WUFR 24
Lauren Lynch
Electronics and Data Acquisition

Power Distribution Module FDR

Summer of 2023

Table of Contents

Overview

The WashU Racing Power Distribution Module (PDM) is a printed circuit board (PCB) designed to efficiently draw power from the car's 12-volt battery and ensure optimal distribution to various sensors and accessories. Last year, Quinn and I prioritized the integration of passive components such as relays, fuses, resistors, and LEDs due to time constraints. However, in my current role as the EDA tech lead, I have taken on the responsibility of enhancing the PDM's capabilities by introducing a microcontroller. This addition will enable direct monitoring of current and temperature on the board and facilitate other outlined goals. Through this document, I aim to provide insights for future EDA PCB designers and highlight the design possibilities they can explore.

PCB Goals

Reliability

To ensure the integrity of the PCB, our goal is to achieve **maximal heat distribution** and manage the temperature effectively. This is crucial for preventing component failures and preserving the overall performance of the board.

Accessibility

Our objective is to design the PCB for easy serviceability and maintenance. The code for the board should be **easily debugged**, and any failed mechanical **components are replaceable** without extensive effort. We also aim for **space optimization** and **manufacturability** by minimizing the size of the board for cost reduction and ensuring that the physical components are straightforward to assemble respectively. This is also important for skill transfer within EDA, where I plan to teach new members how to solder first.

Performance

Our performance goal focuses on **signal integrity** and minimizing electromagnetic interference. Additionally, our goal is also creating **self-diagnostic systems**, which would save time by notifying members of any possible electrical problems.

Reliability Design Choices

Modification	Justification
Addition of temperature sensors	To quantify heat distribution of the board, temperature sensors should be added to the board. This will be useful data for future PCB designers on the team, especially on the board with managing the most current. Link to temperature sensors: [Mouser Link]
Addition of current sensors	Although this idea was thought of by past EDA tech leads, the addition of current sensors will allow us to more carefully monitor the current going to each device. This data can be taken beforehand and passed down to future leads. Link to current sensors: [DigiKey Link]
Addition of heat sinks and fans	Heat sinks and fans are commonly used to cool down a PCB, which is particularly crucial in this case when the PDM is housed in a plastic enclosure, instead of having the entire bunker for cooling. Ideally, heat sinks should be added on free copper plains and a fan should be attached to an enclosure. Link to source: [Cadence Link]
Adjusting trace widths	Last year, we had wide traces to ensure solid connections from one point to another. However, a judge pointed out that this was impractical, so traces will be adjusted according to their current. Link to source: [4PCB Webste]

Accessibility Design Choices

Modification	Justification
Addition of feather board	The feather board will monitor the current through each sensor, allowing us to replace most of the fuses on the board and effectively optimize space by eliminating additional components. Not only has the feather board demonstrated its reliability in the past, it provides the opportunity for a self-diagnostic system.
Removal of LEDs	The feather board's capability to monitor and regulate current to each sensor raises the question of whether the LEDs previously used by tech leads to indicate current flow are still required. Since the PDM is in an opaque enclosure, the LEDs are no longer necessary, which will help reduce the cost and size of the board. Instead, all current will be monitored through the ADC.

Usage of through-hole components	All components, except for resistors, will be through-hole. (Surface mount resistors, despite the potential challenges in manufacturing, are usually justified by their small size and facilitate space optimization.)
Addition of a non-Deutch connector	EDA has always used standard automotive Deutsch connectors. However, I chose to use another connector because we need one with 24 pins, which isn't offered in a space optimized way from the Deutsch brand. Link to connector: TE Connectivity Link

Performance Design Choices

Modification	**Justification**
Switching on/off justifications	Perhaps the most important decision when making the PDM is determining which elements need to stay on or whether they should be changed. Because of mechanical issues the water pump, fuel pump, and fan must always stay on. Everything else may be turned on and off by the featherboard.
Incorporation of digital circuit breaking and alternative fuse placement	In efforts to create a self-diagnostic system, the featherboard will be responsible for digital circuit breaking. This will effectively replace some components and help optimize space on the board.

