DeepCharge

Final Report
April 24th, 2020

Jacob Londa
Project Lead

Anthony Rizzo
Build Studio Engineer

Anuj Gajjar
Build Studio Engineer

Charlene Ng
Build Studio Engineer

Eliot Smullen
Build Studio Engineer

Michael Komatz
Build Studio Engineer

Table of Contents

COVID-19 Note

Please note that this project took place during the start of the COVID-19 pandemic. Thus, the scope and goals changed significantly about half way through the semester. Due to lack of access to prototyping tools and equipment, as well as orders of quarantine, the project shifted to focus on the design side of everything, rather than physical prototypes. The new scope included designing the electrical schematic and PCB layout, and modeling the system in CAD. During this time, Generate also allowed members to reduce or eliminate their workload for the remainder of the semester if desired.

Executive Summary

The client, DeepCharge, is a startup based out of a Northeastern University research lab, focusing on next-generation wireless charging. They are in the process of creating an intelligent multi-device wireless charging pad. The client is also using this same technology to prototype a UAV (unmanned aerial vehicle, or drone) charging pad, called the FlyPad. Currently, UAVs that are used for emergency or time critical purposes suffer from poor battery life. Once the battery is low, the UAV must return to the pilot to recharge or change batteries. This interrupts missions where maximum in-flight time is very important. Thus, the FlyPad is a portable landing pad that can wirelessly and automatically charge the UAV, without a need for positional accuracy.

Upon first meeting with the client, they had a breadboard and PCB proof of concept, and an industrial design CAD model for the overall look and feel of the FlyPad. DeepCharge requested help from Generate to further the development of the FlyPad into a cohesive system. For the internal electronics, since the charging coil modules were being developed by the client, Generate was tasked with designing the power distribution system. This system is responsible for the proper distribution of power from the batteries to the client's coil modules. Firmware would monitor the status of these electronics and report it to the UAV. For the mechanical side, Generate was tasked with continuing and improving the current design. This includes the structure and placement of all internal components, while using the industrial design concept as a guide.

In terms of learning opportunities, the team would be able to learn about how to build off of a product that is at a significant stage in the product development cycle. This includes more interaction with the client about how to integrate their technology, designing with a manufacturing technique in mind, and balancing progression vs. redesigns. Other potential topics include power electronics, injection molding, bluetooth communication, wireless charging, and heat management.

Background Research

In the robotics and UAV charging industry, there are a few competitors that are creating similar products. Wibotic is using magnetic resonance charging, and Skysense is using conductive-based contact charging. Conductive-based charging is not particularly safe or cost effective, because the entire surface of the pad must be covered in metal. Normal magnetic resonance charging is ineffective for drones because exact positional accuracy is needed during landing on the pad, in order to precisely line up the charging coils. A network of coils may solve this issue, but high power consumption during idle operation is the result, which is a huge limitation.

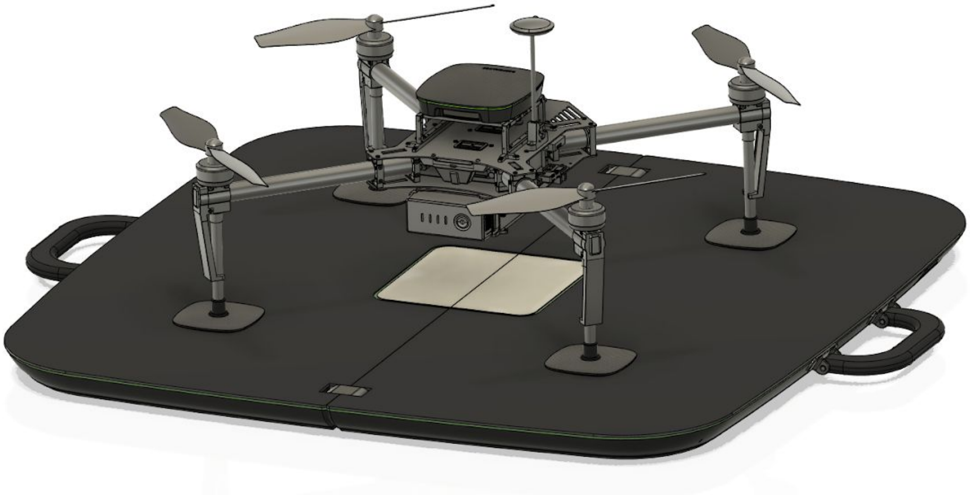
The client has developed a patent pending induction-based wireless charging technology that enables multiple devices to charge anywhere on a surface, all while reducing the total energy consumption. It is based on a network of sensing coils that can accurately determine the position of a device based on a machine learning algorithm. Then, the power is delivered to that position through another network of charging coils. This technology enables the charging of a drone when stationed on a landing pad, without the need for exact positional accuracy. Power is also only applied directly where the drone is positioned, not on the entire pad surface, thus saving energy. These are the main competitive advantages that the client has identified.

The coil networks and charging technology that the client created were pretty complex and already developed. Thus, it was decided to not work on that aspect of the project. This also avoided any time or work conflicts between the client's research team and Generate. The coil network modules were treated as a "black box", with certain power inputs that constrained the outputs of the power distribution system. This made it easier for Generate to just focus on the power distribution system, and let the client still work on their coil network modules, all while specifying an interface to join the two systems in the future.

The client had previously collaborated with a professional design firm to create an industrial design model of the FlyPad. This provided an overall design constraint for the mechanical team, because the client wanted to keep the specific aesthetic and form factor from this model. Although this model was a great resource for identifying the look and feel of the FlyPad, there were multiple problems with using it to continue development. Industrial design models tend to use "hard-coded" CAD constraints to render a desired geometry. However, this is not a good modelling technique if the design is expected to change a lot. Because the design of the FlyPad was still very much in development, Generate came to the conclusion that creating a new model would be better to continue engineering development. This new model would allow the mechanical team to respond quickly to design changes, which was bound to happen because the entire power distribution system and coil modules were still being developed.

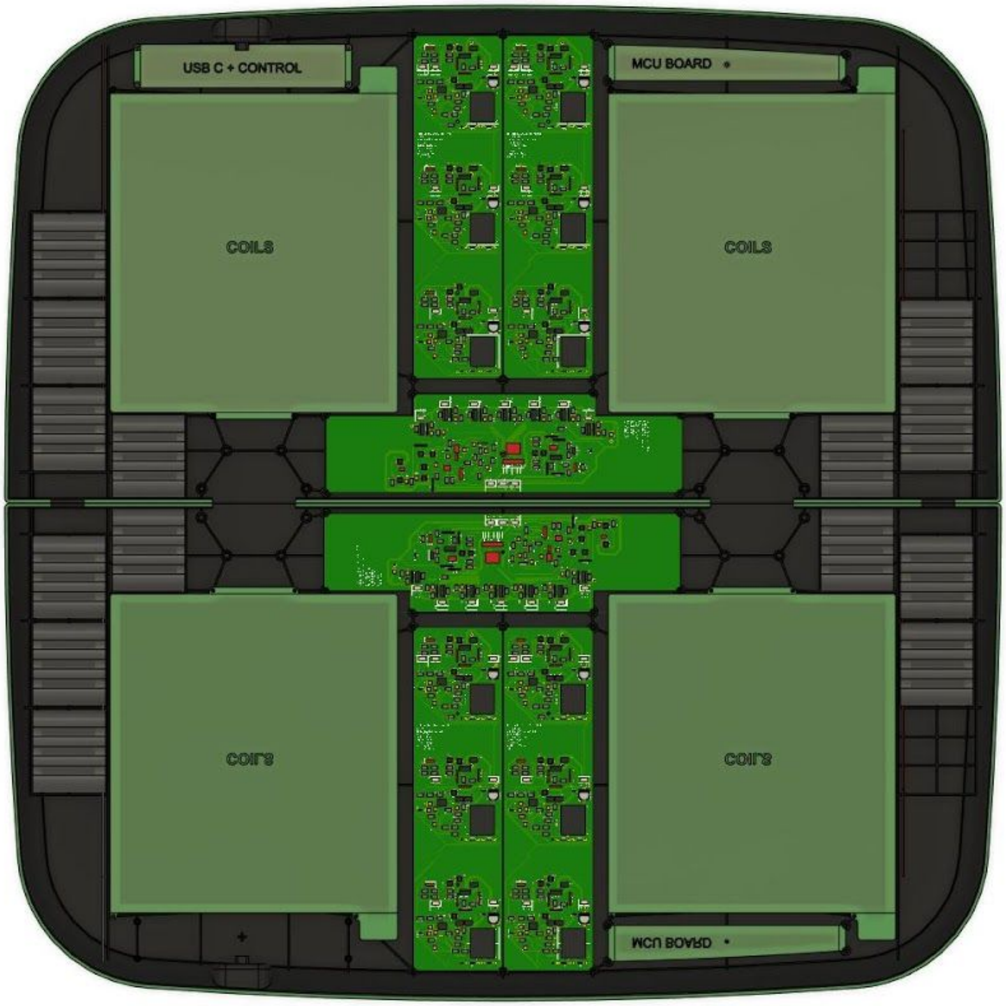
Systems Overview

The FlyPad is a product concept from DeepCharge, which uses their patent pending wireless charging technology to solve the difficulties of UAV battery charging. Currently, UAVs that are used for emergency or time critical purposes suffer from poor battery life. Once the battery is low, the UAV must return to the pilot to recharge or change batteries. This interrupts missions where maximum in-flight time is very important. Thus, the FlyPad is a portable landing pad that can wirelessly and automatically charge the UAV, without a need for positional accuracy during landing.



In terms of its overall shape, the FlyPad has a large, flat, rectangular design, with dimensions of 915mm x 915mm x 75mm. The pad is wide enough to span the distance between all four legs of the DJI Matrice 100, which is the test UAV for this product. A design constraint outlined by the client is portability, so the pad is able to be folded in half when not in use. This results in a smaller footprint during transport, which increases portability. Handles are attached to the outer edge of each half of the FlyPad, or "wing", so that a user can carry the pad like a briefcase when it is folded in half. The height, or thickness, of the FlyPad is constrained by the internal electronics of the system and overall structural integrity. A smaller height would result in a sleeker design, but would result in less space for internal components. The details of the mechanical system are further discussed in the Mechanical Design section.

The FlyPad has four internal coil network modules, which are each a large square PCB positioned near the top surface of the FlyPad. These modules contain the sensing and charging coil networks that enable the multi-device and energy efficient wireless charging technology. These four modules are placed in each quadrant of the FlyPad, so that each leg of the UAV will land on top of one module during landing. Each leg of the UAV will have a receiver charging coil, which receives power from its respective coil network module in the FlyPad. Each leg receiver is connected to a "backpack" on the UAV, which is responsible for summing together the power from all four legs and charging the UAV battery with this resultant power.



The power source of the FlyPad comes from several internal batteries. The power distribution system is responsible for properly transferring the power from the batteries to the four coil network modules. These modules require three different voltage levels, so the distribution system splits and converts the battery sources accordingly. This power distribution system is also responsible for monitoring the current draw from these modules, and has over-current protection to avoid damaging any components or overheating. The details of the internal electrical system are further discussed in the Electrical Design Section

Mechanical Design

Overview

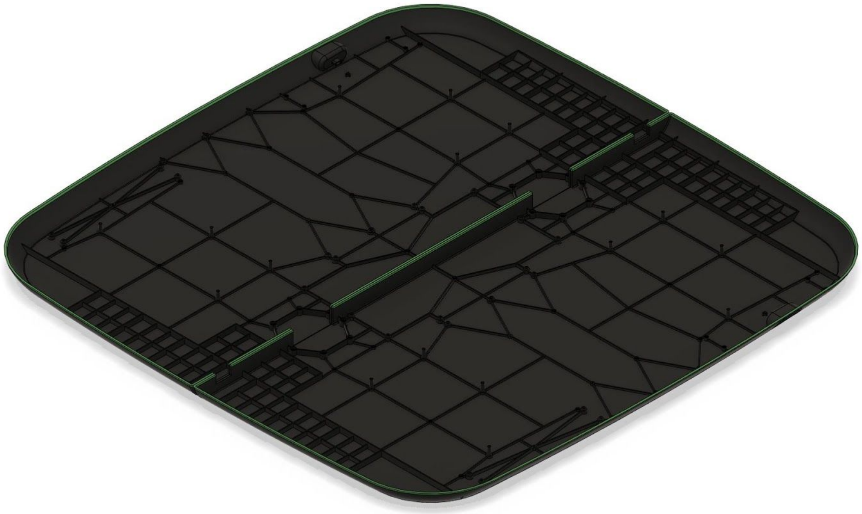
The mechanical design of the FlyPad was largely driven by the client's vision for the industrial design and user experience of the product. We started by translating this vision into a list of design considerations, which prompted additional investigation into the user experience of the FlyPad. In order to better understand the size, weight, and shape of the product, we built a low-fidelity prototype out of plywood. We then planned to build a higher-fidelity prototype to provide a platform for our electrical and computer engineering team to test their designs. This normally would have been an iterative process of designing, building, and testing to develop the mechanical components. However, due to the sudden limitations of remote work, we shifted to fleshing out the CAD model with particular focus on manufacturability.

