PCB Design, Assembly, and Manufacturing of a Power Distribution Board

ECE 499

University of Victoria
Faculty of Engineering

Spring 2021

ECE 499 Final Report (Group 5)

University of Victoria
Faculty of Engineering
Victoria, British Columbia

Kyle Jarvis (V00730008), Bryce Gammon (V00709534), Sean Schocat (V00925804), Dai Tran (V00928014)
ECE 499
Electrical / Computer Engineering
Email: kjjarvis@uvic.ca, bryceg@uvic.ca, xh0katsean@uvic.ca, dait@uvic.ca.

March 28th, 2021

In partial fulfillment of the requirements of the B.Eng. Degree

University
of Victoria

Group ID: Group 5

Faculty Supervisor: Babak Manouchehrinia

Co-supervisor (if any): Kieran Warren

Team Members (include all) Information:

#	Name	V Number	Program
1	Bryce Gammon	V00709534	ECE
2	Dai Tran	V00928014	ECE
3	Kyle Jarvis	V00730008	CENG
4	Sean Schocat	V00925804	ECE

Acknowledgment

We would like to thank everyone who contributed to the success of this project. We would like to thank our project supervisor Babak Manouchehrinia, and project co-supervisor Kieran Warren for their guidance and support. We would also like to thank our friends and family who supported us during this semester of online learning.

Abstract/Executive Summary

UVIC’s Centre for Aerospace Research (CfAR) has a PDB consisting of four separate converter circuits on one circuit board - a 3.3V, 5V, 8V, and 24V converter. Some of these converters have existing issues to be addressed, but it is additionally desired to separate the circuits into distinct boards that can share the same input source to make the design modular. This would allow CfAR to more flexibly replace malfunctioning components, and use combinations of PCBs as desired.

To achieve modularity, board-to-board connectors were chosen so that the PCBs could be stacked on top of each other. Additionally, each board supports the connector for the input supply so that any PCB can be the designated input board and route the input power to all the other boards connected in the stack with it.

The Altium files for CfAR’s current revision of the PDB were provided as a starting point. The 3.3V and 5V converters were redesigned to be on their own PCBs with no circuitry changes since they were stated to be functioning as intended. The existing 24V was not working properly and, along with the 8V, had components that were difficult to replace prompting full redesigns. Following the modular design criteria mentioned before and using recommended design procedures each board was successfully designed to be stackable and able to transfer a 12V input voltage between each board. The final PCB designs were successfully created in Altium to the stage where manufacture files can be generated along with their BOMs, which can be used to order the components necessary to physically assemble and use the PCBs.

Table of Contents

List of Figures and Tables

Glossary

EMI – Electromagnetic Interference
An electromagnetic signal from an external source which interferes with the operation of a electronic circuit.

IC – Integrated Circuit
A set of electronic circuits constructed on a single device, typically called a chip, which are much smaller than discrete components.

PCB – Printed Circuit Board
The mechanical layers that support the conductive layers and electrical components of an electrical circuit.

PDB – Power Distribution Board
A system designed to provide differing voltage outputs from a single voltage input.

VQFN – Very Thin Quad-Flat No-leads
A type of integrated circuit package, which is leadless and allow for surface mounting on the PCBs without through-holes.

PMU – Power Management Unit
A controller that monitors power systems that use DC voltage.

PWM – Pulse Width Modulation
A type of signal chopping, it reduces the average voltage and current by rapidly switching the voltage from its maximum to zero potential and back.

I Introduction

CfAR (UVIC's Centre for Aerospace Research) has an unmanned aircraft that requires a PDB to power its various systems, and its current implementation had limitations and issues CfAR wanted to address. CfAR has provided revision "E" of their current PDB design as a starting point. The aircraft's systems need a PDB that can provide 3.3V, 5V, 8V, and 24V outputs using an input supply of 12V DC. Revision E's converter circuits are currently all on the same PCB, but it was desired to split them into independent PCBs so that the PDB design could be modular.

An extension to designing the PDB for CfAR's purposes was to implement a new PDB in the UVIC Robotics Clubs' Mars rover. This rover requires the use of a 5V and 12V converter for power distribution. As members of the UVIC Robotics Club, we can capitalize on the modularity of the PDB design for CfAR by incorporating the design of the 5V PCB along with a modified version of the 8V PCB to meet its 12V requirement.

This type of circuit design work is important because a modular design allows the use and replacement of individual converters. Cost of development is reduced if one converter requires a redesign while others do not, or design specifications change for components to be powered. Lastly, like in UVIC Robotics Clubs' situation, individual converters can be repurposed for other projects with less work involved.

There are minimal expected social impacts to be brought about by the design of this project, as we are designing a system for a public research institution. According to CfAR's mission statement, "the Center for Aerospace Research aims to spur innovation, investment, and attention to the positive role [unmanned arial vehicles] can play in the betterment of society" [1]. As such, we hope that our work can lead to fulfilling and furthering CfAR's objectives to improving society.

It was identified that there is potential to create and sell modular converter circuits for PDB's. A common example is robotic systems. Their components such as sensors and motors require different supply voltages but use a single power source. A possible user base would be any organization that designs robotic systems such as drones, rovers, or autonomous subs.

II Project Goals

The goal of the project is to design a modular PDB to replace an existing PDB system for an unmanned aircraft's various electrical systems. The existing PDB system needs to be split up into independent PCBs that can be used in any combination required, where two of the converter circuits need to be redesigned to a) incorporate ICs that can be more easily replaced, and b) properly provide the desired output they do not currently meet.

III Design Objectives

The design constraints presented were overall light on quantitative requirements. A design specifications document was not provided for this project, and all listed quantitative values herein were obtained from meetings with the supervisors. For CfAR's intended purposes there was a lot of freedom to source nearly any component necessary to accomplish the goals presented in section II. There was also freedom to design the circuit boards to be any dimensions as long as consideration was made to keep them as small as possible. In particular, the existing PDB system contains four converter circuits powered by a nominal 12V input battery source where this could fluctuate from 9-14V. The four converter circuits have expected outputs of 3.3V, 5V, 8V, and 24V.

The provided 3.3V and 5V converter circuits function as intended so their designs can be reused. The provided 24V converter circuit does not currently function as intended, and the 8V converter uses an IC that is difficult to replace so both converters require new ICs. The new PCB design for both should have ICs that are replaceable and achieve the same power output.

Previous revisions of the PDB board used a four-layer design to reduce noise and allow for closer component grouping. To reduce printing costs, the PCB designed for this revision will be two layers, reduced from the four layers used in the original design. A four-layer board can easily cost 5-10 times as much as a two-layer board depending on its size. Despite the possibility of increased noise at the output, this is deemed an acceptable trade for cost savings.

From the expected power requirements board inputs needed to be able to handle 25A continuous current, and connections between boards must be able to sustain the same amount of current. Expected power outputs of every board are included below in table 1.

Table.1 Board Voltages and Expected Power Output

Board Voltage	Expected Power Output
3.3V	7W
5V	30W
8V	112W
24V	80W

There was no duration of operating time provided for the project's technical specifications. Based on the application for an unmanned aircraft, this would be dependent on the battery used and it's specified ratings on capacity which would influence how long the aircraft could feasibly be in flight. The expected efficiency of each converter was not specified, but it is expected that unless there is a clear reason for accepting lower efficiency the designs should be as efficient as possible.

Noise was not specified to be an issue beyond what is normal for DC-DC converters. Designs should attempt to minimize noise by following recommended manufacturer PCB layouts and suggested bulk capacitance. Similarly, a budget was not specified beyond encouragement to keep costs as low as possible and reducing the number of board layers where possible.