Signal integrity techniques	Although the team utilizes industry-standard automotive components and ECAD software, the implementation of formal signal integrity techniques by EDA have not been explicitly utilized. Here are some techniques and references that I plan to use.
	● Altium Performing Signal Integrity Analysis
	    ○ How to run signal integrity analysis on altium
	● Cadence Signal Integrity Techniques
	    ○ Addresses 4 major signal integrity problems: crosstalk, ground bounce, impedance mismatch, and EMI
	    ○ "Keep signal path traces short and direct."
	    ○ "Sensitive signals should be routed on internal layers and next to or between reference planes whenever possible."
	    ○ "Clock lines and other sensitive high-speed signals should be kept separate from other traces as much as possible."
	● Cadence Impedance Matching Fundamentals

Addition of AI/ML	Now that we have the featherboard, we can code whatever algorithms are most helpful for self diagnostics. Here are some ideas that I hope can come to fruition. Ideally, I'd like someone who knows more about code to help me with this. ● Real time decision making ○ If a sensor is getting too much or too little power, digitally break the fuse and notify the user. ● Feedback ○ Analyze data from previous runs and compare to the current run to see if there are any major anomalies. ● Data Processing ○ Clean the data before it is processed with maybe a moving average line. A little noise filtering would be ideal to capture larger patterns.
Addition of Multiplexer (MUX)	Facilitate Switching

Pre-layout Plan

- Based on the reliable performance of the PDM last year, it can be inferred that all the components utilized were functioning properly. The ultramini relays, fuses, and featherboards, which have been employed in previous iterations, have demonstrated their effectiveness and will be retained in the current design.

- In addition to the existing components, two new components will be integrated into the PCB to connect the featherboard to other board components: a relay driver and an ADC (Analog-to-Digital Converter). The relay driver will establish the connection between the featherboard and the relays. On the other hand, the ADC will facilitate the transmission of signals from the current sensors to the featherboard, allowing for accurate measurement and monitoring.

- The featherboard will be responsible for self-diagnostics and turning the relays on and off, except for the kill switch which must turn on mechanically. I hope to have someone interested in code take on an AI/ML project with me.

Electrical Connections List

Part	Description
Ultra Mini Relays 5 terminal	Coil 1: One side of the coil Coil 2: Another side of the coil COM: Switch NC: Normally Closed NO: Normally Open • 12 volts is connected to COM because that is the voltage we want to flow through the relay when switched on • Nothing should be attached to NC because we don't need the relays to do anything when off. • NO should be attached to the device that needs power. • Coil 1 and 2 should be connected to a pin on the RD.
Current Sensors 5 terminal	GND: Ground IP+: Current In IP-: Current Out VCC: 3.3 volt power supply VIOUT: Voltage signal containing the current measurement
Temperature Sensors 5 terminal	GND: Ground NC: Not connected SCLK: SPI clock SDA: SPI data VDD: 3.3 volt power supply
ADC 20 terminal	AINX: Analog input pins CS: Chip select specific to ADC Data In: MOSI connection to featherboard Data Out: MISO connection to featherboard EOC: End of conversion I/O Clock: SCK on featherboard GND: Ground Ref+: 3.3 volt power supply Ref-: 0 volt power supply VCC: 3.3 volt power supply
Relay Driver 24 Terminal	CS: Chip select specific to RD OUTX_D: Out drain connected to relay OUTX_S: Grounded SI: MOSI SO: MISO IN0, IN1, IDLE: Control the relay driver without SPI

Post-Layout Reflection
Methodology and Approach

I conducted two rounds of routing for the PDM. In the second approach, I addressed issues with electrical connections by rearranging the pin assignments before routing. Although it disrupted the schematic's alphabetical order, swapping pins prior to routing played a crucial role in the success of my second routing attempt because it ensured that the electrical connections were not blocked by any vias and facilitated trace length minimization. Consequently, I advise future WashU Racing PCB designers to **determine the pin assignments in the schematic before beginning routing** to ensure the stability of electrical connections.

In addition to swapping pins, another decision that proved to be helpful was routing the largest components first. For instance, in my initial approach, I routed the smaller components, such as the temperature sensors with six connections, before gradually moving on to larger components because they were more approachable. However, this approach presented challenges in terms of navigating larger traces around the smaller, more intricate ones, and it also resulted in the need for a significant number of vias. To address this issue in the second routing attempt, I reversed the order and started with the multiplexer, allowing the smaller components to be positioned around their larger counterparts. To ensure a smoother routing process, I recommend that WashU Racing PCB designers **begin routing with larger components** and then gradually progress towards smaller ones. This approach will help prevent complications that can arise when larger traces need to navigate around smaller ones, optimizing space utilization on the PCB.