It is important to note that the designs we've come up with should be tested to ensure performance. There are also aspects of the mechanical design that have not been considered due to time constraints and the in-flux nature of other subsystems. Our thoughts on future mechanical design considerations can be found in the *Remaining Design Considerations* section.

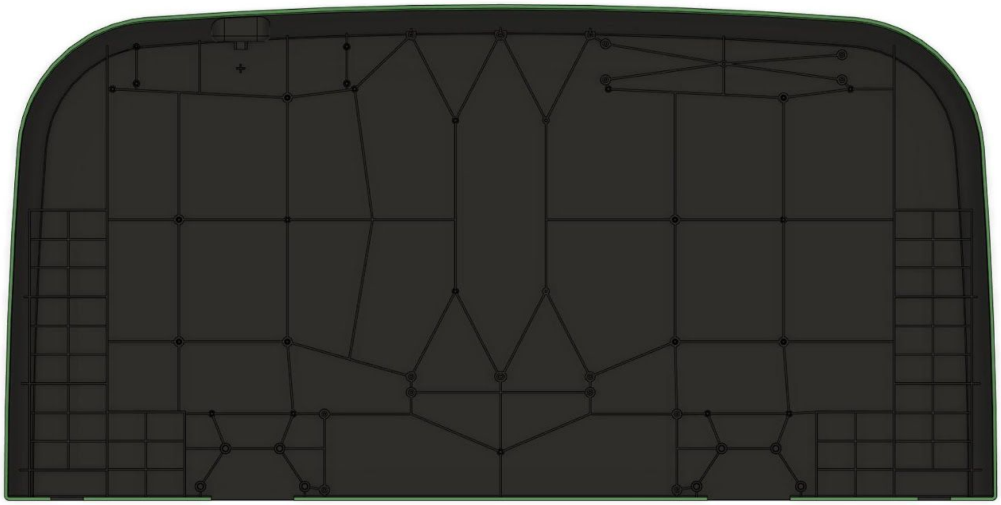
Components

Bottom Enclosure

The design of the enclosure required balancing the sleek and minimalist form-factor of the industrial design concepts with the need to contain the many internal components that enable the FlyPad's functionality.

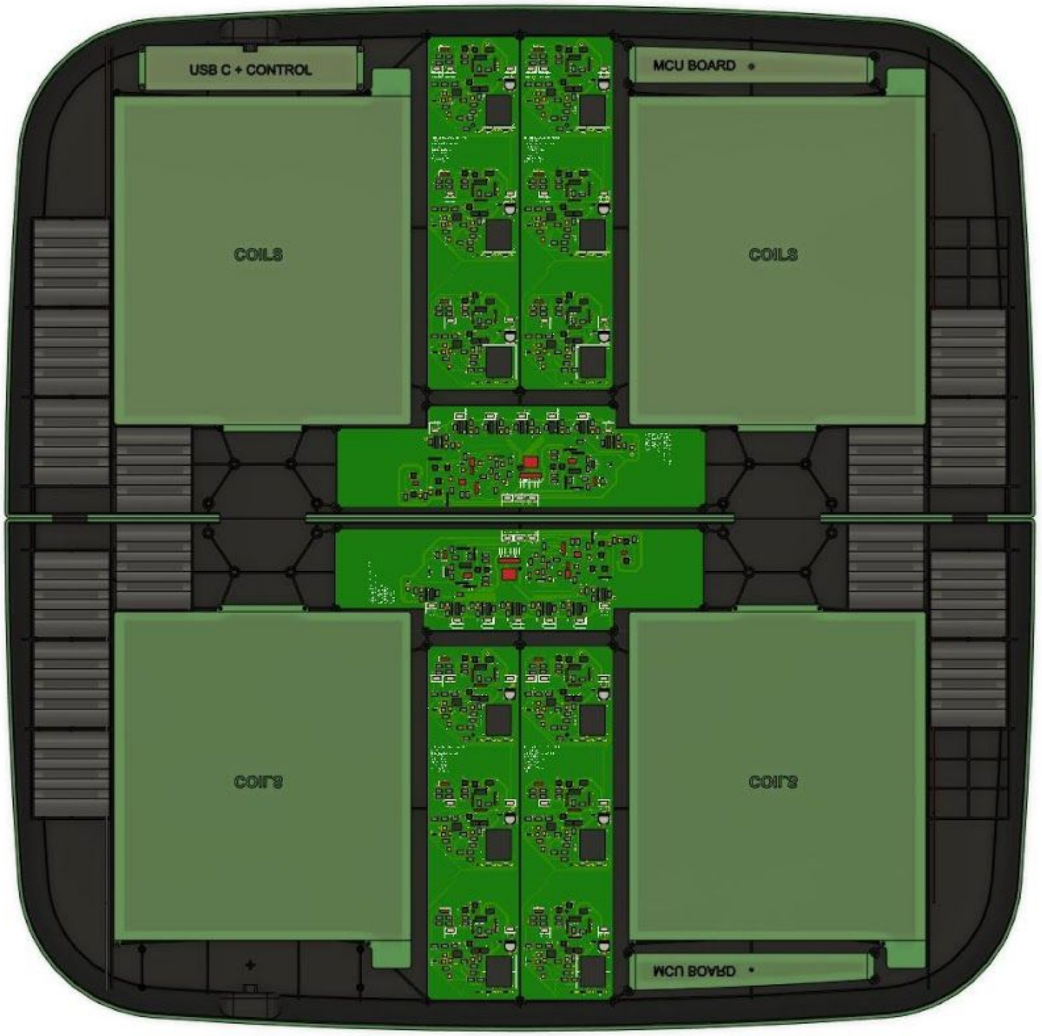


The bottom enclosure is a single, plastic part designed to be injection molded. This was chosen because of the versatility it would allow for the enclosure. It enables the creation of non-rectilinear shapes, such as the exterior of the FlyPad. It also can provide stiffness and durability to withstand the expected operation. It supports the addition of locating and mounting features to hold the internal components as well. All of these elements are combined into a single part.

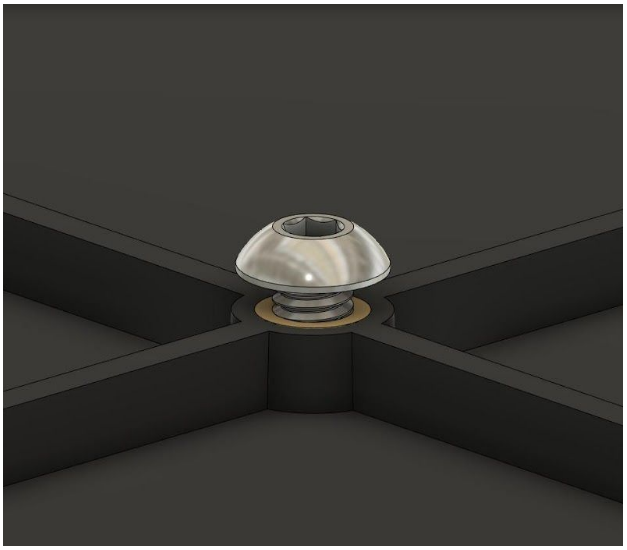


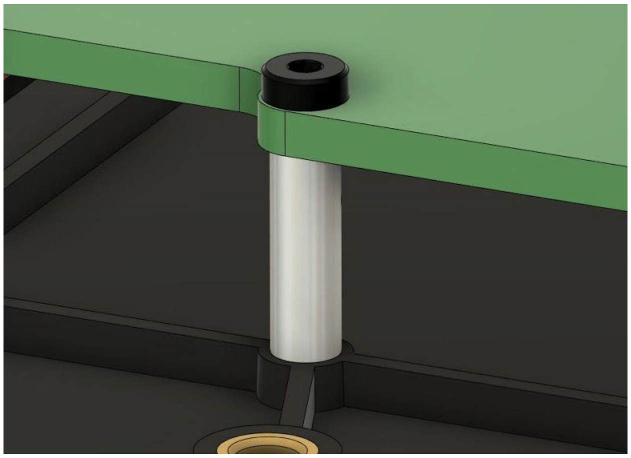
Since the FlyPad folds in half, two bottom enclosures will be needed. An important design decision was making the bottom enclosures identical. This avoids the need for two sets of tooling to produce negligibly different parts. Because tooling for injection molding is quite expensive, minimizing the number of unique injection molded parts is suggested.

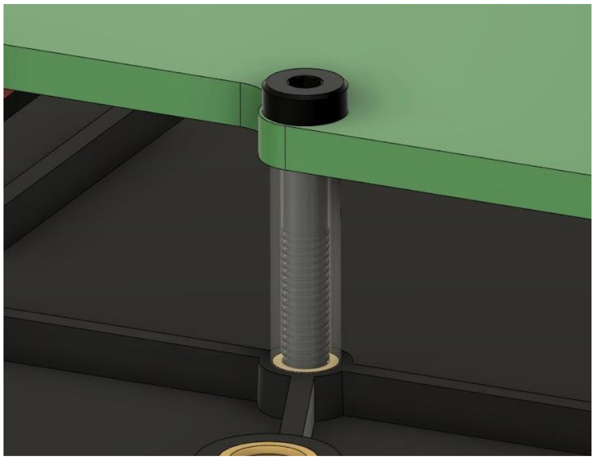
When arranging internal components within the bottom enclosure, we had to account for using the same part for both halves. The arrangement chosen is symmetrical across the hinge axis, except for the addition of the USB-C board only on one of the halves. The port on the non-functioning half can be plugged with the addition of another small part.



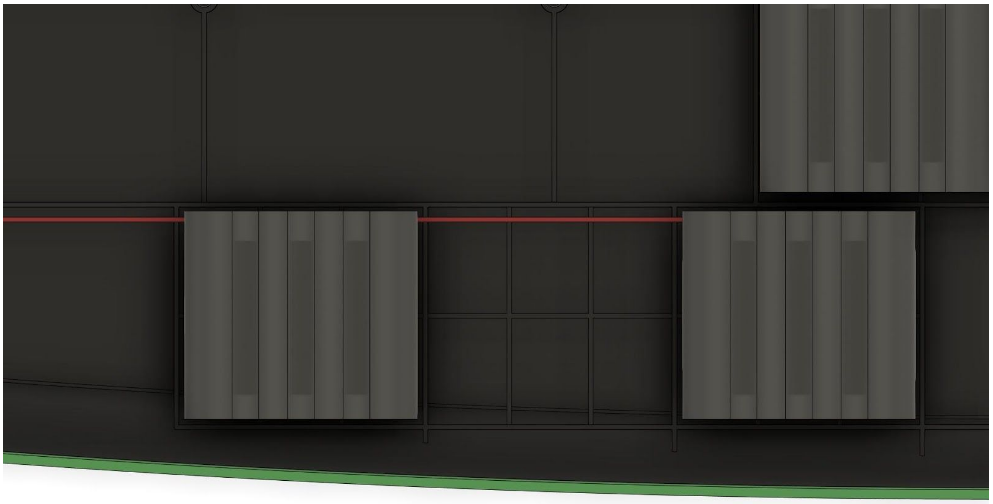
We added bosses to the internal enclosure to attach the boards onto using screws. There are fewer boss/screw features for boards that we did not expect to experience loads (DC/DC, MCU, Battery board), and more of those features for boards that experience load or require positional accuracy (USB-C plug, coil modules).







Ribs were added to the inner bottom face to provide stiffness to the enclosure. In some cases, these ribs functioned as positioning features such as creating “nests” for the batteries to sit in.