IV Literature Survey

DC-DC conversion is a necessary component for many devices, so possible solutions are numerous. The scope of the project encouraged minimizing cost, designing a modular system, and matching the power specifications provided as closely as possible. With these points in mind there are still various means to achieve the project objectives. In this literature review three different DC-DC converter design approaches are discussed, with one other topic dealing with board modularity.

Combined Single Unit Supply

The approach most similar to previous revisions of the CfAR PDB is a single combined supply unit. A high-end example researched for this was a 250W unit designed by Millsworth Engineering as shown below in figure 1 [2].

Figure 1: Millsworth Engineering 250W PMU.

This unit has a wide range of input and output voltages, high power output, and is compact. A combined module is the go-to approach for power solutions in many areas such as avionics, remote industry, and other activities involving devices that require a large range of voltages. For example, a similar power distribution unit intended for racing drones is shown below in figure 2 [3].

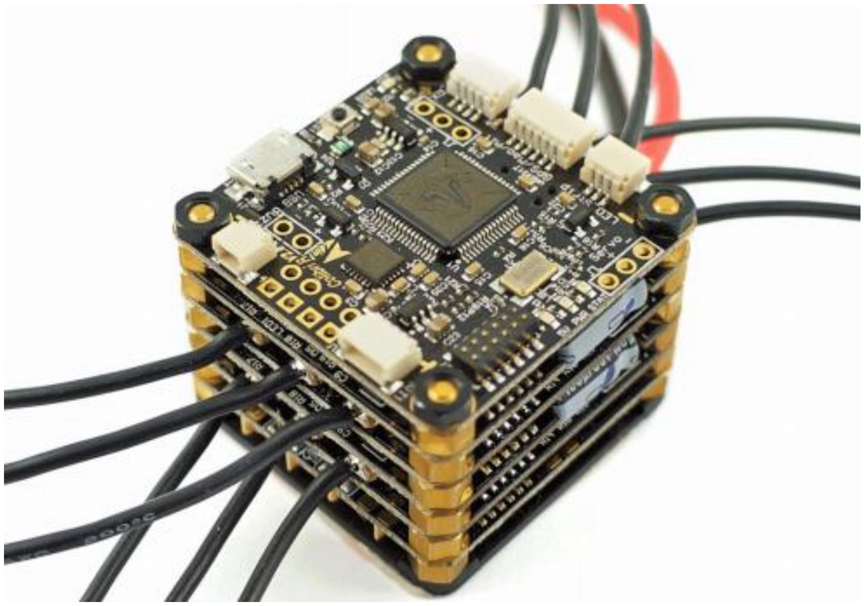

Figure 2: PowerCube V2-F3.

This unit has 3.3V and 5V outputs with the capability to perform PWM with a 12V input that produces lower voltages. While both designs are impressive and have excellent capabilities, they do have some disadvantages. The first is that neither unit is modular. While this would not be a problem if they matched the necessary output voltages, both lack at least one of the expected outputs CfAR needs. The second disadvantage is cost, which is dependent on the capabilities of each unit. The PowerCube is available at an acceptable price of $109.95 USD [3]. This is reflected in that it only produces 3.3V and 5V outputs. The 250W Millsworth PMU is extremely expensive at $3990.00 USD [2], but it can output between 10-24 volts with several different output modes and has battery management incorporated. This pricing is found to be reflected by other similar units, where capability and power rating increase price beyond what is reasonable for the project.

Discrete Component Design

The traditional approach to designing a custom power solution is specifying and creating a PCB based around discrete components to achieve the desired voltages. As previously stated, there are many ways of creating a DC-DC converter. The possibilities in creating a PCB from sourced passive and active components were examined.

There are a large variety of ICs available for DC-DC conversion. They typically fall into two distinct varieties: ICs containing switching MOSFETs, and ICs that drive external MOSFETs. Internal MOSFETs often limit current output typically to less than 5-6A due to heating of the circuit. ICs that drive external switches are limited more by the current rating of the switches. Resistors, diodes, and other capacitors are used to control what voltage and current is found at V_{OUT}. An example buck converter circuit using a PWM IC is shown below in figure 3 [4].

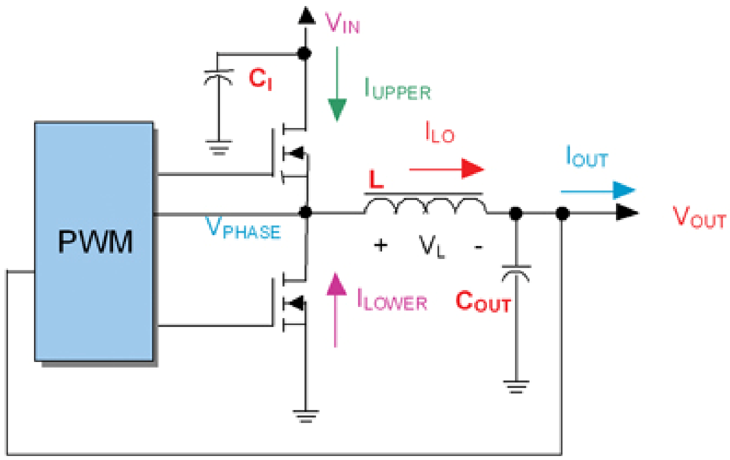

Figure 3: Example of buck converter layout with PWM IC.

The advantage of this approach is its flexibility. A power solution can be designed from the ground up to fit the exact specifications required. Costs for this kind of approach are quite variable depending on the power requirements. While ICs are not expensive on their own, all passive components must be sourced along with a manufactured PCB. As power requirements increase component costs and board complexity increase. This can cause solutions to quickly become expensive, and sometimes less effective. Since these solutions use custom PCBs there can be unforeseen issues arising from noise, spot heating, component failure, and design issues that are not discovered until prototyping. Generally, these solutions require several iterations and tests before a final solution is found.

This was the approach decided on for the 3.3V, 5V and 8V boards. These boards have minimal power requirements and were able to be made with minimal complexity while maximizing the flexibility to change and modify the designs to better suit the projects requirements. It should be noted that the 3.3V and 5V circuits from the previous revision were found to be acceptable and were reused, albeit with different layouts and fewer layers for the new PCB designs.

Single Unit Converter

Single unit converters, also called “brick converters” due to the way many are packaged, are an industry standard for PCB power supply solutions. An example of what these look like are included in figure 4 below [5].

Figure 4: Examples of brick converters.

Each unit has a wide input voltage range and a single output voltage. This allows the unit to be very efficient for a specific application. A variety of models are available for a range of voltage inputs and outputs, so there is usually not an issue finding one that fits the voltage requirements of a project. Many also come with additional features such as thermal protection, operational monitoring, and ripple control [5]. The standardized packaging also allows them to be easily swapped out if they are found to be unsuitable, or a better alternative is found. This packaging also allows higher power rated components like electrolytic capacitors and inductors to be compressed into a smaller form factor than would normally be possible by discrete components.

The disadvantages of brick converters are that they can be very hot and electrically noisy. All DC-DC converters can be noisy, but brick converters have no leeway in dealing with that noise internally so it must be addressed externally. These units also generate significant heat due to their condensed nature, and some require heat sinks or fans to keep them at operating temperatures.

Board to Board Connections

To meet the goal of modularity in this project several ways of connecting PCBs were researched. While there are numerous ways to connect distinct PCBs they generally fall into three different types: header connectors, cable connectors, and stand-off connectors. Header connectors are rows of pin connections that can be adjusted to suit different power and voltage requirements. These connectors can carry both power and logic voltage levels and can be specified to fit either. Their disadvantages are that they are rigid and need to be positioned precisely to connect to another header. An example of what a header connector looks like is included below in figure 5 [6].

