circuit diagram of how to implement the device

All capacitors were chosen at E24 values and added to prevent any AC noise from interfering with the motor driver circuit.

Battery Management

The TP4056 ICs all have the correct rated input voltage (5V) and charge voltage (4.2V). The following ICs were considered for the design:

Table 3.2: battery management ICs

Possible solution	**JLCPCB part #**	**Rated Input Voltage (V)**	**Quietient current/Iq (uA)**	**Price per IC($)**	**Availability**	**Dimensions (mm x mm)**
TP4056	C725790	4.0 - 6.5	35	0.0620	29409	4 x 5
TP4056	C17702041	4.5 - 8.0	55	0.0587	10658	5 x 5
TP5000X-4.2-ESOP8	C5447152	4.0 - 9.0	180	0.2465	5022	5 x 6

From Table 3.2, the first solution (C725790) was chosen. The reasons are as follows:

- Cheaper than option 2.
- Largest available supply.
- Smallest component (reduces PCB size).
- Lowest I_{q} ensures lowest possible current when mouse is active.

The charging current is controlled by the R_{prog} resistor. From the given equation in the datasheet:

$$I_{bat} = \frac{V_{prog}}{R_{prog}} 1200$$

Setting R_{prog} at 1600 Ω ensures I_{bat} is (750mA) which is less than the maximum rated current (1A) and that the resistor is of E24 value. The rest of the circuit was designed with the aid of a typical charging circuit on the datasheet of a similar IC [TP4056].

*The selected IC has built-in voltage regulation and protection, so this was not needed in the design

Analog Sensing

This consists of a voltage divider circuit that reduces the battery's read voltage to a maximum of 3V3 (ADC pin's required voltage).
Since the battery has a maximum of 4.2V, simply feeding this into the ADC pin could break it, so the divider circuit must ensure $V_{\mathrm{out}} \leq 3.3$V. from the equation of a voltage divider:

$$V_{out} = V_{in} \frac{R_7}{R_5 + R_7}$$

Setting $R_5 = 10\mathrm{K}\Omega$ and $R_7 = 33\mathrm{K}\Omega$, V_{out} is set at 3.22V which is safely below its rated voltage. By setting these resistors to a magnitude of more than 10^3, it ensures that the current in this circuit is minuscule.

Circuit Switching

To ensure that the battery is fully disconnected from the circuit, the switch is connected directly after the battery (Battery -> switch -> circuit). Some of the viable JLCPCB switches that match these requirements are tabulated below:

Table 3.3: Possible switches available on JLCPCB

Possible solution	**Type**	**Rated Voltage (V)**	**Rated Current (mA)**	**Price per Switch($)**	**Availability**	**Dimensions (mm x mm)**
EG1224	Slide	15	500	0.7695	70	7 x 12
PS-5850A-6PL	Push (Latching)	30	500	0.1185	1514	5 x 5
SS12D07VG4 087	Slide	50	500	0.0278	28097	8 x 9
YTSPS-22E58LM	Push (Latching)	50	500	0.1588	1705	5 x 8

From Table 3.3 the PS-5850A-6PL was chosen. The reasons are as follows:

- Meets regulated voltage (> 4.2V) and regulated current (> 400mA).
- Push switches take up less horizontal space (reduce PCB space).
- Cheapest push switch.

Final design

The design choices were made to increase the efficiency of the power subsystem [3]. This includes selected parts that were relatively cheaper when compared to similar components, consumed less current than other components/consumed as close to the required current as possible, and had smaller dimensions which helped to reduce the size of the final PCB and in doing so helps keep the micromouse's center of rotation as close to the front as possible.