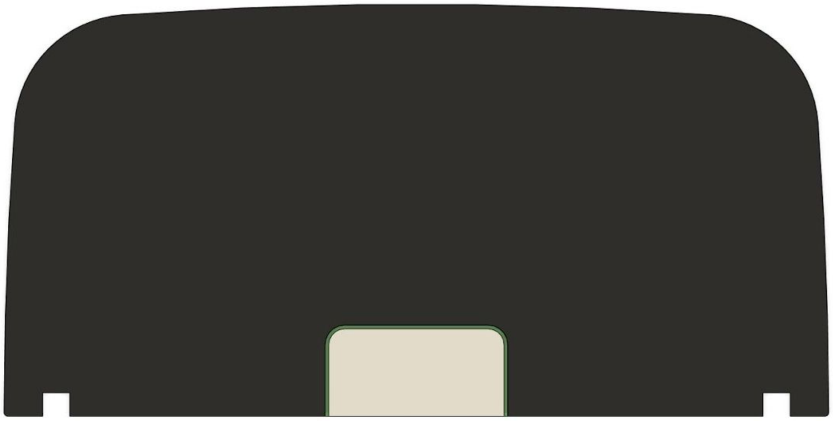
Additional holes and mounting features were added for the attachment of three other components: the hinges and stiffeners, the wire hinge, and the handle mounting brackets.

Due to the high-stress nature of the bottom enclosure caused by its large size and folding capabilities, we expect there to be a large amount of stress around the hinge points. The hinge stiffener was our attempt at mitigating this anticipated stress.

An alternative design can be considered for creating the bottom enclosure. Instead of an injection molded part, an alternative is to create a structural framing out of bent metal sheets or extrusions. Fastening features for the hinges, handles, and internal components could be added via pem-style nuts, rivets, or other means. To maintain the organic shape of the exterior of the FlyPad, a vacuum-formed plastic cover could be made that attaches to the structural framing. This cover would be like the skin to the bone that is the framing. This alternative design may provide more structural support for attaching the hinge, handle, and internal components, however due to time constraints it could not be explored to its full extent. It is recommended for the client to further explore both of these options, assessing them on feasibility and manufacturability, before going forward with one design.

Lid

The lid of the FlyPad functions as a cover for the bottom enclosure and a platform for a UAV to land. Similar to the bottom enclosure, two identical parts makeup one FlyPad.



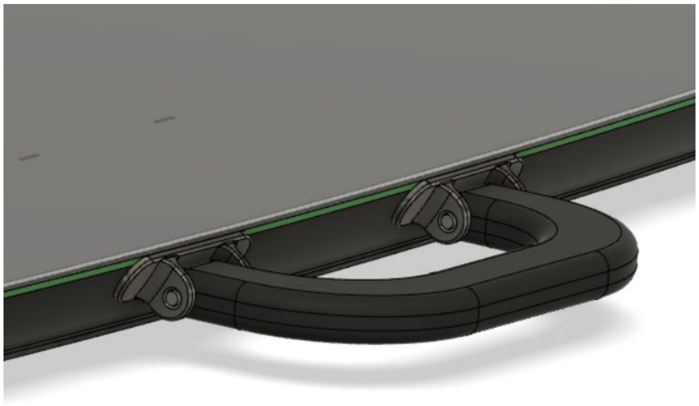
Due to the large footprint but flat and thin geometry, the lid would be a good candidate for CNC machining. Additional features are needed to allow for the lid to be secured to the bottom enclosure. The desire to maintain a plain top surface makes adding features to achieve this difficult. However, we recommend looking into using a jig to position and adhere standoffs to the underside of the lid.

The lid staying attached to the bottom enclosure is critical to the functioning of the FlyPad. As such, the method of gluing standoffs to a part may not suffice (prototyping and analysis is recommended). A potential alternative could be to make an injection molded part and add boss features. This could mate to a bolt through the bottom enclosure, which would secure the lid in place. Thermoforming a lid with attachment features is a potential option as well. A simpler solution that would prevent the need for additional hardware would be snap-fit features. However, they would prevent ease of disassembly if maintenance was needed.

A compromise to achieve strong attachment of the lid at a lower cost than an injection molded part would be to allow for small raised areas around the edges of the lid. These raised areas are where standoffs could be implemented to capture the top surface and bolt to the bottom enclosure. This could be designed in such a way as to ensure these raised areas were around the perimeter of the lid and not likely to interfere with the UAV landing zone or charging capabilities.

Handles

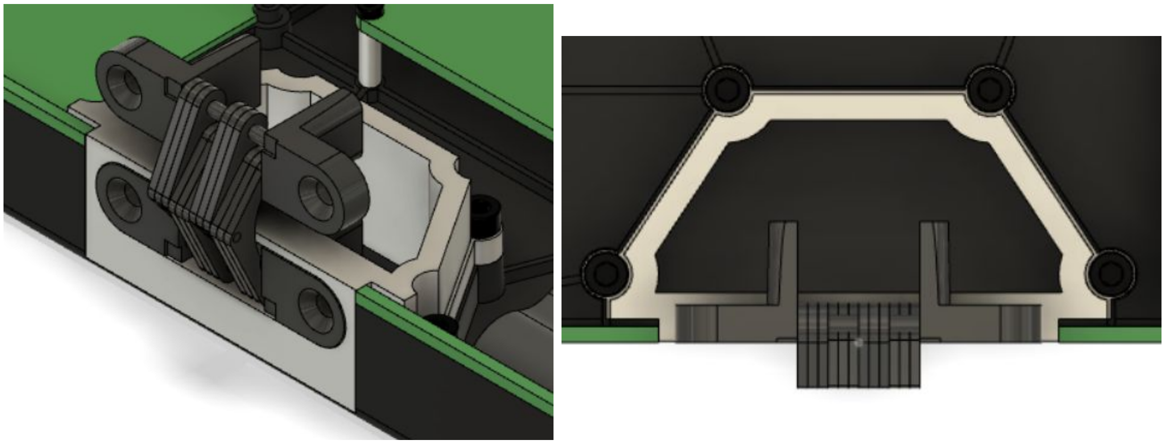
The location of the handles was moved from the hinge axis to the outside edges of the pad to make them easily accessible when the pad is open. The handles were also placed as close to the lid as possible so that when the pad is closed, they are not located too far away from one another and can be held with one hand. Considering how the part would be manufactured, the handles would be an plastic injection molded part that would be attached with metal brackets. This metal bracket would then be attached to the flypad using screws. Since the handle would be experiencing a lot of stress, the handle would require a stiffener similar to the hinge stiffener (mentioned in the hinge section). However, the handle stiffener was not added to the design due to space constraints of the pad.



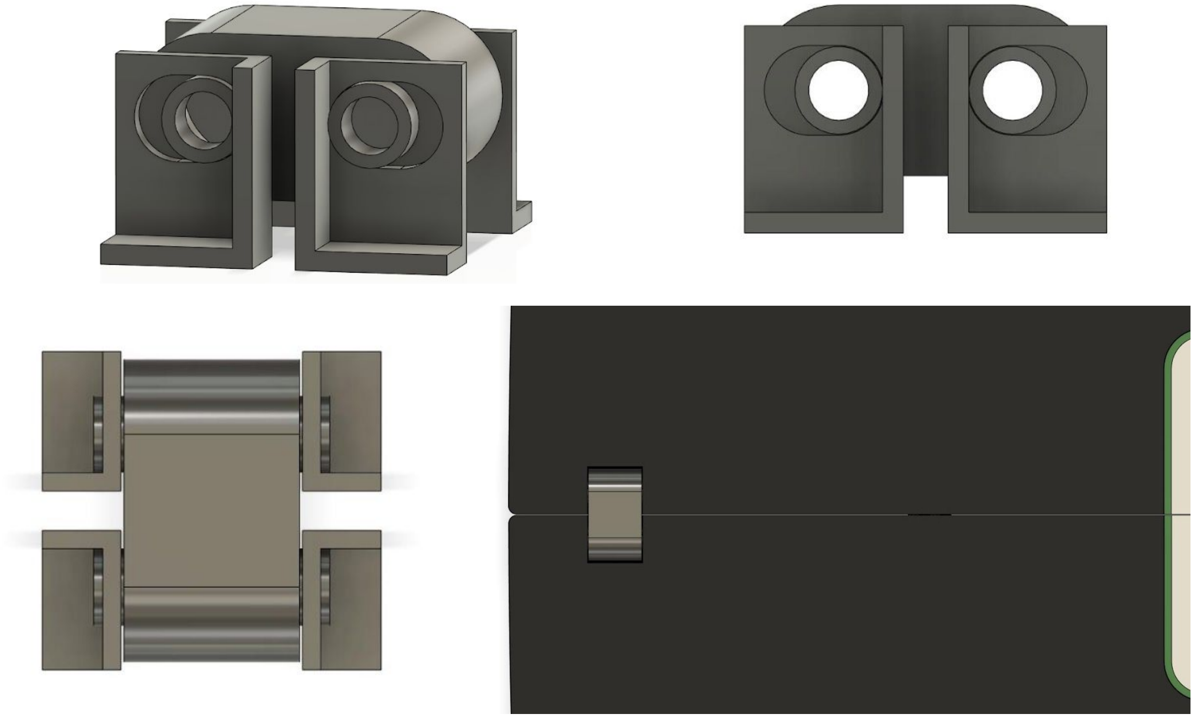
Hinges

Because of the folding nature of the pad, 2 hinges are necessary to attach the two wings. A design consideration in hinge selection was the need for a concealed hinge. This was necessary, as in order for the QR code to function, the surfaces of the two wings needed to sit flush with one another. Another design consideration was to have the hinge be a friction hinge. This was to improve the usability of the pad, as having a friction hinge would allow the two halves of the pad to slowly open when unfolding, preventing accidental damage to the pad if one side were to be dropped suddenly. A hinge that would produce both of these functionalities is a dual axis torque hinge. These specialty hinges were not able to be obtained in small quantities, so for testing purposes, a concealed hinge without friction was chosen for the design.

The part of the enclosure that is in direct contact with the hinge was determined as an area of high stress, as it needed to support the weight of all the components when the hinge was folding and unfolding. As a result, a hinge stiffener was added to create better attachment points between the hinge and the enclosure and distribute the stress to a larger area of the enclosure. The hinge stiffener would be machined out of metal, and attached to the enclosure with screws, and the hinge would be attached to the stiffener using screws and bolts.



A non-load-bearing “wire hinge” was created to accommodate wires running across the folding axis. We looked to the design of laptop hinges for inspiration to accomplish this while minimizing exposure to the elements. What we designed was a hollowed feature that spans the bottom enclosure and allows wires to pass through. It was important that this hinge did not interfere with the functionality of the other hinges, so this hinge “floats” - enough clearance was added for this “hinge” to be able to move in place, despite the position or movement of the two halves of the FlyPad.



The design of the wire hinge still needs refinement and prototyping, but if this concept is pursued it is recommended that the parts be made of cast or CNC’d metal.

Remaining Design Considerations

Listed below you will find our thoughts on aspects of the mechanical system we did not work on due to time constraints, bandwidth, or need for more developed subsystems.

Wire Routing

The wiring and routing of the many internal components will require some serious thought and creativity. Changes will have to be made to the bottom enclosure to accommodate the wiring. Additional features will need to be added as well, such as strain reliefs. We recommend positioning the connector ports on the boards as close as possible to their end destination to minimize the total length of wires.

Latching

We recommend some sort of latching feature to ensure the pad remains folded when being carried and transported. Based on our experience with the plywood prototype, it won’t be an easy task to carry the FlyPad. A simple latch would mitigate any hassle caused by the Pad inadvertently unfolding during transportation.

Heat Dissipation

The relatively high wattage produced by the electrical system is certain to give off noticeable heat. It is important for the safety of the users and performance of the components that an analysis thermal behavior is done to ensure the FlyPad will not fail, overheat, or catch fire.

Based on our intuition, we expect the need for a fan cooling system to aid in the circulation of air around and out of the internal part of the enclosure. This would require a redesign of the internal component arrangement, as well as shrinking of some component sizes where possible.

Electrical Design

Initial Goals

The initial goal of the electrical design was to design a power distribution and management system that interfaces with a microcontroller and provides user feedback to the system. Another goal of the design was to manage a battery system that could theoretically recharge a typical DJI drone twice (without needing to recharge these batteries). So the initial design was to design a battery management system, power management system (that converter the battery voltage to the three desired isolated voltage levels, a control system that interfaced with the management systems, and user feedback to a user. Figure 0 shows the system level design devised for this power distribution system.