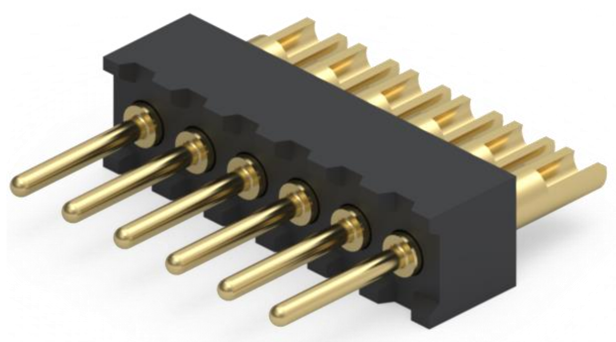

Figure 5: A high current rating header connector.

Cable connectors are ubiquitous and can be specified to carry any type of power signal. Some example formats of industry cable connectors standards are Molex, USB, and the broader term of ribbon connectors. The advantage of cable connections is their flexibility. They allow for board placement to be adjusted to fit any enclosure. The variety of connector types also allow for cables specific for any power rating. The disadvantage is cost and complexity since they require multiple components on each board to connect to compared to header and standoff connectors. An example of what some cable connectors look like are included below in figure 6 [7].

Figure 6: A Variety of Molex connectors.

Stand-off connectors are less common due to ease of using cables or headers. With this method board-to-board stand-offs are used to carry power from each board. This requires modification to the stand-offs to handle continuous conduction as many are not designed for this. The advantages of this approach are easy high-power connections, as stand-offs offer large conductive surfaces, and that they simplify the board-to-board connections. This simplification allows boards to be designed with smaller dimensions. One of the disadvantages are its cost. It is not a common solution, so finding suppliers who make stand-offs with copper or gold coatings can be difficult. Secondly, it requires users of the board to be careful or include extra non-conductive guards to prevent accidental contact to the exposed conductors. An example of this being used in a design can be seen in figure 2, as the PowerCube uses the stand-offs to route battery power to output connectors.

Chosen Design Approach

For this project it was determined that a combined unit supply would not allow for the modularity that is the projects goal. Therefore, the best option is discrete component design. For the 3.3V, 5V, and 8V boards this option offers flexibility with the layout and component selection to achieve the modularity and output goals. The 24V board designs investigated had issues with component size. Large capacitors and inductors are needed to achieve the power requirements. A brick converter was found that offers a smaller footprint while achieving the required output. This sacrifices some flexibility and forces the other boards to be larger to line up connectors but minimizes vertical spacing issues and simplifies cooling requirements. It was determined that the best board-to-board connector for the project would be a header connector as it is easier to increase its current rating if necessary and is the cheapest option with minimal parts needed.

Review of Chosen Approaches

There are a myriad of ICs available to do step down conversion. Because the previous 3.3V and 5V revisions circuit designs were found to be acceptable, there was minimal research time expended to investigate different ICs. The main consideration for these two were cost compared to other designs. Both designs can be found in single units for prices with comparable costs. An example of such a 5V converter circuit can be found below in figure 7 [8].

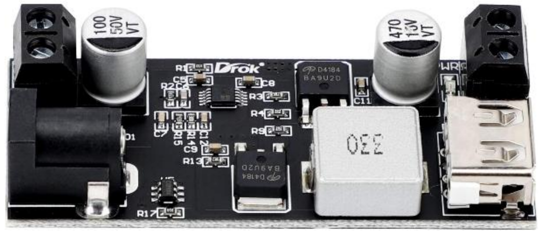

Figure 7: Drok 12V to 5V converter

A 12V to 8V converter is an uncommon design. There are few designs that offer single units with this output, and those that can supply the required power CfAR's design calls for are expensive. Many brick converters exist which can output 8V and 15A, but not at a nominal 12V input voltage. This means there are no truly comparable solutions. Conversely, there are many solutions for a 12V to 24V boost converter. This is a common conversion that is used in many industries. An example of such a converter can be found below in figure 8 [9].

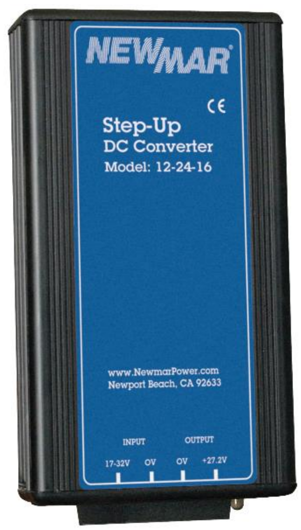

Figure 8: A 12V to 24V DC-DC boost converter.

However, options for a 12V to 24V converter skew towards brick converters and other single unit supplies like in that found in figure 8. This is because high power conversion is heat intensive and requires cooling to run at continuous operation. The sheer variety of options provide many avenues to meet project specifications.

V Team Duties & Project Planning

Roles in this project were mainly divided with regard to each person's strengths, with some bias based on what they wanted to do most. As the three members who are electrical engineering students and have practical experience from co-op work terms, Bryce, Dai, and Sean held the most responsibility for designing the converter circuits. As a computer engineering student who was less knowledgeable about concepts useful to circuit design, Kyle was most responsible for research into simulation using Altium and took on the bulk of the responsibility for project management. In terms of project management Kyle managed the team's Trello board, organized weekly team meetings, fostered discussion during the weekly meeting to focus on what was accomplished and what needed to be done next, compiled questions for project supervisors, and the deliverable documents to be submitted for the ECE 499 course. Sean aided Kyle in project management by being the primary contact between the supervisors Babak and Kieran and e-mailing questions to them as necessary, while Bryce kept track of meeting minutes during team and supervisor meetings.

Sean's primary deliverable was to redesign the 3.3V converter circuit. Since this circuits previous design was found to be effective, the provided four-layer design was tasked to be redesigned to be two layers with dimensions that fit the largest PCB that would be created, the 24V boost converter. As secondary deliverables, Sean researched an appropriate 8V IC convertor for Dai to use and did the 24V circuit's redesign. The 24V converter was also designed as a two-layer board since the IC only needed a small number of components for signal control to support its purpose. The main challenges encountered were learning basic PCB design rules and how to lay out components appropriately. Through email communication with the supervisors design reviews were done, with the suggestions being applied to the PCB and re-evaluated until satisfactory designs were approved.

Bryce's primary deliverable was to redesign the 5V converter circuit. Like the 3.3V circuit this circuits previous design was found to be effective and was also redesigned to be two layers with dimensions that fit the 24V boost converter. The challenge with the 5V design was converting a four-layer layout to two-layers without compromising noise mitigation and matching the recommended component placement [10]. Bryce's secondary deliverables were sourcing a 24V boost converter and assisting in the Altium design of the 8V converter. There was some difficulty in finding a 12V to 24V converter with that would satisfy CfAR's needs. 24V converters with a nominal 12V input are less common than 36V inputs which limited choice.

Dai's primary deliverable was to redesign the 8V converter circuit. This circuit required a complete redesign as the provided design did not work as intended, and the IC was impossible to replace. This circuit had the biggest deviance from its provided design and was therefore the most involved redesign. The main challenge with redesigning the 8V converter was finding an IC that could output high current, since this board was required to have a current output of 15A. Due to a lack of experience in PCB design, circuit analysis, and using Altium, significant research and time were invested to produce the final 8V circuit design.