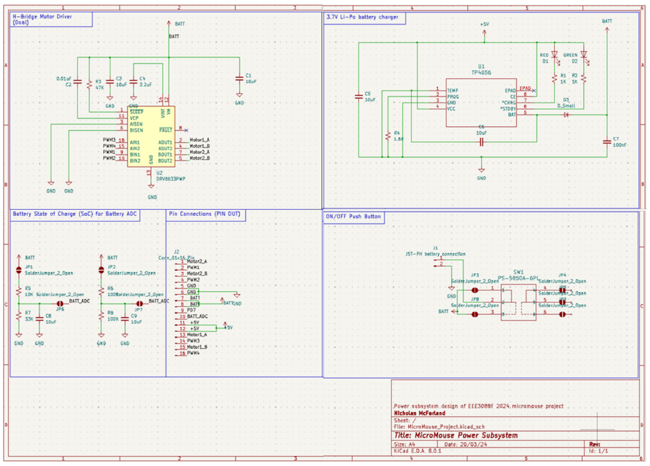
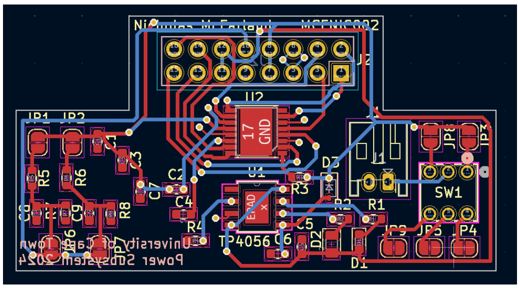
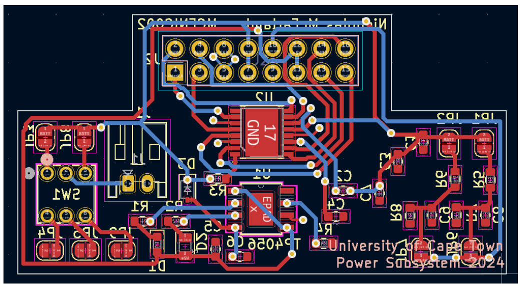


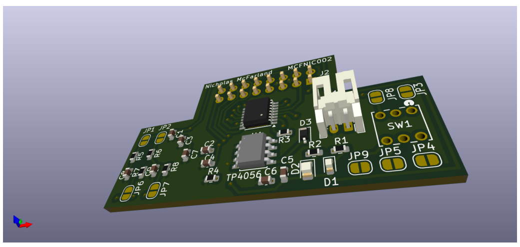
Figure 3.1: Schematic



(a) Front PCB (with ground pours omitted for clearer view of tracks)



(b) Back PCB (with ground pours omitted for clearer view of tracks)



(c) 3D PCB (with all components populated to depict the final PCB layout)

Figure 3.2: Final PCB design layouts

3.2 Failure Management

Table 3.4: Failure Management Table

Name	Description
Track current handling [4]	Since the motors carry currents too large for standard track width, track width was increased. All tracks' widths were increased, due to uncertainty as to which tracks would carry the highest currents.
Battery/5V Tracks	Tracks leading directly from the switch and 5V charging input were put as close to the edge of the PCB as possible to minimize any interference with other tracks. This is because they are likely to draw high currents.
Switch placement	Switch placed directly after the battery to ensure the entire circuit is cut off from power when the switch is on.
Switch solder bridges	Due to the vague diagram on datasheet, uncertainty arose concerning pin connections. Each connection is joined with a solder bridge to ensure the right connections are made in the future.
Additional analog voltage-divider	In case the voltage-divider output voltage is too close to the rated ADC pin, a second one was designed with a much lower output voltage (half of the maximum battery voltage)
Analog solder bridges	Both circuits are separated from battery input and pin output by solder bridges. This ensures that the correct one can be connected, whilst the unused one does not draw any current from the battery (remains completely idle).
E24 component calculation rounding	All calculated E24 component values rounded to ensure the relative currents/voltages are well below rated maximums and well above rated minimums.
ERC on KiCad	After the final circuit design was chosen, the Electrical Rules Checker was run to ensure no electrical rules were broken and no avoidable errors would occur on the PCB.
DRC on KiCad	Design Rules Checker was run to ensure no design rules were broken on the PCB.

3.3 System Integration and Interfacing

To integrate the subsystem with the rest of the system ....

Table 3.5: Interfacing specifications

J201	Motor2_A	• Provides voltage output of Motor2_A
J202	PWM1	• BIN1 Logic control pin of BOUT1 (Motor2_A)
J203	Motor2_B	Provides voltage output of Motor2_B
J204	PWM2	BIN2 Logic control pin of BOUT2 (Motor2_A)
J205/J206	GND	• GND pin
J207/J208	BATT	• Provides battery voltage
J209	PD7	• reserved pin
J210	BATT_ADC	• Provides 3V3 ADC voltage
J211/J212	5V	• Provides 5V charging voltage
J213	Motor1_A	Provides output voltage of Motor1_A
J214	PWM3	• AIN1 Logic control pin of AOUT1 (Motor1_A)
J215	Motor1_B	Provides output voltage of Motor1_B
J216	PWM4	• AIN2 Logic control pin of AOUT2 (Motor1_B)