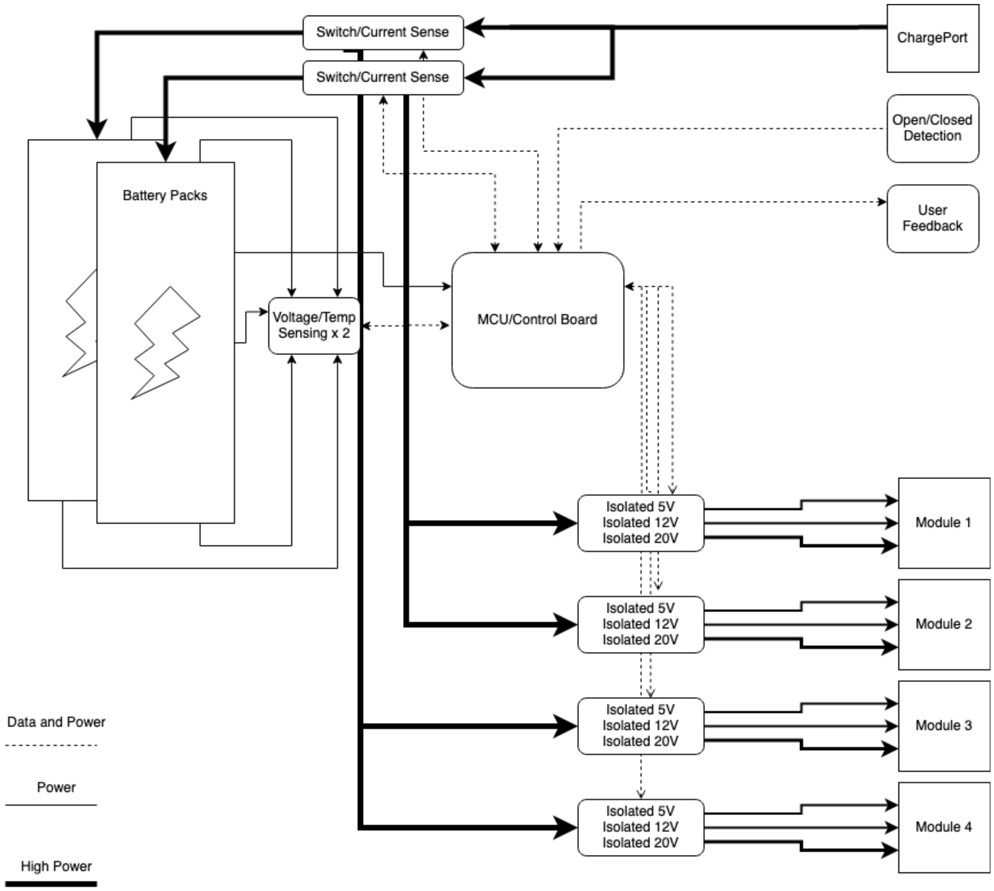


Figure 0. The initial system level design of the electrical systems.

However we had to revise our plan halfway through the semester, and we could not implement any user feedback and a microcontroller system. Our main goal was to make the primary electrical design (for the battery management system and power management system).

However, despite not implementing a microcontroller system and user feedback, one of the goals with the design is to allow access points for a microcontroller system to ‘understand’ the power distribution system (through voltage, current, and temperature sensing) and to communicate with any user feedback systems, based on the voltage, current, and temperature data. The electrical design described below should be easy to modify, and should provide a base for a comprehensive power distribution design.

System Level Design

Before physically designing, the following system level design was devised for the overall power management system. In this system, the overcurrent protection modules help manage the power output of the system and protect against any power/current surges. There is also an attached charging and cell balancing system for battery management.

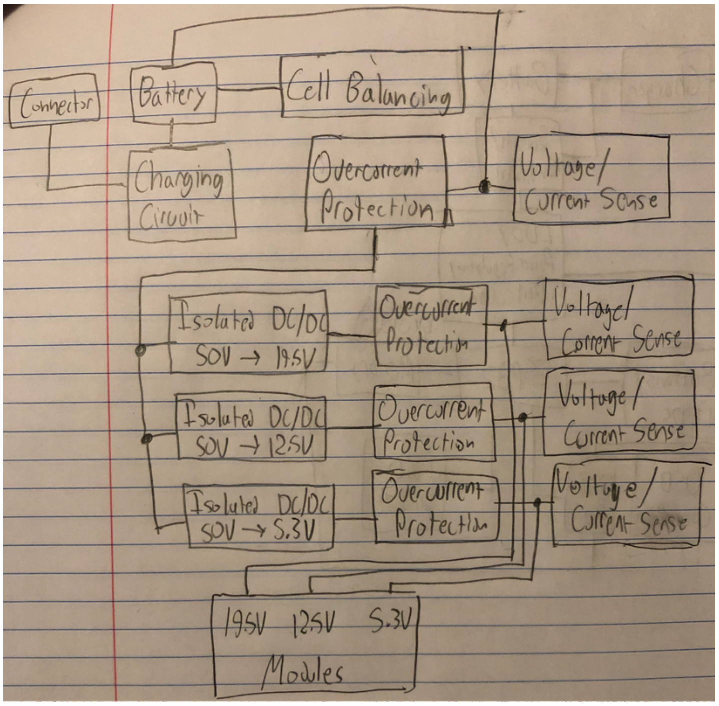


Figure 1. The system level design. Note that control circuitry is not featured here as this could not be developed.

General Design

The basis for the power management is based on two main components: regulation of voltage through conversion or cell balancing, with an overcurrent module to protect against current surges and limit current. Note the power management circuitry was not designed for use with a microcontroller, as no work could be done at a microcontroller. Every voltage level has an attached overcurrent module, and an attached voltage sense module; the overcurrent module provides current sensing. There are four voltage levels with their own four individual grounds (as a result of the DC/DC isolation). The four voltage levels are the battery pack voltage (50V typical), 19.5V, 12.5V, and 5.3V. The three voltage levels (besides battery pack voltage) are for the three voltage levels required for the module. The 19.5V level can supply 3.1A, 12.5V level can supply 0.3A, and the 5.3V level can supply 2.6A. This matches the specified power levels required for each voltage level, which are shown in Table 1.

Component	Voltage Level	Current Requirement	Power Requirement	Designed Voltage Level	Designed Maximum Current	Designed Power Level
Battery Pack	50V Typical	3.00A	150W	N/A	3.00A (typ.)	150W (typ.)
Isolated 19.5V	19.5V	3.10A	60W		3.10A (typ.)	60W (typ.)
Isolated 12.5V	12.5V	100mA	0.5W		300mA (typ.)	1.5W (typ.)
Isolated 5.3V	5.3V	2.58A (max)	13.1W (max)		2.60A (typ.)	13.8W (typ.)

Table 1. The required power levels and what the designed power levels are. Note that the isolated 12.5V voltage level has a slightly higher power level than required as this voltage level is the supply voltage for the op-amps/comparators in the 19.5V and 5.3V planes.

With the following overall design, this design is split into two separate boards: a battery management board, and a board for each isolated voltage levels (board for isolated DC/DC conversion). This is done so as the battery ground can remain separate from the isolated voltage grounds, and so that there are not too many separate boards for each isolated voltage level. In the isolated DC/DC board, each voltage level and its ground have their own layer (to ensure voltage and ground isolation). The overall circuit that is designed is shown in Figure 2. With the battery management design and isolated DC/DC designs shown. Note this overall circuit is designed for one wing of the pad. The connection of the wings is mentioned below.

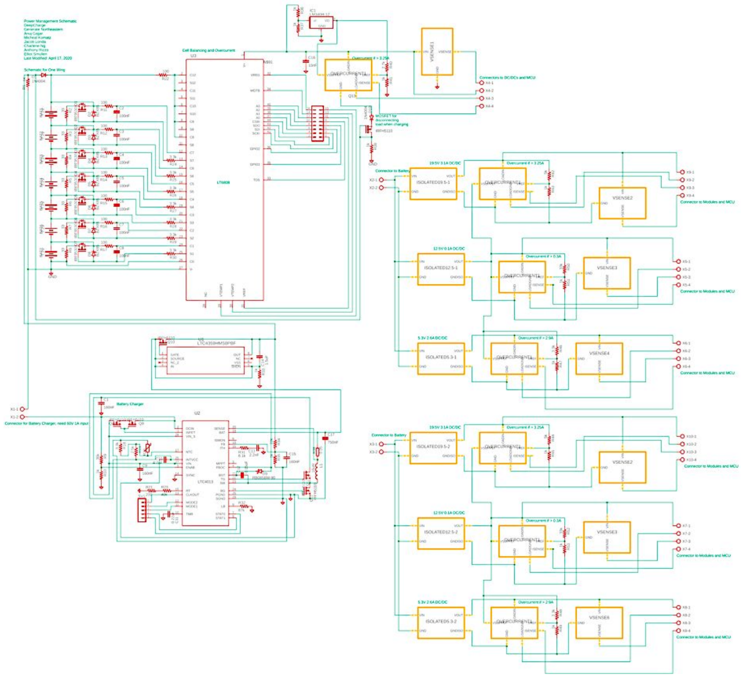


Figure 2. The overall circuit designed. This design is for one wing of the pad.

Battery Management Design

For the battery management system, there are three main components: a cell balancing system, overcurrent protection with voltage and current sensing, and a charging system. In the cell balancing system, the LTC6808-3/4 integrated circuit is utilized. This IC is utilized as it is robust, can handle the battery voltage, and can handle up to 12 cells, and can feed voltage and temperature data to a microcontroller via I2C/SPI (the IC is also specialized for cell balancing). Since the LTC6808 IC can only take a limited amount of current into its pins, the resistor MOSFET circuitry is meant to limit the current going into the pins, but still allow cell balancing to occur. This MOSFET circuitry is from the reference design for the IC [1]. An RC circuit is attached to the pins as well, as this limits any noise into the MOSFET (this is also from the reference design for the IC). The control/data pins in IC are attached to pins for now, as a control circuit has not been developed for the system. Note that the 12V LDO is for the voltage supply for the op-amps and comparators in the sensing and overcurrent protection circuit.

The charging circuit utilizes two ICs, with one IC for charging control, and another one for reverse voltage and current protection to the charging circuit and battery. The IC for charging control is the LTC4013 IC which specializes in high power charging of battery packs. This IC was chosen as it can provide the correct charging voltage and current for the battery pack, has accessible data and control pins for a microcontroller. Additionally, the IC also can provide various charging modes, with 2-Stage charging, 3-Stage charging, 4-Stage charging. One of these charging modes can be selected through a microcontroller.

The use of the IC was designed to supply a charging voltage and current of 54V typical and 1A typical (with the IC getting 60V and 1A input from the external connector of the pad), as this power level matches the charging specifications for the battery chosen [2]. The selection of the batteries will be described below. The placement of external resistors, inductors, diodes, and capacitors for the IC were based on the datasheet for the IC [3]; with the resistances, inductances, and capacitance values determined based on a calculator developed for the IC [4]. This calculator was also based on the datasheet for the LTC4013 IC [3]. This IC could not be simulated as no models were available. Additionally, there is a power MOSFET (shown in Figure 1) that disconnects the load (power to the modules) when the charging circuit is active, this simply works by disconnecting the modules/DC converters from the battery ground.

The integrated circuit for the reverse voltage and current protection is the LTC4359 IC. The utilization of this IC and its design is from the datasheet for the LTC4013 IC [3]. The overcurrent protection with voltage and current sensing modules will be described below.

DC/DC Conversion Design

For the power management system, there are two main components: isolated voltage converters, and overcurrent protection with voltage and current sensing.

In the isolated voltage converters, it consists of two main components: a transformer driver and a transformer. The transformer driver chosen is the LT8316 IC. We chose this IC as it was able to handle the input voltage and current levels from the battery (with each IC utilizing a maximum current of 1.5A RMS, or a maximum power of 75W RMS). The IC also conducts a switching regulation (control) of the transformer, thus providing voltage regulation at the output of the transformer. In other words, each converted voltage level should be regulated. The IC also utilizes its own internal current regulation (and shutoff), thus allowing further power management for voltage conversion. The transformers utilized in the design are fly-back transformers specifically designed for the LT8316 IC, and thus the transformers chosen are fly-back transformers that best match the specified power levels and conversions we need. These transformers are specified in the LT8316 datasheet [5].