Kyle's primary deliverable was to investigate using Altium for simulation. Since the 3.3V and 5V circuits were considered feature functional, and an off-the-shelf IC that nearly perfectly handles the design specifications for the 24V circuit was used, the 8V circuit was most interesting to simulate in Altium. There are other simulation software that could have been used, however, it was desired to show that it was possible to perform full development in Altium without needing to switch to other software suites. It was found that Altium was incredibly challenging to use for simulation in the intended context in part due to recent changes in Altium, and that simulating in an external suite like LTspice was more appropriate. Details regarding these challenges are further explored in section IX, Discussion & Recommendations.

VI Design Methodology & Analysis

Design Methodology Followed

With the goal of creating a modular PDB system, it was decided to follow a microcontroller shield structure. This design method uses PCBs dimensioned in a way that connector holes line up on every board that is desired to be stacked. This requires that the connectors chosen can handle the maximum load from the input supply in terms of current capacity.

A minor design consideration was the physical dimensions of the components selected. The largest component determines how much clearance each board requires for the next one to be stacked. This was ultimately not a hindering factor because of two solutions for large components. The first is that the largest board could simply be added as the topmost board in the stacked configuration. Second, since Altium physically places holes for components following their footprints, any connector with a similar footprint could be chosen if size was an issue.

As mentioned in section III, dimensions of the boards were not indicated as an important factor for the design. This was because each converter's complete circuit takes up a relatively small amount of area on the PCB. The largest board was the 24V converter. Since the 24V brick takes up the majority of the space on its PCB, additional space was only needed for signal filtering resistors and the connectors. This allowed the boards to have dimensions of 86mm x 82mm. The 24V PCB design as seen in Altium's 3D view can be seen below in figure 9. All other circuits use this 24V board as a reference for placing their connectors and dimensions.

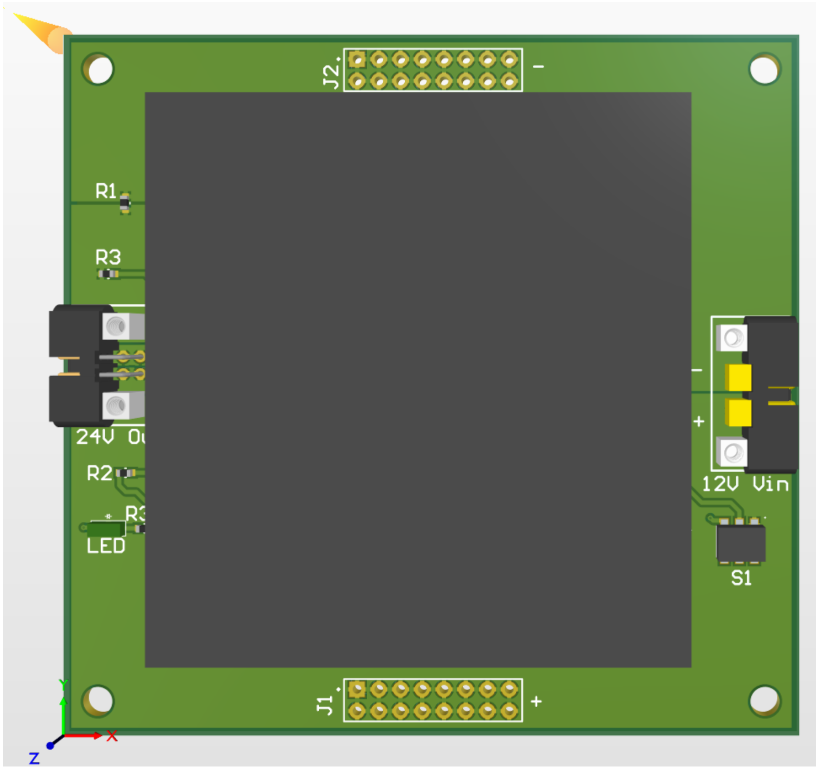

Figure 9: 24V boost converter board 3D Altium model.

For the board-to-board connectors, the ESQ-108-24-L-D connectors were sourced [11]. These connectors have an expected maximum current of 23A being provided. These connectors support up to 2.85A per pin, so it should be sufficient if the design includes at least 10 pins. A footprint that supports 16 pins was used to allow the design to be adapted if the voltage routing proves to be inadequate with the number of pins and allows freedom to source other connector options with similar footprints.

The input connectors for each board used the M80-5000000M5-02-333-00-000 connector provided in the given revision because it supports the high current load [12]. The output connectors were the M80-5400442 used on the provided 5V schematic because the outputs of the converters did not utilize the same level of current on the input [13].

This design also features each board being capable of accepting the 12V input source. If any particular converter is not necessary it can simply not be added into the system. Considering how simple each converter schematic was, this was a feasible feature to add to each circuit. It was also considered having exactly one board support input connections that could route power to the others, but this would have reduced the modularity of the system as that board would then always be required even if its supported output was not needed. Lastly, this design also allows for new converters to be designed and stacked as long as they adhere to the same dimensional requirements.

Other Methodologies Explored

There are other valid options for creating a modular PDB. One option considered was the use of a "base" board that other converter PCBs slot in to. Such a design would require an additional board to be designed that would only route the 12V input to four connection points for the other boards to connect to. This base board would be wide but reduce the height that the stacking method introduces. While this would work for the known converters being designed, any additional boards would not be able to be added to the system. They would have to replace one of the existing positions, limiting expansion options for the system.

Another option considered was to use ribbon connectors to bridge each input connector on the boards instead of using the connectors that allows for stacking. Since these connectors don't typically carry high current, they are not feasible for the high current needed. The benefit of having this type of connection method, if it were feasible, would be that each board could be designed independently of one another. Board size could be optimized, with additional freedom on where to physically put each PCB.

Component Selection Decisions

8V IC Sourcing & Redesign Decision

While the existing 8V design functions as intended in delivering the desired output, its IC, a TPS56121DQPR, uses the VQFN package which does not allow for easy removal in the case of a malfunction [14].

The initial redesign for this board used the LTC1890 IC [15]. This IC uses lead plastic MSOP pins, which means that the IC is easy to remove if desired. The datasheet for this IC does not have a reference design for 8V with at least 14A output, so an original design might have been needed. Upon further research, a reference test board "DC2236A-A" was found to use the same IC and generated outputs similar to CfAR's requirements [16]. This reference board was modified to output 8V. However, this circuit was very complex, and utilized the feature of the IC to generate two outputs. Using this design would take significant time routing connections in Altium that would be more manageable with four layers, increasing the manufacturing cost.

After receiving feedback on this circuit, it was decided that it was excessive for 8V design and a new IC should be sourced. Upon further research into the LTC lines of analog devices the LTC7803 [17] was found to be capable of achieving the desired input and output specifications. The datasheet also specifies that the IC is optimized for transportation, military, and avionic applications, which matches CfAR's desired use. Figure 10 below showcases the IC's efficiency for a 3.3V input for varying output current.

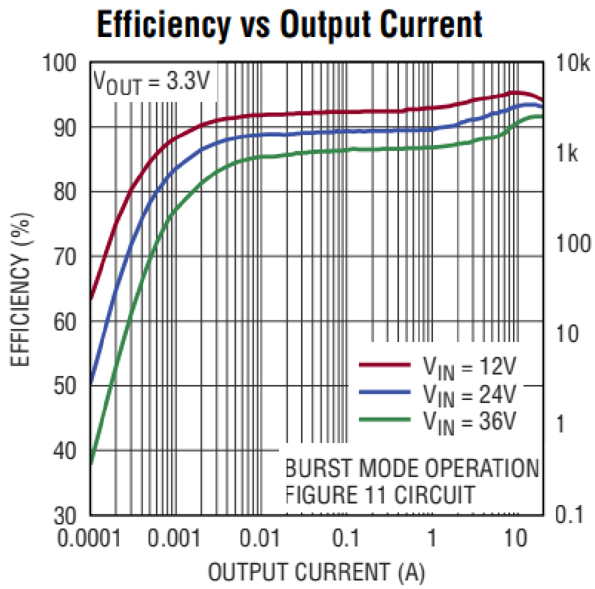

Figure 10: Efficiency vs. Output current graph of the LTC7803 at 3.3V output [17]

From this figure, it can be seen that the IC is most efficient with current output of around 10A. At 15A output current the IC should operate at around 95% efficiency. It is safe to assume that a similar efficiency could be achieved when outputting 8V.