Polygon Justifications

Initially, I made specific decisions regarding the placement of polygons and layers in the PDM design. On the top layer, I positioned the four 12-volt polygons on the top layer, considering that they would generate the most heat on the board. On the top-middle layer, I surrounded each of the current sensors with polygons and allocated a plane for +3.3 GPIO power. For components that couldn't fit on the top-middle layer, such as the PE3 fan and the fuel pump, I designated two extra polygons on the bottom-middle layer and grounded the feather board. Lastly, I dedicated a plane for ground on the bottom layer.

However, to maximize the available space for heat sinks, I made the decision to **switch the placement of the bottom and top layer polygons** in my second routing attempt. This adjustment aimed to optimize heat dissipation capabilities. Furthermore, I reorganized the inner layers to ensure that the layers with higher current flow were positioned closer to the bottom layer. This arrangement aimed to minimize resistance and maximize the efficiency of power distribution. To maintain optimal signal integrity on the 12-volt planes, I concentrated the majority of the **traces on the top two layers to prevent interference** that could impact signal quality.

Trace Width Justifications

Last year, the design judge criticized our design for not engineering the trace widths to match the amount of current flowing through the board. This year, the majority of our traces connect to the microcontroller, which intrinsically contain a smaller amount of current thereby justifying their small size. Additionally, JLCPCB can only manufacture trace widths at a minimum of 10 mil.

Active Cooling Justifications

Last month, a part of the PDM enclosure started melting, so this year we will use active cooling by including a fan and heat sink. The fan will be powered by a 12 volt and ground vias in the corner of each board.

PCB Design Research

Thermal Management Altium Academy Series

- Can replace components with thermal resistance and add a voltage source representing ambient temperature.
- The thermal mass can be represented with a capacitor in parallel with each resistor, but this is only necessary for unique applications.
- Recommendation of oversizing heat sink by 30%
- Multiply the RθJC, RθI, and RθHS, by the watts being dissipated and sum results together to get the total heat drop then compare to the maximum operating temperature Tjmax. Must perform this process for each component on the board.

Heat Dissipation for a Component

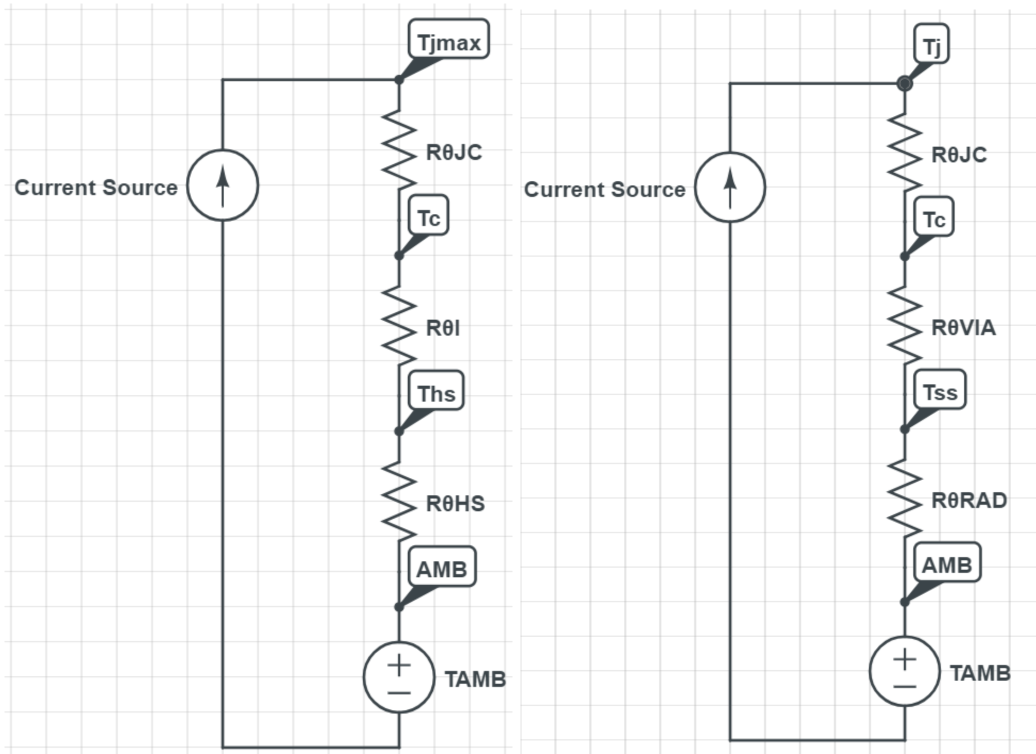
Name	Abbreviation	Units	Notes
Junction-to-case thermal resistance	RθJC	°C/W	
Interface material thermal resistance	RθI	°C/W	Can add thermal layer between uneven component and the board for even more heat dissipation
Heat sink thermal resistance	RθHS	°C/W	Must determine operating point from natural and forced convection characteristics as a function of power dissipation and get power dissipation from there