For each voltage level, there are three separate isolated DC/DC designs. The placements of the transformer, diodes, capacitors, and resistors for each DC/DC converter are from the reference design in the LT8316 datasheet [5]. Similarly, the determination of inductance, capacitance, and resistance values are from a calculator developed for the LT8316 IC (this calculator is also based on its datasheet) [6]. Thus, with the reference design and the calculated values, the following designs shown in Figures 3 to 5 were made for each voltage conversion. The designs were also optimized for the required power levels (as described in Table 1).

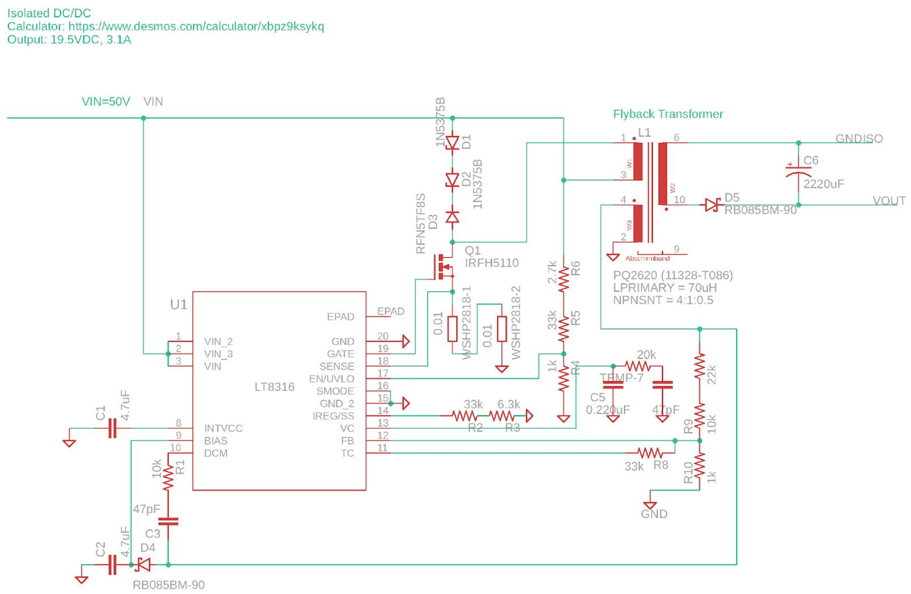


Figure 3. The 19.5V isolated DC/DC design.

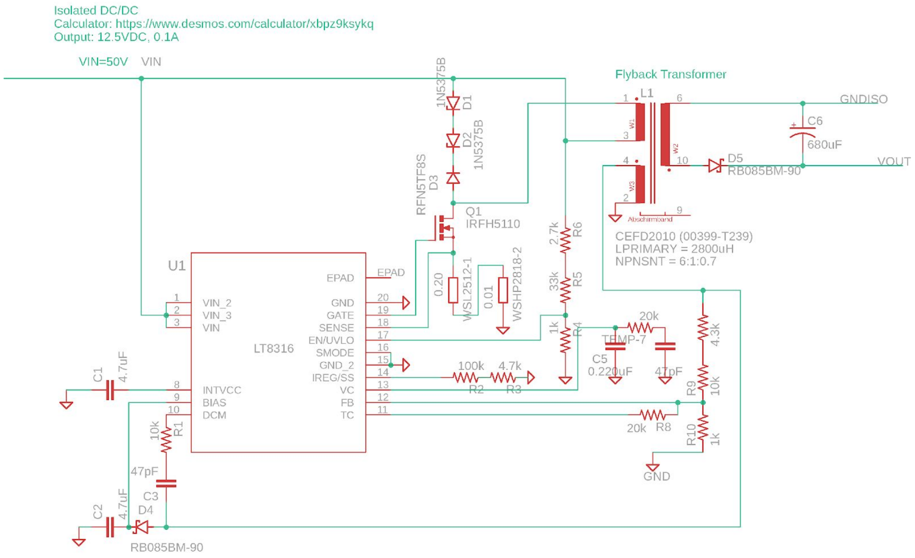


Figure 4. The 12.5V isolated DC/DC design.

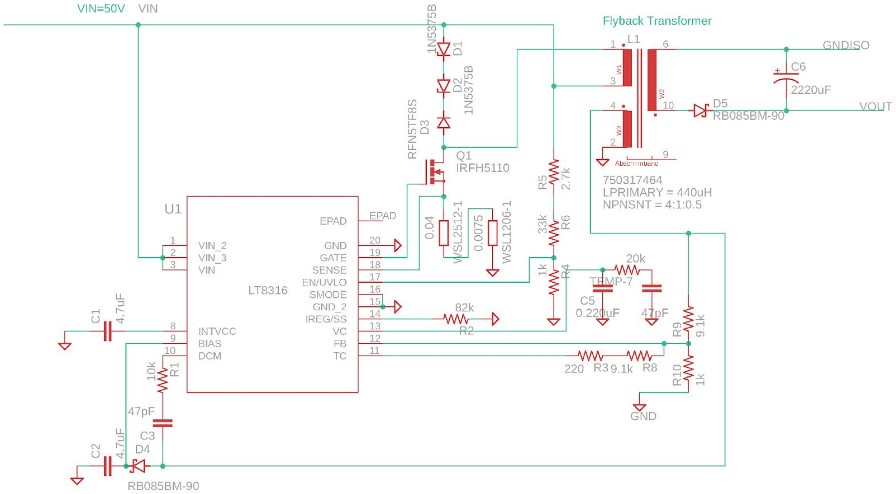


Figure 5. The 5.3V isolated DC/DC design.

Additionally, these designs were verified through simulation, as Analog provides simulation models for the LT8316. These simulations were conducted in LTSpice (Analog/Linear Technologies simulation software). A simulation was conducted for each voltage conversion, and a simulation was conducted with all the converters connected to a test battery voltage source (this simulation will not be shown here, but the data for this simulation are in the files shared for this project). Figures 6 to 8 show the results of the simulations for each voltage level, with a test load for each voltage level. Figures 9 to 11 also show the final RMS voltages.

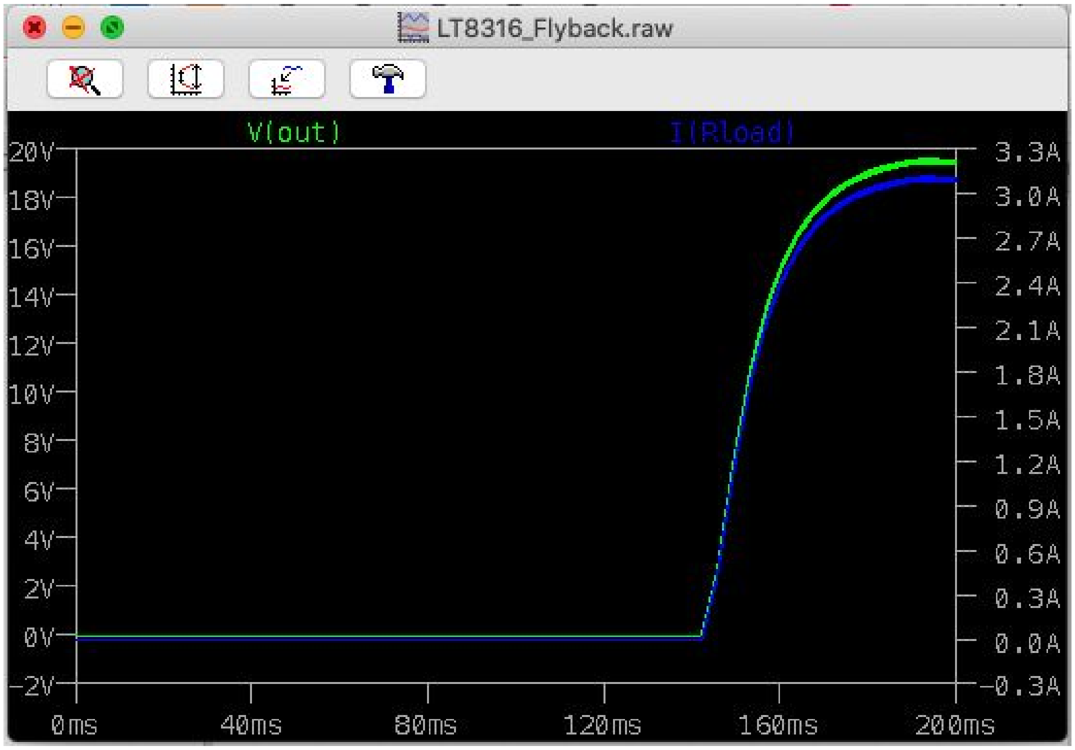


Figure 6. A 200ms transient simulation of the 19.5V isolated DC/DC converter design. The voltage at 200ms is the final RMS voltage which is shown in Figure 9.

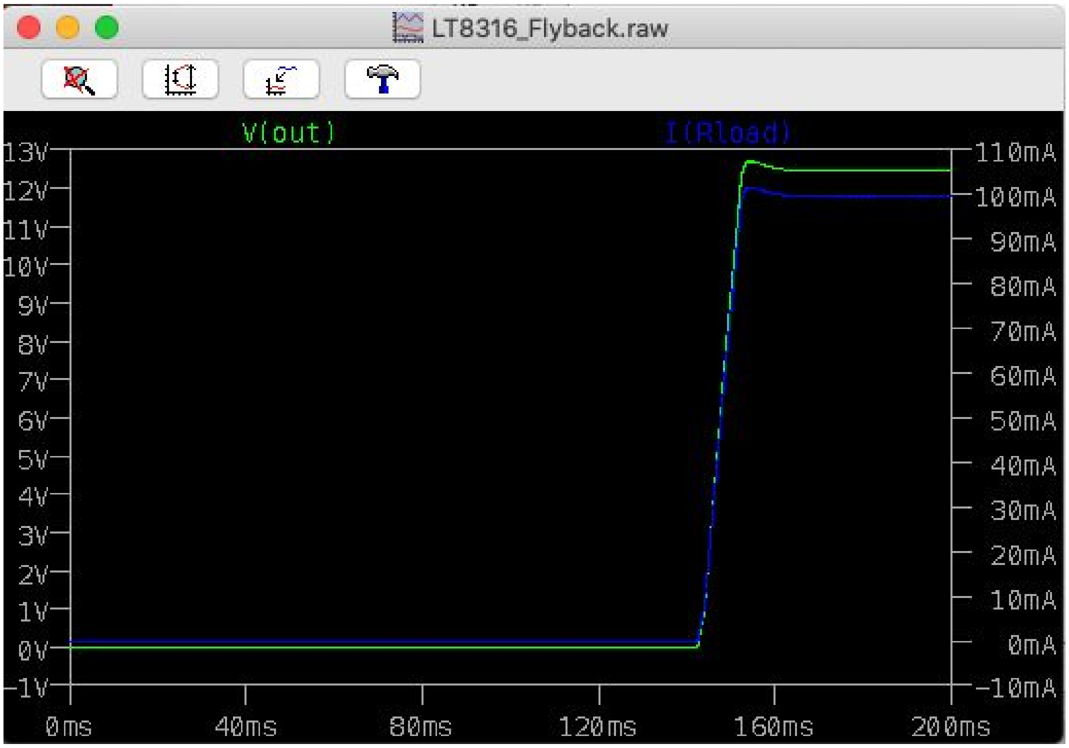


Figure 7. A 200ms transient simulation of the 12.5V isolated DC/DC converter design. The voltage at 200ms is the final RMS voltage which is shown in Figure 10.