Component Selection Decisions

To design the circuit schematic and select components, key formulas and concepts needed to be understood from referencing the LTC7803's datasheet [17]. A voltage divider using the IC can be seen below in figure 11.

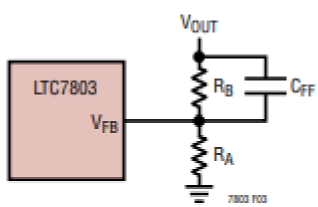

Figure 11: Voltage divider circuit at LTC7803's Vfb pin [17].

The output voltage can be found using the following formula.

$$Vout = 0.8V * \left(1 + \frac{RB}{RA}\right)(1)$$

Where R_B and R_A are the resistors as seen in figure 10. To achieve an 8V output the value for R_B is set to 180 KΩ and the value of R_A is set to 20 KΩ. The following equation can be used to determined maximum output current.

$$Rsense = \frac{50mV}{Iout}(2)$$

From equation 2, the R_{sense} value ideally should be set to 3.333mΩ to achieve a 15A output. To account for higher current output this value was set to 2mΩ in the design, which could allow an output of up to 25A.

The input capacitance should be set to account for the highest RMS current at the input to prevent large voltage transients. The output capacitance should be set to achieve an optimal effective series resistance. Using the ICs recommended schematic and using a similar input and output to our application, C_{in} was set to 77uC and C_{out} was set to 360uC. With these values, the circuit should be able to take 9-38V input voltage and output 12V. Similarly, the MOSFET BSC022N04LS6ATMA1 was chosen using the manufacturers recommendation, and the inductance of the inductor L1 was determined to be 4.7uH using a recommended schematic that has at 15A output current. Figure 12 below shows the Altium schematic using these component values.

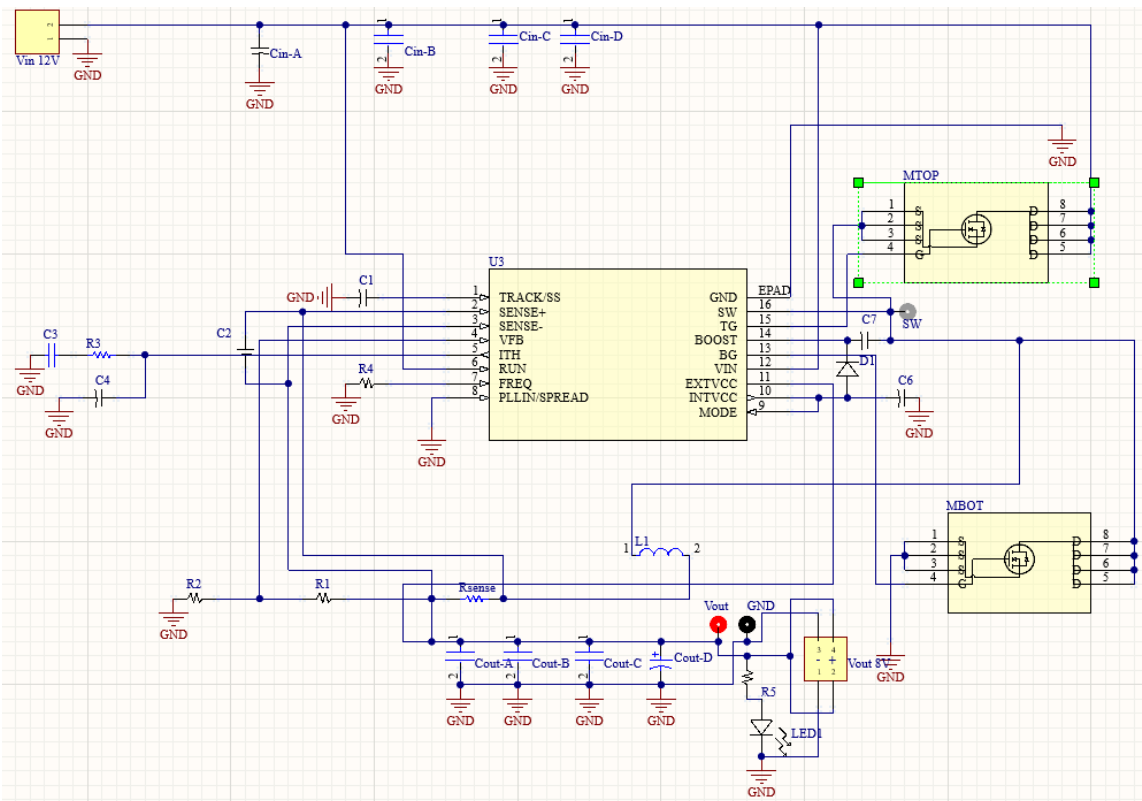

Figure 12: Altium schematic for 8V converter board.

PCB Design Layout

Since the 8V converter used a new design, the rules specified from the ICs datasheet were followed when creating the PCB to minimize noise and ensure optimal operation of the IC [17]. The design rules used are as follows.

- The ICs ground pin and the ground net of C_{INTVCC} are connected to the same ground terminal as C_{out}(-) terminal. The (-) terminals of the output capacitors are connected to the (-) terminals of the input capacitors.

- VFB pin's resistive divider connects to the (+) terminal of C_{out}. The feedback capacitor is not on the same feeds as the input capacitors.
- SENSE– and SENSE+ pins are routed together with minimum PC trace spacing. The filter capacitor between SENSE+ and SENSE– is positioned close to the IC.
- The INTVCC decoupling capacitor is connected close to the IC and between the INTVCC and the GND pin.
- The SW, TG, and BOOST nodes are kept away from sensitive small-signal nodes.
- There is a low impedance, large copper area central grounding point on the same side of the PC board as the input and output capacitors.

The board's dimensions were set to 86mm x 82mm to match with the 24V board's dimensions. The location of the four screw holes on each corner matches with the other boards to allow for stacking, along with matching board-to-board connector placement. The 8V PCB design as seen in Altium's 3D view can be seen below in figure 13.

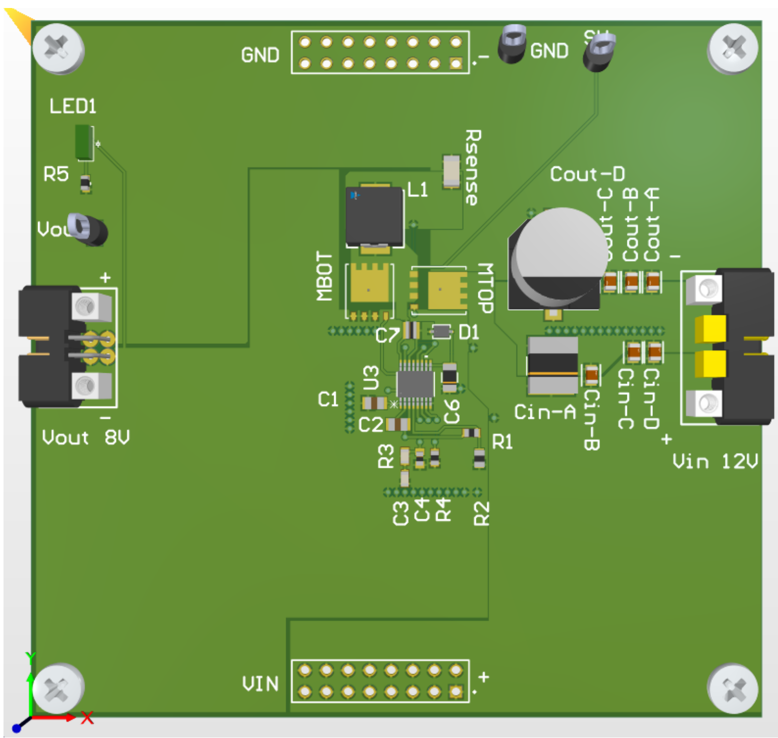

Figure 13: 8V buck converter board 3D Altium model.