Ambient temperature voltage source	TAMB	℃	
Junction Temperature	Tjmax	℃	Between current source and RθJC Maximal operating temperature
Case Temperature	Tc	℃	Between RθJC and RθI
Heat sink temperature	Ths	℃	Between RθI and RθHS
Ambient Temperature	AMB	℃	Between RθHS and TAMB

Heat Dissipation for a Via

Name	Abbreviation	Units	Notes
Junction-to-case thermal resistance	RθJC	℃/W	
Via Barrel thermal resistance	RθVIA	℃/W	Height of via divided by surface area of via and thermal conductivity constant (S/AK)
Radiating Plane thermal resistance	RθRAD	℃/W	1 divided by heat transfer constant and surface area of VIA (1/HA) To decrease the thermal resistances, increase the surface area of the vias
Ambient temperature voltage source	TAMB	℃	
Junction Temperature	Tjmax	℃	Between current source and RθJC Maximal operating temperature
Case Temperature	Tc	℃	Between RθJC and RθI
Secondary Side temperature	Tss	℃	Between RθI and RθHS
Ambient Temperature	AMB	℃	Between RθHS and TAMB

Schematics[1]



Readily Available Thermal Characteristics of PDM Components

	ADC	Connector	Current Sensors	Multiplexer	Relays	Relay Driver	Temperature Sensors
$R\theta$JC (℃/W)	40.4	0	7	0	0	5	7
$R\theta$I (℃/W)	0	0	0	0	0	0	0
$R\theta$HS (℃/W)	0	0	0	0	0	0	0
Total Thermal Resistance (℃/W)	40.4	0	7	0	0	5	7
Max Operating Temp (℃)	85	125	165	85	125	150	150

From the chart above, we should try to keep the temperature to **64 degrees celsius** or lower, which is about **75% of the maximum operating temperature** of the 2 most fragile components. This may be a really optimistic goal because so many components have higher operating temperatures.

[1] https://www.circuitlab.com/editor/#?id=7pq5wm&from=homepage

Analog to Digital Converters (ADCs) in the PCB Altium Academy

Introduction

- Overview of signal conditioning and noise reduction techniques for ADC inputs.
  RC Filters for ADC Inputs
- Explanation of using RC filters to attenuate high-frequency noise and prevent aliasing.
- Importance of selecting appropriate resistor and capacitor values based on the desired cutoff frequency and sampling time.

Grounding Techniques

- Recommendation to keep analog and digital grounds separate to prevent interference.
- Explanation of the benefits of grounding digital and analog signals together at the ADC.

Isolated ADCs

- Definition of isolated ADCs with galvanic isolation between their DC and AC pins

Signal Separation

- Recommendation to physically separate analog and digital signal paths for signal integrity.
- Benefits of minimizing crosstalk and electromagnetic interference (EMI) by keeping analog and digital signals on opposite sides of a PCB.

Optocouplers for Noise Reduction

- Introduction to optocouplers as a means of electrical isolation to reduce noise.

Quantization and Noise Analysis

- Explanation of how ADCs quantize and record signal values over a dynamic range.
- Comparing noise levels to the quantized voltage to determine SNR.
- Techniques for measuring and analyzing noise in ADC systems.