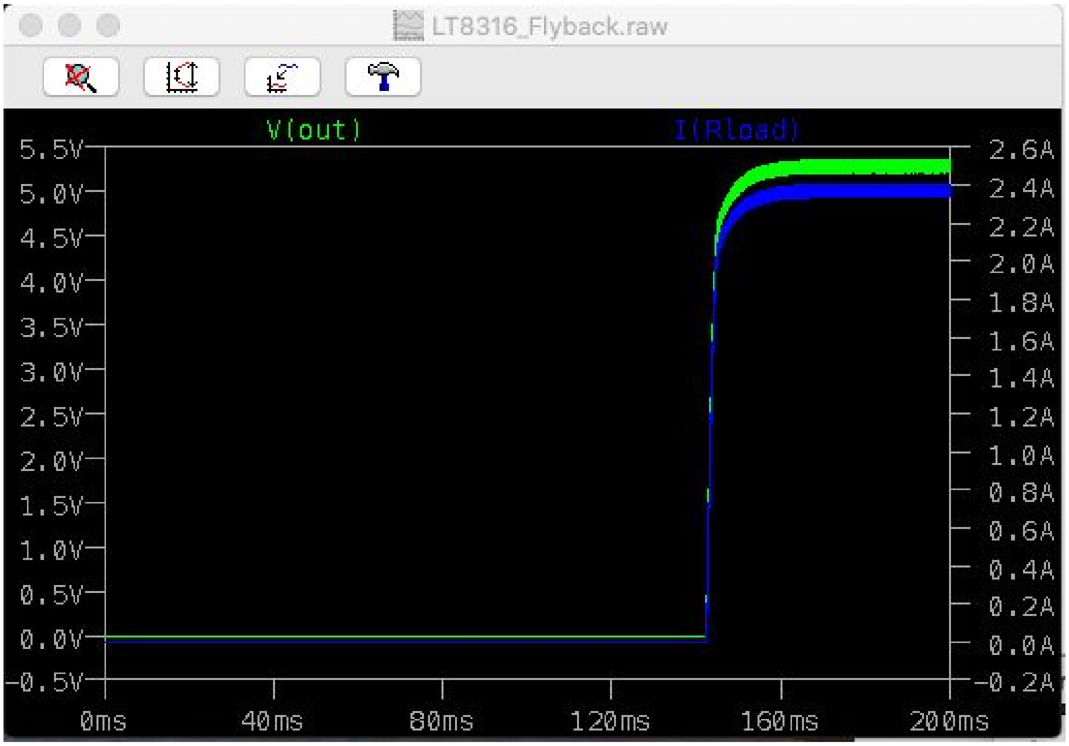


Figure 8. A 200ms transient simulation of the 5.3V isolated DC/DC converter design. The voltage at 200ms is the final RMS voltage which is shown in Figure 11.

V(out)

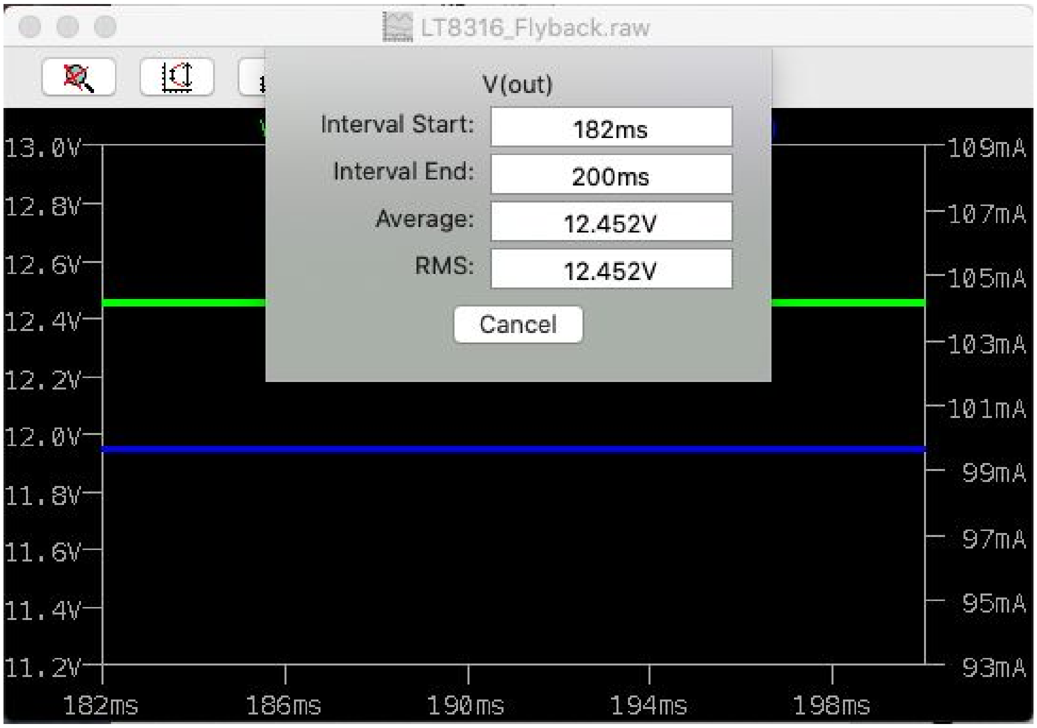
Interval Start: 192.5ms

Interval End: 200ms

Average: 19.469V

RMS: 19.469V

Figure 9. The final simulated RMS voltage for the 19.5V isolated DC/DC converter design. The green signal is voltage, and the blue signal is a test current.



V(out)

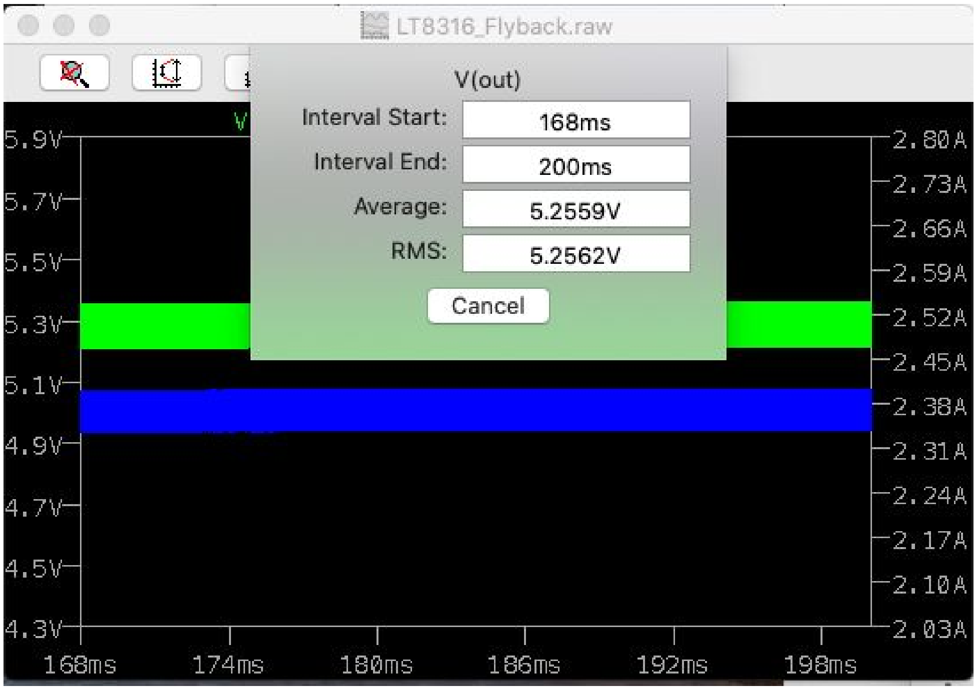
Interval Start: 182ms

Interval End: 200ms

Average: 12.452V

RMS: 12.452V

Figure 10. The final simulated RMS voltage for the 12.5V isolated DC/DC converter design. The green signal is voltage, and the blue signal is a test current.



V(out)
Interval Start: 168ms
Interval End: 200ms
Average: 5.2559V
RMS: 5.2562V

Figure 11. The final simulated RMS voltage for the 5.3V isolated DC/DC converter design. The green signal is voltage, and the blue signal is a test current.

All the above simulations show a generally correct design for each isolated voltage levels, with some error in the final RMS voltage. One note of concern is that there is a lot of high frequency noise in the output voltage. Figure 12 shows the Fourier transform of the 19.5V output, which shows a spike in intensity at a frequency of 1GHz.

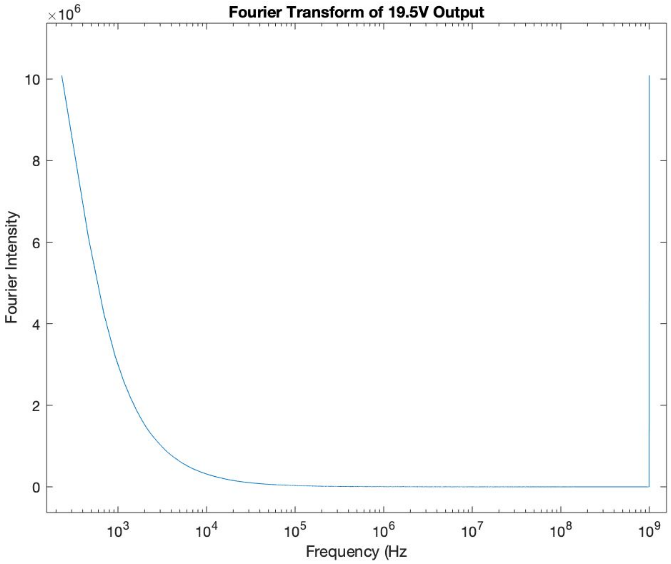


Figure 12. The Fourier transform of the simulated 19.5V output.

One way to mitigate the high frequency noise is the addition of a capacitor with a relatively large capacitance, and this was added to the circuit (with a minimum capacitance determined in the calculator used for the LT8316 IC). This capacitor was chosen to be as high as possible, however if the capacitance is too high, the output voltage can attenuate.

Nonetheless, based on simulation (with test loads), the isolated DC/DC circuits should function as expected.

Overcurrent Protection/Power Management

A general overcurrent protection circuit was utilized for each voltage/power levels. This overcurrent circuit has been designed to protect circuitry against current/power surges, but also can actively limit the current to a specified reference level (in other words, when a current surge occurs, the current will be limited, without the circuit being turned 'off'). A custom design (based on reference designs utilizing a shunt resistor with a gate controlled MOSFET) was made for the overcurrent protection circuit. This design is shown in Figure 13.

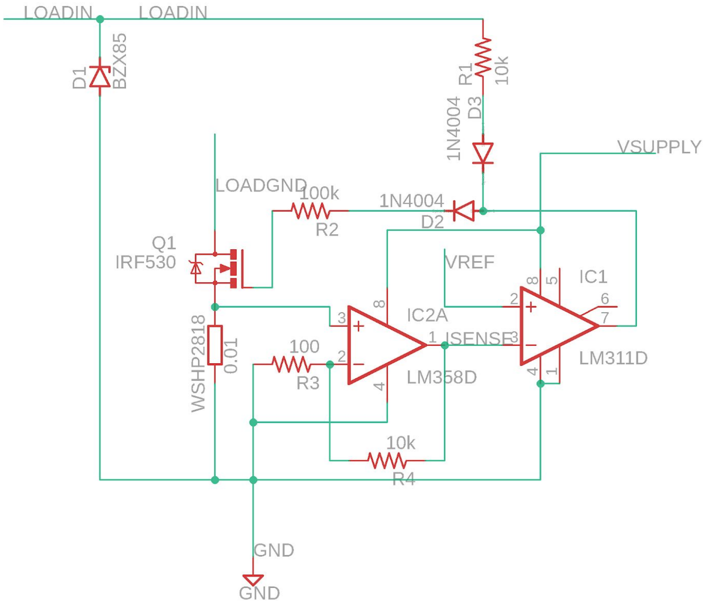


Figure 13. The overcurrent protection circuit design.

The way the circuit work is that any attached load is placed between the input node (positive node) of the load, and the ground (or negative) node of the load. This load is then connected to a power MOSFET, that has a shunt resistor (the voltage detected across the shunt resistor is proportional to measured current) connecting the source to ground. The voltage across shunt resistor is then inputted into an op-amp that amplifies the input voltage value (multiplication by 100) to the input current value. This input current value is then inputted into a comparator that outputs the comparator input voltage (this input voltage is roughly equal to the voltage across the load) if it is less than the inputted reference voltage (this inputted reference voltage is the maximum allowable current for the attached load). In other words, the MOSFET generally operates in saturation when the input current is less than the maximum allowable current for the load. Otherwise if the input voltage (for the comparator) is above the inputted reference voltage, then an output equal to the threshold voltage for the selected MOSFET (which in this case is 3V). In other words, the MOSFET operates in its triode/linear region when the input current is above than the maximum allowable current for the load, but the MOSFET will not operate in cutoff. The purpose of the 100k resistors are to limit the input impedance to the gate of the MOSFET, and also attaches a pull-up resistor for the voltage to the gate of the MOSFET. The diodes prevent any reverse voltages/current going back into the comparator/op-amp circuitry. Additionally, the Zener diode provides some minor voltage regulation for the circuit (this Zener diode improved the circuit performance in simulation).