Four-layer To Two-layer Design Changes

The provided revision used a single four-layer board that could support the differences in the converter layers, and reduce thermal strain on the board by using large power and ground planes. This methodology was adopted for the redesign of the 3.3V and 5V PCBs initially, but after a design review it was suggested to consolidate both into two-layer boards to save on cost since a four-layer design was not necessary with only one converter per PCB. This was successfully implemented, along with other suggestions to via styles and sizes, polygon layers, and silkscreen placement.

The 3.3V placement followed its ICs datasheet [13] almost exactly while the 5V had deviations. The recommended 5V layout was designed with a four-layer board to allow for power and ground connects to reduce the space required for the components. Lacking four layers means that the distances between the IC and inductor had to be increased to allow for signal traces. This is not expected to impact the operation of the 5V converter but if issues are found when testing the design, this may be a cause. Other factors in the layout of the 5V PCB were the increased AGND plane size, since it would lack the extra internal ground planes. The 3.3V and 5V PCB designs as seen in Altium's 3D view can be seen below in figure 14 and figure 15 respectively.

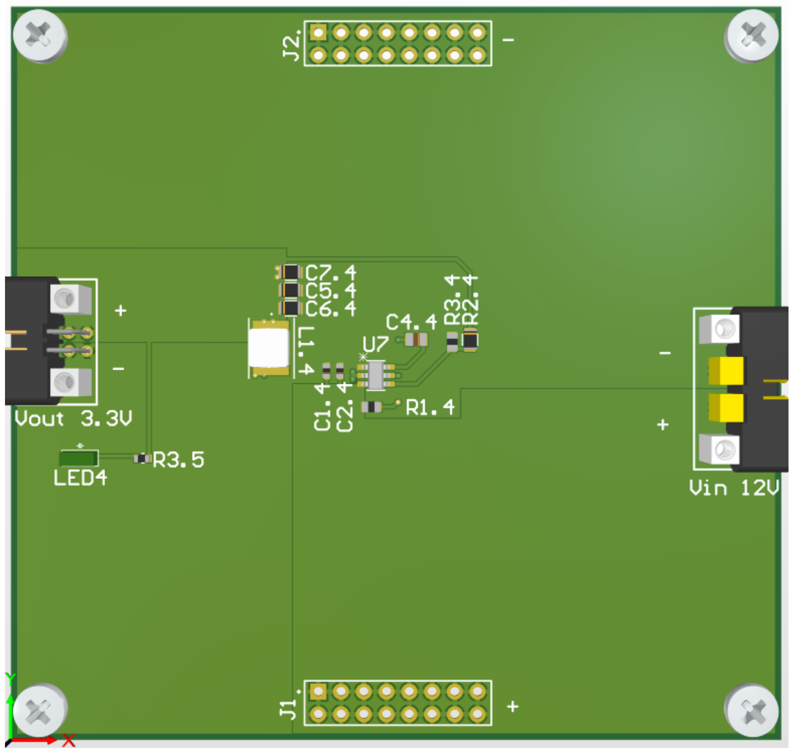

Figure 14: 3.3V buck converter board 3D Altium model.

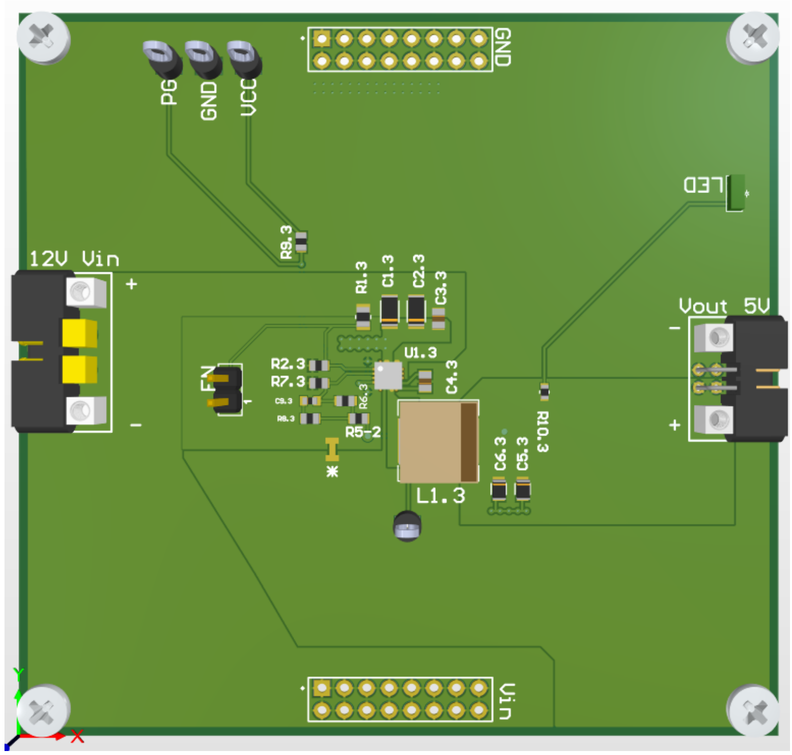

Figure 15: 5V buck converter board3D Altium model.

VII Final Design Details

Final Design Specifications

The PDB system consists of four separate voltage converter PCBs that by using board-to-board connectors are stackable and share a common input voltage provided by one connector on one of the four PCBs. From the feedback received for the final PCB designs they follow all the recommended rules for layout that minimize electrical noise, EMI, and other factors that would compromise reliable operation of the converters.

The operational boundaries are that each converter can accept an input voltage of 12V nominal and output its intended voltage of 3.3V, 5V, 8V, or 24V for each corresponding circuit. The 3.3V converter efficiency is around 80% [18], the 5V at around 92% [10], the 8V at around 95% [17], and the 24V at around 87% [19] with reference to each ICs datasheet's application efficiency graphs.

There are two constraints to follow when stacking the boards. First, the 24V must be the topmost board due to its large size and need for cooling the brick component used. Second, since each connector has no reverse polarity protection when connecting power or stacking the boards, the silkscreen labeled polarity must be followed or lined up. For example, the board-to-board connector with the label '-' for the 24V PCB must connect to the label '-' on the 8V PCB.

Goals Not Reached

There was originally intention of having the PCBs assembled and physically tested using a lab setup. This goal was removed when the progress for PCB design took longer than anticipated. This is in part due to remote working and communication via email having long turnaround times. Additionally, all team members had little or no experience using Altium or performing PCB design prior to this project, which led to time consuming redesigns when feedback was received. Things such as routing sizes, polygon layouts, via hole styles and sizes need to be specific for proper PCB design, so these parameters needed to be thoroughly researched. Another reason for this goal cancellation was that since most of the team members are not in Victoria it was not feasible to perform physical testing within the project timeline. Finally, the cost for some of the materials such as the 24V exceeded the $160 funding that UVIC will cover, which would require additional project funding to print and source components for each board.