System Block Diagram

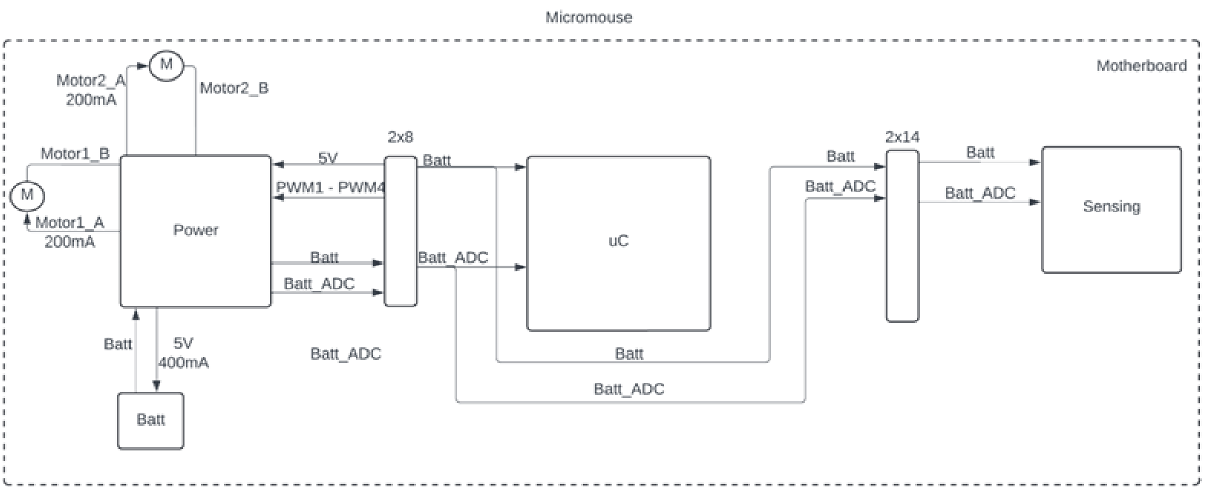


Figure 3.3: High-level block diagram showing subsystem's link into the larger system. (Information on sensing inputs/outputs limited)

Chapter 4

Acceptance Testing

Table 4.1: Subsystem acceptance tests

Test ID	Description	Testing Procedure	Pass/Fail Criteria
AT01	All ground connections correctly grounded	• Use multi-meter to measure all ground connections • Measure both ground pours	Pass if ALL voltages = 0V
AT02	All resistor values match the design values	• Use multi-meter to measure the resistance of all resistors	Pass if all resistors are within design values' tolerances
AT03	Switch turns system ON when pushed DOWN and OFF when RELEASED (personal preference)	• Solder JP3 and test switch • If inverted results, desolder JP3 and solder JP8	Pass if switch ON when pushed DOWN
AT04	Switch provides discreet ON/OFF states	• Use multi-meter to measure battery's current voltage • Push switch down and use multi-meter to test voltage at pins 7/8	Pass if pin voltage = battery voltage
AT05	Quiescent current less than maximum rated	• Connect multi-meter between the positive battery terminal and positive JST pin and turn switch ON • Set all motor input pins low to keep the motors off OR keep motors disconnected	Pass if current < 500uA
AT06	Voltage provided by 5V pin is indeed 5V	• Use multimeter to measure voltage at pins 11/12	Pass if voltage = 5V
AT07	Charger charges battery	• Provide 5V to pins 11/12 • Connect battery and turn switch ON • Check green LED is ON	Pass if after 2 hours battery voltage = 4.2V
AT08	Battery draws < 400mA when charging	• Connect multi-meter between the positive battery terminal and positive JST pin and turn switch ON • Set 5V input pin to 5V	Pass if current < 400mA
AT09	Motors reach operating current	• Connect motor A and B to system • Connect ammeter between Motor_A pin and motor	Pass if currents = 200mA
AT10	Motors reach operating voltage	• Connect motors to correct pins • Set relative input pins high [see datasheet in Table 3.1]	Pass if pin voltage = rated voltage
AT11	Analog sub-circuit produces calculated voltage	• Solder JP1 and JP6 • Use multi-meter to measure output voltage • (If voltage > 3.3V, desolder JP1 and JP6, then solder JP2 and JP7 and repeat)	Pass if voltage < 3.3V

Bibliography

[1] [Online]. Available: https://jlcpcb.com/parts/all-electronic-components

[2] "Preventing pcb failure: Causes, symptoms, and maintenance tips." [Online]. Available: https://www.pcbaservices.com/preventing-pcb-failure/

[3] "Kicad design pcb: Tips and tricks for successful printed circuit." [Online]. Available: https://hillmancurtis.com/kicad-design-pcb/

[4] "Trace width calculator." [Online]. Available: https://forum.kicad.info/t/trace-width-calculation/16091