To verify this circuit, simulations were conducted in OrCAD PSPICE (as this software had available models for the op-amp and comparator utilized). In this simulation a test load of 10 Ω was utilized, with an maximum allowable current of 1A set (reference voltage of 1V to the comparator). Two simulations were made: one where a current source was placed in series with the test load, and one where a voltage source was placed across the load. Figures 14 and 15 show the results of these simulations.

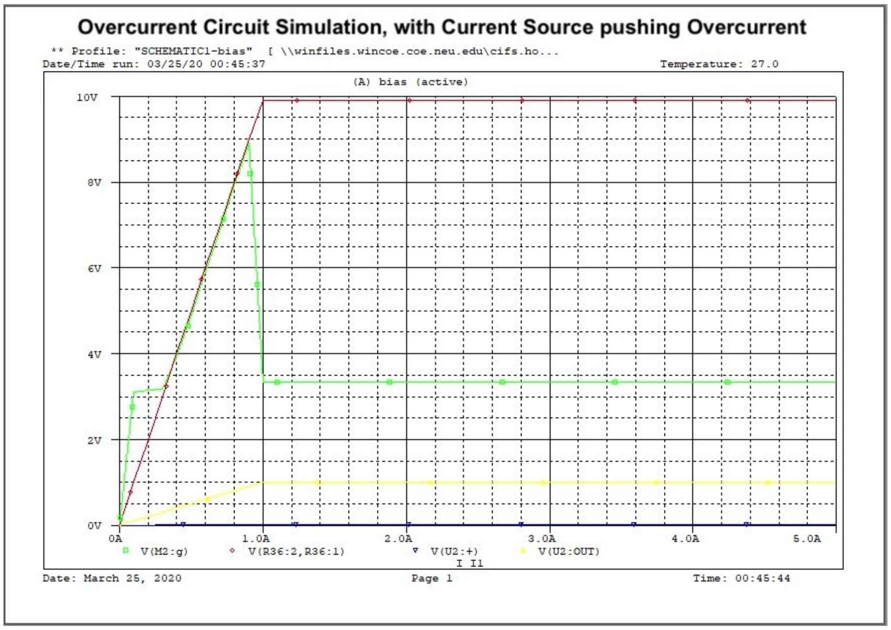


Figure 14. A simulation of the overcurrent circuit when a current source was placed in series with a test load of 10 Ω, and a maximum allowable current of 1A (reference voltage of 1V to the comparator. The red signal is the voltage across the test load, the green signal is the voltage into the gate of the MOSFET, and the yellow signal is the measured current from shunt resistor and op-amp. The x-axis in the input current into the test load.

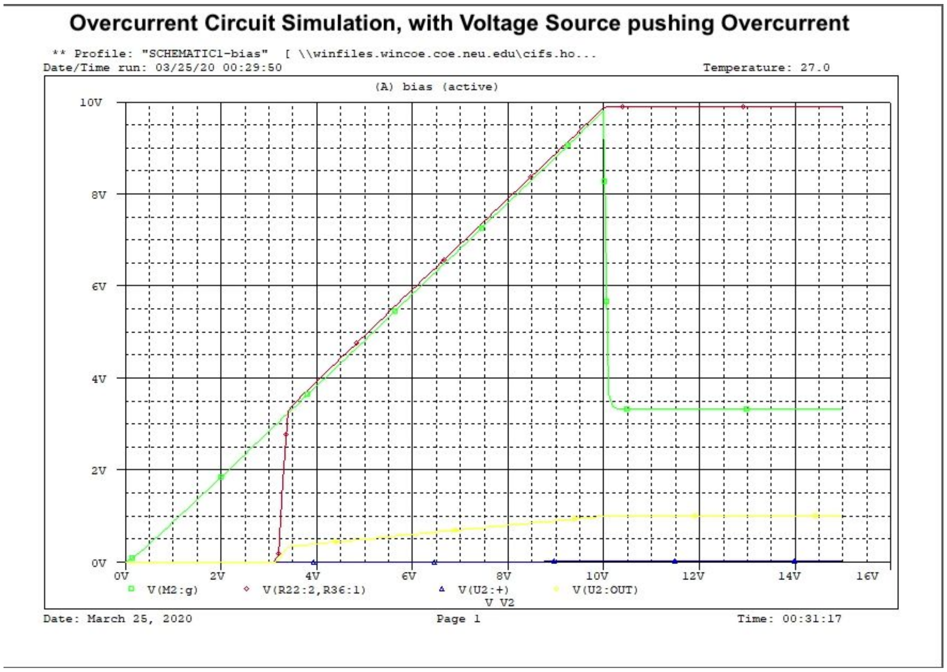


Figure 15. simulation of the overcurrent circuit when a voltage source was placed across a test load of 10 Ω, and a maximum allowable current of 1A (reference voltage of 1V to the comparator. The red signal is the voltage across the test load, the green signal is the voltage into the gate of the MOSFET, and the yellow signal is the measured current from shunt resistor and op-amp. The x-axis is the input voltage into the test load.

As seen by the simulations, there appears to be the correct operation of the overcurrent circuit, in which when the measured current was above (or approaching to be above) the maximum allowable current for the load (reference voltage into the comparator) the current was mostly limited below that value (with the voltage being limited to 10V and current being limited to 1A for both simulations. Hence from the simulations, the overcurrent limiting circuits should function as expected.

Additionally, voltage sensing and current sensing is attached to the power management circuitry; with current sensing being implemented in the overcurrent protection circuit. The voltage sensing circuitry is simple. Simply the input voltage node for a load is placed in parallel to the voltage sensing circuitry that feeds the supply voltage and input voltage for an op-amp, that divides the input voltage by approximately ½ (to prevent the input voltage from being too high for an analog-digital converter (ADC) reading the input voltage. These sensed voltage and current values would then be fed into a microcontroller with an ADC attached. No circuitry was designed to allow the microcontroller to use the sensed voltage and current values and control the power management circuitry. The voltage sensing circuit is shown in Figure 16.

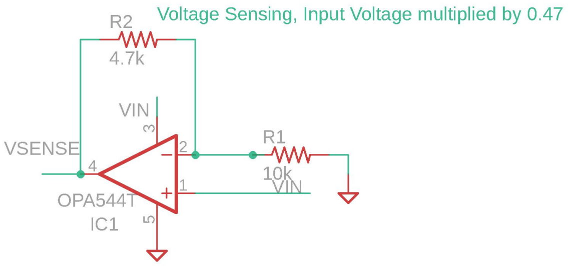


Figure 16. The voltage sensing circuitry model

Power Requirements/Overcurrent Conditions

With the overall design, the power requirements are summarized for the entire system below. This is also based on the battery and part selection, which is outlined below in Table 2.

Component	Voltage Level	Current Level	Power Level
Battery Pack	50.8V (typ.) 58V (max.)	3.00A (typ.) 3.10A (max.)	152W (typ.) 180W (max.)
Charging Circuit	60V (input) 54V (output)	1.00A (input) 1.00A (output)	60W (input) 54W (input)
Isolated 19.5V	19.5V (RMS)	3.10A (typ.) 3.25A (max.)	60W (typ.) 64W (max.)
Isolated 12.5V	12.5V (RMS)	100mA (typ.) 300mA (max.)	0.5W (typ.) 1.5W (max.)
Isolated 5.3V	5.3V (RMS)	2.60A (typ.) 2.80 A (max.)	14W (typ.) 15W (max.)

Table 2. A table summarizing the power levels for each component in the overall design.

With the overcurrent circuits for each voltage level, we set certain overcurrent conditions for the voltage levels; which are slightly higher than the designed voltage. This is to allow the power levels to reach the power demand from the module (in other words, handle transient changes in power levels when the system turns on). The overcurrent conditions that we selected are shown in Table 3. Note that the battery overcurrent condition is a relatively strict limit as this prevents the battery pack from being drained too quickly.

Component	Voltage Level	Current Requirement	Overcurrent Condition
Battery Pack	50V Typical	3.00A	3.10A
Isolated 19.5V	19.5V	3.10A	3.25A
Isolated 12.5V	12.5V	100mA	300mA
Isolated 5.3V	5.3V	2.60A	2.80A

Table 3. The overcurrent conditions set for each power level.

Battery and Parts Selection

With the general power requirement of 60W for each module (total power requirement of 120W for one wing), we then chose batteries that optimized for this power requirement, but also limiting the weight. The primary factor for choosing the batteries was the total watt-hour capacity that allows for two charges; which is 480Wh. The pack we chose Is the [2]. This pack provides 34.6Wh per battery, requiring a total of 14 packs (7 packs per wing), and a total weight of 2.67kg. This battery is ideal in comparison to the relatively light MacBook battery, that provides 55Wh of energy, requiring 9 packs in total, it results in a total weight of 4.08kg.

The batteries we chose provide enough energy for two recharges of a drone battery and limits the weight of the pad. These batteries also have attached thermistors that can be used to monitor the temperature of the batteries and allows us to ensure that the pad is not dissipating too much heat. Additionally, these batteries have their own internal overcharge, overcurrent, and overvoltage circuitry, which provides another level of power management to the system and provides critical safety circuitry to prevent any potential battery fires or battery mismanagement.

With parts selection, one of the main considerations was the power levels at each part in the system. In other words, we chose parts that were able to handle the voltages and currents in the systems. The ICs we chose are all able to handle power more than the maximum power expected in the system of 75W. Power MOSFETS were generally used, as these MOSFETS have limited thermal and power dissipation when utilized in high power electrical systems.

With regards to the op-amps and comparators (and the single LDO in the battery management plane) used in the design, these op-amps generally can generally up to 30V in voltage supplies and inputs, but these op-amps and comparators are used in a way in which they will not encounter any voltages higher than 30V, and any currents above 100mA.

Finally, the resistors, capacitors, and diodes that were selected, were selected to first have the 'worst-case' footprints for PCB design (in other words, take the maximum size on the PCB, so that the PCB can be designed in a way which there will be additional space for any additional circuit components in the PCB). For the surface mount resistors on the PCB, the SMD2512 (6332 metric footprint) footprint was utilized, as these are 1W resistors that can handle any potential (relatively) high power dissipation for any resistor in the system. Similarly, the SMD1812 (4532 metric footprint) was utilized, as these should also be able to handle the relatively high voltages and currents in the system. Finally, with the diodes selected, some diodes were selected to be through hole, as these diodes are able to handle the high power in the system. A surface mount diode was utilized if that diode could meet the power requirements of the system.

Additionally, one note is that the connectors selected are temporary, as these connectors also provide the 'worst-case' footprint for part selection (these connectors are large screw-in connectors). Another note is that the transformer footprint selected was also another 'worst-case' one, this was a fly-back transformer footprint available in the default Eagle library and provides a large footprint that covers the transformers selected. This was done so, there was no reason to create custom footprints for the selected transformers, due to not being able to physically test the system. Nonetheless, these connectors and transformers should be easy to replace in the final PCB design. A spreadsheet of all the selected parts are attached to the shared documents.

Integration with Control Circuitry

One component that has not been developed for the power management system is any control circuitry or microcontroller integration. This was a part of the system that could not be developed (due to COVID complications). However, within the mechanical design of the pad, there should be room to add control circuitry. One of the main developments is the integration of the control circuitry with the microcontroller, which does require two design considerations: allowing the microcontroller to control power management, and sending current, temperature, and voltage sensing to the microcontroller. No control circuitry for a microcontroller was developed, however if this control was to be implemented, this control can be used for gate driving power MOSFETs in the power management circuitry. The schematic and PCB design should be easy to modify in this case. However (as also outlined above) there is current, temperature, and voltage sensing throughout the power management circuitry. Table 4 below shows what current, temperature, and voltage sensing connections there are. Each current, temperature, and voltage sensing nodes are connected to a pin or connector (with exception to the battery thermistors).