VIII Testing & Validation

Ideally the manufacture files for each PCB designed would be generated using the Altium fabrication files outputs along with their BOM and uploaded to a site like Digikey for PCB manufacturing. However, due to limitations resulting from COVID-19, building a prototype was not feasible during this project. As such the next best method for determining if a PCB performs as expected is LTspice or similar simulation.

Typical Tests To Perform

The testing that would have occurred if the PCBs were able to be assembled would have been to use a power supply to supply a 12V DC input. Then, using banana wires and alligator clips, connect this input to the PCBs and use a DMM to take voltage measurements on the output pins where the connectors would go. Considering that the connectors for the input are not readily available on the current market direct connections to the PCBs holes where the connectors would go would suffice since this performs the same function that the actual connectors would use. An exhaustive list of test plans for each board is included below in point form.

3.3V

- Output test, check if 3.3V output is maintained across a 9V-15V sweep with a test load
- Check effect of maximum continuous current for spot heating and other damage
- Vary load and input voltage to see how output voltage changes
- Measure output noise

5V

- Maintain check on PG test point while conducting all tests (this test point checks the internal MOSFET’s drain for voltage in the expected range 0.9-13V and asserts low if it is out of range)
- Output test, check if 5V output is maintained across a 9V-15V sweep with a test load
- Check effect of maximum continuous current for spot heating and other damage

- Vary load and input voltage to see how output voltage changes
- Measure output noise

8V

- Output test, check if 8V output is maintained across a 9V-15V sweep with a test load
- Check output of SW pin to ensure that correct PWM frequency is being generated
- Check effect of maximum continuous current for spot heating and other damage
- Vary load and input voltage to see how output voltage changes
- Measure output noise

24V

- Output test, check if 24V output is maintained across a 9V-15V sweep with a test load
- Check effect of maximum continuous current to determine if active cooling is necessary
- Vary load and input voltage to see how output voltage changes
- Measure output noise

Combined Boards

- Test all board outputs with boards connected as a single stack
- Test effect of maximum load on all outputs on board-to-board connections
- Test to see if any input connection fails to run at maximum load
- Test what happens when all boards have an input voltage
- Re-test output noise on all outputs and noise at the input

The passing criteria would be measuring 12V input and 3.3V, 5V, 8V, and 24V outputs on their respective boards. If these were not measured to be within an acceptable range. Further measurements could be made on the IC pins, and visual inspections and short circuit testing could be used to confirm that no shorts from the soldering occurred during assembly. Another required testing point would be the board-to-board connectors that expect the 12V to be successfully maintained on all the boards that are bridged together without losing a significant amount of voltage on the board furthest away. To test various cases, random combinations of the boards would need to be put together with the board that is taking the input supply changed as well. This would ensure that any combination of boards can be used, and that any single board could act as the supply board for the others.

Simulation Performed

Simulation was not an explicit deliverable for CfAR. This is likely for a few reasons. The first was that the original expectation that the PCBs should be ordered and assembled in person. The second is that the 3.3V and 5V circuit boards have already been printed once before and can be verified to function. Redesigning the layouts would not affect this outcome. This leaves the 8V and 24V PCB designs to be simulated. Since the 24V design is effectively an off the shelf IC with a few signal filtering resistors, it becomes most important to look at how the 8V board simulates. In designing the 8V circuit, LTspice was used to verify its functionality. Figure 16 below shows the resultant graphed results of the circuit at steady state given a 12V input.

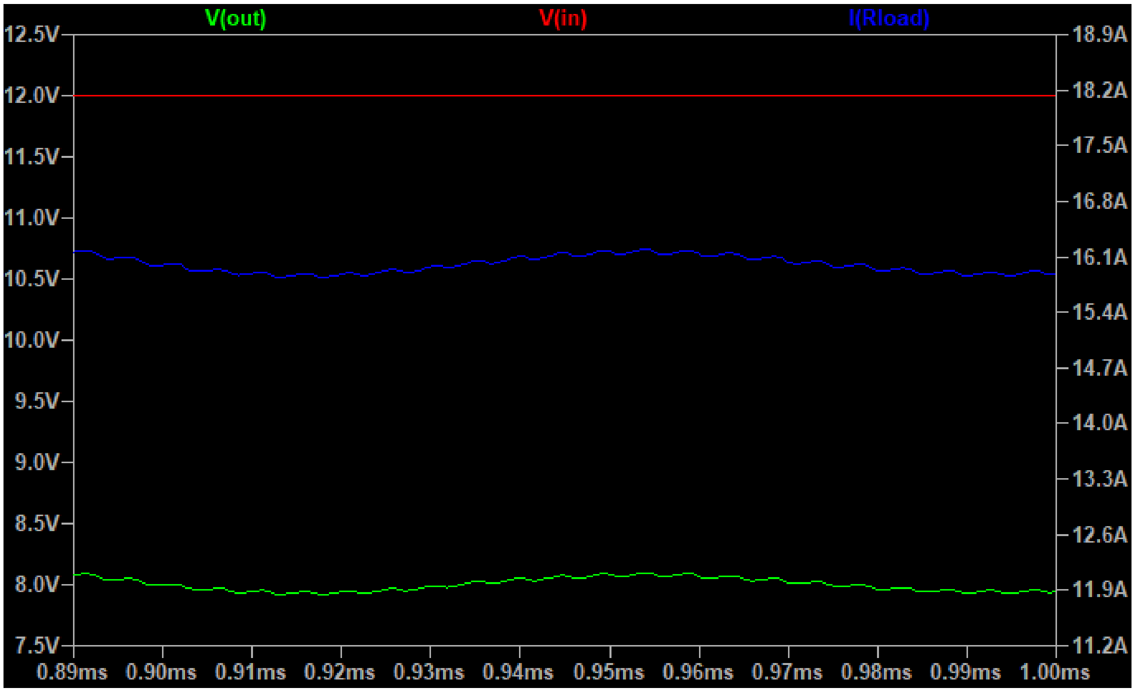

Figure 16: LTspice simulation of 8V converter at steady state.

The above simulation show that the design can take in 12V input and output 8V while being capable of supply more than 15A current at steady state. For curiosity since the supervisor from CfAR is not experienced with simulating circuits in Altium, an attempt was made to perform simulation in Altium. Since the LTC7803 is a new product on the market, a simulation ready model for use in Altium could not be sourced. There were other issues found with Altium simulation that is detailed further in section IX, Discussions & Recommendations.

IX Discussion & Recommendations

PCB Designs

The 3.3V converter PCB closely followed the IC's datasheets recommended layout for component placement and polygon layers. As such it is expected to perform similar to the circuit provided in the datasheet with similar observed voltage values. The 5V converter PCB also followed its ICs layout closely, with changes only made to accommodate the transition from a four-layer to a two-layer design. It is also expected to perform identically to the datasheets expected voltage values.

Since the 24V converter used a Single Unit Converter, the PCB was used generally to provide connection points for its pins and add filtering resistors, so it is expected to perform exactly as specified in its datasheet.

The design for the 8V converter followed the IC’s data sheet’s recommended layout, schematic and equations to ensure desired input and output as well as optimal working conditions for the IC. A simulation for the schematic was performed in LTspice which shows that the design worked as intended in theory. However, the LTspice simulation only used the default parameters for the capacitors, inductors, MOSFETS and resistors instead of manufacturer parts. Furthermore, this simulation does not account for noise and heating where physical testing is required to accurately quantify the effectiveness of the board.