Oversampling and Anti-Aliasing

- Introduction to oversampling as a technique to increase the effective resolution and reduce quantization noise.
- Overview of anti-aliasing filters to remove high-frequency components before ADC conversion.
- Explanation of the Nyquist filtering criterion and its role in preventing aliasing.

https://www.youtube.com/watch?v=D0X76Kbf8fQ

- Vary trace widths, space out traces as much as possible, check with the manufacturer to see if they can manufacture the width of required traces and vias.

- Make sure to have an annular ring of at least 0.15 mil
    - (Via diameter - hole diameter)*0.5 = annular ring size

Manufacturing Dates and Times

Process	*Time Required*
Ordering Parts	2 Weeks
Ordering the Board	2 Weeks
Time to manufacture board	1 Week
Test Plan	2 Weeks

Manufacturing Plan

Step	Description	Operator	Date
1	**Gather Parts** Gather all the parts you need for building the PDM (Power Distribution Module). You can find a list of these parts and how many you need in a document called the BOM (Bill of Materials) in the Electronics FDR folder.		

2	**Test Parts**
	Bench test each part by following the instructions provided in the component's datasheet to ensure that their electrical characteristics match the original design constraints. Record the voltages, currents, and power dissipation in a table for each of the following parts.
	☐ Relays
	☐ Female Connector
	☐ Male Connector
	☐ Current Sensors
	☐ Resistors
	☐ Capacitors
	☐ ADC
	☐ Relay Driver
	☐ Multiplexer
	☐ Temperature Sensor
	☐ Diode
	☐ Featherboard Microcontroller
	☐ Resistors
	    ☐ Make a table for every resistor’s true value
	☐ Capacitors
	    ☐ Make a table for every capacitor’s true value		

3	**Continuity Check** Use a handheld device called a DMM (Digital Multimeter) to check if all the electrical connections on the circuit board are actually connected.		
4	**Soldering Surface Mount Components** Put a special template called a solder stencil on the PCB to make sure you apply solder (a material that connects electronic components) to the right places for the small components that sit on the board's surface. Then, place these components on the board. Finally, use a toaster that is set to the temperature recommended on the solder's packaging to melt and bake these components into place.		

	* In this case, your surface mount components are the relay driver, ADC, multiplexer, resistors, capacitors, regulators, fuse, and temperature sensors.		
5	**Soldering Through Hole Components** For the through hole components, solder them to the board by hand with a soldering iron. Follow safety rules for soldering. Stay safe. Do not solder in the connector at this point. * In this case, your through hole components are the current sensors, diode, feather board microcontroller, relays, and fan.		
6	**Solder the Connector** Attach the connector to the circuit board by soldering it in place. This connector is an essential part of the board that allows it to connect to other devices.		
7	**Place PCB into Enclosure** Place the circuit board in the enclosure to see how well it fits. Stay at this step until the connector fits well with the board.		
8	**Code** Write corresponding code to match the original intent of the circuit.		

Testing Plan

Step	Description	Operator	Date
1	**Visual Examination** Visually examine the solder on the board to check for any possible burns or possible places where there is too much or too little solder. Alert an EDA lead if something looks incorrect.		
2	**Continuity Check** Use a handheld DMM to check if there is a complete and unbroken path for electricity to flow through all the visible lines on the circuit board and any components that are connected to each other electrically after the solder has been placed on.		
3	**Power Supply Test 1** Use the power supplies to check each output of the PDM. Ensure that the current sensors are tracking reasonable data.		

4	**Power Supply Test 2** Use the power supplies in the Electrical Engineering Laboratory to provide the maximum amounts of current we expect to have going to the board at maximum. Take measurements on the current sensors and temperature sensors to ensure that the maximum temperature near the connection is below the maximum operating temperature of the nearby components.		
5	**Data Analysis** After the PDM has been put on the car and thoroughly debugged on behalf of the coders, analyze the data on the PDM to see if it makes sense and matches theoretical expectations. Alert technical leads if it doesn't make sense.		

Verification of Completion for Testing and Manufacturing Plans

Technical Lead	Initials
Quinn Trent	
Connor Irwin	
Hayden Schroeder	
Lauren Lynch	