Location	Part	Sensing Mode	Node amount	Description
Battery Management	Voltage Sensing	Voltage	1	Sensed voltage from voltage sensing module
Battery Management	Overcurrent Protection	Current	1	Sensed current from overcurrent module
Battery Management	Battery Pack	Temperature	7	Thermistors within each cell, these do not have attached connectors
Battery Management	Charging Circuitry	Temperature	1	Thermistor utilized for the LTC4013 IC
Battery Management	Charging Circuitry	Voltage	1	Sensed voltage for the LTC4013 IC
Battery Management	Charging Circuitry	Current	1	Sensed voltage for the LTC4013 IC
Isolated DC/DC board	19.5V DC/DC, 12.5V DC/DC, 5.3V DC/DC	Voltage	3	Sensed voltage from voltage sensing modules
Isolated DC/DC board	19.5V DC/DC, 12.5V DC/DC, 5.3V DC/DC	Current	3	Sensed current from overcurrent modules
Extraneous thermistors	Not implemented	Temperature	Unknown	Thermistors placed around the pad to monitor temperature (not implemented yet)

Table 4. The current, voltage, and temperature sensing nodes implemented in the circuitry.

Remaining Design Considerations

Throughout this document, certain design considerations that haven't been made have been outlined. Here, the remaining design considerations are outlined. Firstly, the connectors throughout the system have not been determined. This includes the external connector for the pad itself, and the internal connectors for the system. Decisions on what connectors to use haven't been made as it was difficult to make such considerations. Nonetheless, the connectors that are not accounted for is the external connector for the pad, the connectors to all seven of the individual batteries (within a wing), the pins and connectors for each voltage, current, and temperature sensing node, the pins and connectors for accessing the control inputs/outputs of the cell balancer and charging integrated circuits (LTC6808-3/4 and LTC4013 ICs), the connectors between the boards, and the connectors/wires for connecting the sensing, control inputs/outputs, and charging power wires between the two wings. Table 5 shows all the connectors that have not been considered for one wing, and Table 6 shows all the minimum wire gauges for all the required connections [7].

Another design consideration that hasn't been made is user feedback from the power management system. This was not implemented as no control system was created for the pad. The idea with this, is that the microcontroller implemented will use the sensed voltage, current, and temperature ideas, and send inputs to any user feedback systems implemented. This will prevent any major changes being required for the power management schematic and boards designed.

Additionally, a major design consideration that hasn't been made is the power input/connection to the charging circuit. At the moment, the power input/connection is designed to be a 60V 2A DC input from the external connector of the pad, in which 60V 1A is for each battery management board (parallel connection to external connector for both battery management boards).

In summary, the design considerations that haven't been made are the connectors within the pad, the connection between one wing and the other, any control circuitry/systems, the specific footprints of transformers and connectors implemented, and any user feedback systems.

Board/Location	Connector Purpose(s)	Number of Connectors Needed	Notes
External connector for both wings	DC Power for Battery Charging	1 for Power 1 for Ground (2 Total)	Need 60V 2A input
Connectors between both wings	Sharing DC Power for Battery Charging Control Board Power Control Input/Output	1 for Power 1 for Ground 1 for Control Power 1 for Contorl Ground 37 for Control Input/Output (41 Total)	Power for control board Control input is any sensed values, and control output is any control for an iC

Battery pack	DC Power Output to Battery Management Board Thermistors	7 for Power (Output) 7 for Ground (Output) 7 for Thermistors (21 Total)	
Battery management board	DC Power for Battery Charging Battery Pack Input Control Input/Output Power Output to DC/DC Board	1 for Power (Input) 1 for Ground (Input) 7 for Power 7 for Ground 31 for Control Input/Output 1 for Power (Output) 1 for Ground (Output) (49 total)	Need 60V 1A for input power/GND
Isolated DC/DC board	Battery Power Input 19.5V DC Output 12.5V DC Output 5.3V DC Output Control Output	1 for Power (Input) 1 for Ground (Input) 1 for Power (Output) 1 for Ground (Output) 1 for Power (Output) 1 for Ground (Output) 1 for Power (Output) 1 for Ground (Output) 6 for Control Input/Output (14 total)	Control output is the sensed currents and voltages
Control Board (Not Built) (Connectors for connections from one wing)	Power for Control Board Receiving control inputs, and sending control outputs	1 for Power (Input) 1 for Ground (Input) 37 for Control Input/Output (39 total)	This is still to be determined

Table 5. All the required connectors needed for the pad (connectors for one wing are shown).

Board/Location	Connector Purpose(s)	Number of Connectors Needed	Minimum Wire Guage
External connector for both wings	DC Power for Battery Charging	1 for Power 1 for Ground (2 Total)	16AWG 16AWG
Connectors between both wings	Sharing DC Power for Battery Charging Control Board Power Control Input/Output	1 for Power 1 for Ground 1 for Control Power 1 for Contorl Ground 37 for Control Input/Output (41 Total)	16AWG 16AWG 16AWG 16AWG 24AWG

Battery pack	DC Power Output to Battery Management Board Thermistors	7 for Power (Output) 7 for Ground (Output) 7 for Thermistors (21 Total)	16AWG 16AWG 24AWG
Battery management board	DC Power for Battery Charging Battery Pack Input Control Input/Output Power Output to DC/DC Board	1 for Power (Input) 1 for Ground (Input) 7 for Power 7 for Ground 31 for Control Input/Output 1 for Power (Output) 1 for Ground (Output) (49 total)	16AWG 16AWG 16AWG 16AWG 24AWG 16AWG 16AWG
Isolated DC/DC board	Battery Power Input 19.5V DC Output 12.5V DC Output 5.3V DC Output Control Output	1 for Power (Input) 1 for Ground (Input) 1 for Power (Output) 1 for Ground (Output) 1 for Power (Output) 1 for Ground (Output) 1 for Power (Output) 1 for Ground (Output) 6 for Control Input/Output (14 total)	16AWG 16AWG 16AWG 16AWG 16AWG 16AWG 16AWG 16AWG 24AWG
Control Board (Not Built) (Connectors for connections from one wing)	Power for Control Board Receiving control inputs, and sending control outputs	1 for Power (Input) 1 for Ground (Input) 37 for Control Input/Output (39 total)	16AWG 16AWG 24AWG

Table 6. The required minimum wire gauges for each connector type. The chosen wire gauges are based on this chart [7].

PCB Design

Finally, even if not all the design considerations have been made, PCB board designs were constructed for the battery management board and isolated DC/DC board. Note that the isolated DC/DC board combines all isolated DC/DC boards for each board.(with each isolated DC/DC level getting a dedicated power and ground planes).

The PCB designed for battery management, is a two-layer PCB with a top layer for power and signals, and a bottom layer for ground. This was done so as there are not many nodes connecting to the combined battery output power (besides a connector for the isolated DC/DC board), and the power traces and signal traces were able to be separate. In this battery management PCB the following trace widths were utilized: any high current areas and the power/ground to the DC/DC board had trace widths of 2mm, battery connections also had a trace width of 2mm., and any signal connections had trace widths between 0.3-0.5mm (as there is relatively low current for the signal connections). These trace widths were selected based on two calculators for determining minimum needed trace width for certain voltage/current requirements [8] [9]. Some components had traces going under them (components which did not have any keep-out violations). While we did try to avoid doing so, some traces needed to be placed under components.

The isolated DC/DC board is an 8-layer PCB with a top layer for signals (and battery power input), the second and third layer for 19.5V power and ground, the fourth and fifth layer for 12.5V power and ground, the sixth and seventh layer for 5.3V power and ground, and the eight (and final) layer for the battery ground. This was done so as the power levels and grounds are isolated from each other on the board. Vias were placed from the top layer to each necessary power and ground level. For this PCB the following trace widths were utilized: 2mm for any high current connections, 0.5mm for overcurrent signals, 0.75mm for the isolated DC/DC IC signals, and 0.75mm for the battery power and ground. These trace widths were selected based on two calculators for determining minimum needed trace width for certain voltage/current requirements [8] [9]. As with the battery board some components did have traces going under them components which did not have any keep-out violations). While we did try to avoid doing so, some traces needed to be placed under components.

It is important to note that both boards were designed with overall footprints that matched the mechanical footprints in the mechanical design, and were designed with the idea that other components may be added to the PCBs and the pad (hence the compactness in the PCBs and the unused space in the PCBs). Additionally, as mentioned above, some footprints were ‘mock’ or ‘worst-case’ footprints for sizing, especially the temporary screw-in connectors/2.54mm pins placed on the PCB, and the temporary footprints for the transformers.

Nonetheless, the PCB designs should provide a strong base for the final pad design.

Software Design

The goal of the software portion of the FlyPad is to monitor the status of the internal electrical systems, and communicate this status to the drone. Due to complications of the pandemic, a majority of the software design could not be completed. However, details of this system are outlined below, so that future development will have an understanding of how it plays into the overall design.

A key role of the software is to monitor the status of the electrical subsystems within the FlyPad, which includes current, voltage, and heat measurements. Sensors, such as thermistors and analog to digital converters, would be placed throughout the pad at critical points. Readings from these sensors would be polled on a regular basis, and actions would be taken according to these measurements if needed, such as shutting down a subsystem if it gets too hot. These readings would also be arranged into a message format to be sent to the nearby UAV via bluetooth low energy communication. The “backpack” of the UAV would have similar software running, which receives the message from the FlyPad and acts accordingly. The UAV backpack and FlyPad would continue transferring status messages back and forth throughout the landing process, which forms a handshaking protocol. This protocol ensures that both the UAV and FlyPad are in nominal condition, and ready to start charging. During charging, the two systems would continue to communicate about the amount of power being transferred to each leg from the coil modules, which allows the backpack to perform an efficient power summing operation.

The microcontroller Generate picked for this design was the STM32WB55 [10]. The company STM was chosen because the client was already using STM microcontrollers for their coil network controllers, so it allowed for a common environment when developing these two systems. The 32WB series have a certified Bluetooth 5.0 radio stack, up to 64 Hz clock, and access to most standard peripherals such as an ADC and SPI ports. The P-NUCLEO-WB55 [11] eval board was ordered to start development, but could not continue because of the previously mentioned complication.

Next Steps

Overall, this project is still in the early stages of the product development cycle. A significant amount of design work has been completed by Generate, but due to the pandemic, a majority of this work has not been thoroughly tested. Once in-person work can be resumed, it is highly recommended that everything be prototyped and iterated on first, before moving on to next steps. In terms of the electrical design, this means breadboarding all subsystems with real components. For the mechanical design, an initial functioning prototype should be created so that all of the structural design choices can be tested, such as the hinge and support ribbing. Testing these systems before using them in a later design is crucial to finding any shortcomings or mistakes, and allows user feedback as well.

As mentioned, there are some electrical design considerations that have not been made. The connectors throughout the system have not been determined (internal and external connectors). Another design consideration was any integration or control points for a microcontroller (for example, letting the microcontroller determine when to turn “on” certain electrical subsystems). Additionally, no user feedback was implemented (for example, no indications of how much charge the batteries have, or any indications of temperature in the system). These design considerations are also further described above, where the electrical design is described.

It is critical to make significant progress in the electrical design of the FlyPad in order to continue mechanical development. Much of the mechanical design relies on the specific shape, weight, required clearances etc. of the internal components. Details such as these must be narrowed down before more work can be done on production-ready mechanical design. Additionally, thermal management should be considered once estimates can be made about heat dissipation from the electronic components. As for progress the mechanical team can make now, the hinges and structural features can be tested and further developed at this point. There should also be conversation between the mechanical and electrical teams regarding board sizes and shapes. Arranging the internal components will require compromises from both sides.

References

[1]. LTC6808-3/4 Datasheet
https://www.analog.com/media/en/technical-documentation/data-sheets/680324fa.pdf
[2]. UBBL24 Battery Datasheet:
https://www.mouser.com/datasheet/2/419/DS_UBBL24-1122006.pdf
[3]. LTC4013 Datasheet:
https://www.analog.com/media/en/technical-documentation/data-sheets/4013fa.pdf
[4]. LTC4013 Calculator: https://www.desmos.com/calculator/ra94ggimfg
[5]. LT8316 Datasheet:
https://www.analog.com/media/en/technical-documentation/data-sheets/LT8316.pdf
[6]. LT8316 Calculator: https://www.desmos.com/calculator/ehqz6zopbi
[7]. Wire gauge chart: https://www.powerstream.com/Wire_Size.htm
[8]. PCB trace width calculator: http://circuitcalculator.com/wordpress/?p=25/
[9]. PCB trace width calculator: https://www.7pcb.com/trace-width-calculator.php
[10]. STM32WB series microcontroller:
https://www.st.com/en/microcontrollers-microprocessors/stm32wb-series.html
[11]. P-NUCLEO-WB55 evaluation board:
https://www.st.com/content/st_com/en/products/evaluation-tools/product-evaluation-tools/stm32-nucleo-expansion-boards/p-nucleo-wb55.html