Altium Simulation

Simulation in Altium was researched as a potential alternative to simulation in other environments like LTspice to show that Altium could be integrated into the design process past simply designing one's circuits. This research showed mixed results.

The first issue encountered was that many resources about simulation are currently outdated. Altium has had some significant revisions recently that have changed how you use Altium for simulation, what the UI looks like, and where some options are located. For example, Altium Limited had recently uploaded a video in November 2019 about using Altium for simulation to their YouTube channel, but most of the content refers to UI elements that have been changed and no longer exist [20]. Similarly, most other tutorial type materials refer to outdated versions of Altium. This presents a currently steep curve to using Altium for simulation, as those without prior experience have minimal information to learn from.

The second issue encountered was one of logistics. To simulate a component in Altium a simulation model is required [21]. This simulation model provides Altium with a netlist of how ports interact with each other and should be included with the footprint of a component. Manufacturers can generate these simulation models for their components, and there is compatibility for simulation models native to Altium called MixedSim, simulation models native to LTspice, or simulation models native to PSpice. Generic components such as resistors, capacitors, and more can be simulated with Altium’s built in simulation libraries then replaced with specific manufacturer components later.

The problem is with how Altium performs simulation with respect to the above. If a component in the schematic library does not have a simulation model it cannot be simulated. This is problematic because manufacturers are not required to include a simulation model along with a component footprint. Perfectly valid component footprints may not have associated simulation models, making them unusable for simulation. Being restricted by which components can be used in the design process to those with valid simulation models would make the design process take much longer. This was especially important when considering that the IC used in the 8V board simulates properly in LTspice, however, appeared to not have a simulation model when imported to Altium. It was speculated that this may be because the chosen IC is a product from Analog Devices, the developer of LTspice.

In summary, it appears that Altium could be useful for some simulation purposes such as designing smaller component footprints. However, it is less useful for simulating larger circuits that incorporate component footprints, since these component footprints may not have simulation models.

Recommendations

The size for the 3.3V, 5V, and 8V could be smaller since they were sized to match the large layout for the 24V PCB. This could be done by making sure that the board-to-board connections are lined up and only use the PCB space needed for the converter circuit and its input and output connectors. To remove the need to put the 24V converter on the top of the stacked system, a cooling system such as a powered fan could be added for forced airflow onto its PCB. This would allow any combination of boards to be used. Finally, adding reverse polarit and overvoltage protection into each converter would eliminate the need to make sure that the polarities of the board-to-board connectors are lined up correctly to avoid damage, and would not require an additional protection board for the input supply to pass through before reaching this PDB.

X Conclusion

The project goal for creating a PDB system that has a 3.3V, 5V, 8V, and 24V was successfully implemented up to the stage where manufacture files were able to be created and parts ordered that could be assembled into physical PCBs. With feedback received on the final design revisions, and simulation test results, the four converters are expected to function as intended when assembled. The limitations for the final PDB systems are that the 24V converter must be the topmost board in the PCB stack for thermal reasons, and the connections between boards and input power supply must follow the polarity indicators on the PCB silkscreen to avoid damage. While there were scope changes needing to be made and challenges from remote working, a modular PDB system was successfully able to be designed. Simulation testing in Altium is not recommended at this point of time for this type of system. Using simulation softwares external to Altium such as LTspice would be preferred.

References

[1] "UVic CfAR", Cfar.engr.uvic.ca, 2021. [Online]. Available: https://cfar.engr.uvic.ca/ [Accessed: 28- Mar- 2021].

[2] "Millsworth Engineering," [Online]. Available: https://millswoodeng.com.au/250w_pmu.html [Accessed: 20- Feb- 2021].

[3] "Quad Questions," Quad Questions, 2020. [Online]. Available: https://quadquestions.com/product/tbs-powercube/ [Accessed 27- Feb- 2021].

[4] "Selecting Passive Components with a Buck Converter," EEWeb, 2013. [Online]. Available: https://www.eeweb.com/selecting-passive-components-with-a-buck-converter/ [Accessed 20- Mar- 2021].

[5] "Brick Power Supply, Brick DC0DC Converter Manufacturer Cincon," Cincon, 2020. [Online]. Available: https://www.cincon.com/Brick_Converter.html [Accessed 20- Mar- 2021].

[6] "PCB Designs," 2015. [Online]. Available: https://pcbdesignsdl.blogspot.com/1969/12/high-current-pcb-header.html [Accessed 28- Feb- 2021].

[7] "Molex," Molex, 2021. [Online]. Available: https://www.molex.com/molex/products/group/power_connectors [Accessed 8- Mar- 2021].

[8] "Amazon Canada," Amazon, 2021. [Online]. Available: https://www.amazon.ca/DROK-Converter-Regulator-Inverter-Transformer/dp/B01NALDSJ0 [Accessed 20- Mar- 2021].

[9] "Step-Up DC Converters," NEWMAR, 2020. [Online]. Available: https://dcpoweronboard.com/12-to-24v-step-up-converters/ [Accessed 20- Mar- 2021].

[10] "TPS56637 data sheet, product information and support | TI.com", Ti.com, 2021. [Online]. Available: https://www.ti.com/product/TPS56637 [Accessed: 28- Mar- 2021].

[11] "ESQ-108-24-L-D - Elevated Socket Strip, 0.100" Pitch | Samtec", Samtec.com, 2021. [Online]. Available: https://www.samtec.com/products/esq-108-24-l-d [Accessed: 28- Mar- 2021].

[12] "Datamate Mix-Tek Horiz. M80-5000000M5-02-333-00-000 | Harwin", Harwin, 2021. [Online]. Available: https://www.harwin.com/products/M80-5000000M5-02-333-00-000/ [Accessed: 28- Mar- 2021].

[13] "Datamate J-Tek Horizontal PCB Connector M80-5400442 | Harwin", Harwin, 2021. [Online]. Available: https://www.harwin.com/products/M80-5400442/ [Accessed: 28- Mar- 2021].

[14] Ti.com, 2021. [Online]. Available: https://www.ti.com/lit/ds/symlink/tps56637.pdf?ts=1616882639231 [Accessed: 28- Mar- 2021].

[15] Analog.com, 2021. [Online]. Available: https://www.analog.com/media/en/technical-documentation/data-sheets/3980fa.pdf [Accessed: 25- Mar- 2021].

[16] "DC2236A-B: 12V @ 25A, 16 ~ 60V in, Buck", DigiKey, 2021. [Online]. A vailable: https://www.digikey.ca/reference-designs/en/ac-dc-and-dc-dc-conversion/dc-dc-smps-single-output/1771 [Accessed: 28- Feb- 2021].

[17] Analog.com, 2021. [Online]. Available: https://www.analog.com/media/en/technical-documentation/data-sheets/ltc7803.pdf [Accessed: 25- Mar- 2021].

[18] "TPS562200 data sheet, product information and support | TI.com", Ti.com, 2021. [Online]. Available: https://www.ti.com/product/TPS562200 [Accessed: 28- Mar- 2021].

[19] "CHB200W", Cincon.com, 2021. [Online]. Available: https://www.cincon.com/productdetail/CHB200W.html [Accessed: 28- Mar- 2021].

[20] Altium. "Simulating your Design in SPICE - Jackson," 16 – Nov - 2019. [Online]. Available: https://www.youtube.com/watch?v=W9kPCPHfpY0 [Accessed: 28- Mar- 2021].

[21] "Managed Simulation Models | Altium Designer 21 User Manual | Documentation", Altium.com, 2021. [Online]. Available: https://www.altium.com/documentation/altium-designer/managed-simulation-models-ad [Accessed: 25- Mar- 2021].