Spring 6-9-2023

Hybrid Photovoltaic + Battery Energy System Grid-tied Converter Capstone Review Thesis

Archer Taylor
Portland State University

Recommended Citation

Taylor, Archer, "Hybrid Photovoltaic + Battery Energy System Grid-tied Converter Capstone Review Thesis" (2023). *University Honors Theses.* Paper 1371.
https://doi.org/10.15760/honors.1400

Hybrid Photovoltaic + Battery Energy System Grid-tied Converter Capstone
Review Thesis

By Archer Taylor

Background

As costs of photovoltaic (PV) and battery energy system (BES) technologies fall, interest in renewable energy infrastructure increases. Hybrid power converters that incorporate both technologies have added benefits of reduced losses due to the devices being collocated as well as a centralized control scheme. Currently the power architecture for these hybrid systems is uncertain. My senior capstone group, which consists of myself and three other peers similarly graduating from the Electrical Engineering program at Portland State University, was tasked with developing a novel hybrid PV+BES topology with a corresponding control system. Our proposed architecture provides a possible solution to future challenges as these hybrid power plants become the norm.

Many renewable energy sources, such as wind and solar, are intermittent and require unique specialized facilities to maximize their production. (Gupta) Since the intensity and availability of these energy sources can vary day to day and even hour to hour, the systems that capture that power are often designed to underestimate the expected output. Therefore, PV arrays can often outproduce their rated output, referred to as having an inverter loading ratio (ILR) above 1. This clipped energy is normally lost to the system, providing the opportunity to introduce a hybrid PV+BES plant that can store clipped energy in a coupled battery system. Additionally, both systems being collocated allow for higher efficiency and lower losses since the energy has to travel through less power converters. For example, when weather conditions permit and a PV array is producing more energy than the grid is demanding, power flow from the PV array to the BES only has to move through at minimum 2 converters. If they weren’t collocated, power flow would still need to move through the grid in order for excess energy to be stored in an offsite storage system, likely involving at least 4 converters. Hybrid PV+BES power plants present an exciting opportunity as society moves away from fossil fuel based power plants.

Our project group was assigned this task from within the Electrical Engineering department at PSU, overseen by Dr. Mahima Gupta, with the goal of designing a topology and control system while demonstrating the feasibility of hybrid converters. The capstone project involves the design and implementation of multiple different power converters. The basic DC-to-DC converters are boost, buck, and buck-boost converters. Boost, as the name suggests, has an output voltage larger than that of the input. Buck works the opposite with a smaller output voltage than the input. And Buck-boost converters can both have a larger or smaller output voltage than the input. These circuit topologies were our starting point when designing the power converters connected to both the PV array and the BES, but connecting to the grid required an AC-to-DC conversion since the grid is supplied via 3 phase AC power. These were all considerations our team had to balance during the Research portion of the project. Each of us had a small amount of familiarity with these topics as they were touched on in our required courses, but the undertaking of the complete unsupervised design process was daunting.

Research

Prior to starting the project, my group and I had little specific knowledge regarding power converter design and the associated theory. We each had taken a course that had us building and tuning a closed-loop buck converter, but unfortunately that project didn't empower us to make many decisions and we lacked much of the design experience needed for this project. As such, we spent multiple months gathering background research to start building a collective knowledge base to work from. Our initial plan was to start with a research focused period and then move onto a design period. However, the research never really concluded as we moved into design and testing. Our understanding of what was necessary to complete the project continued to expand throughout the entire process.

We spent a majority of our beginning months gathering as much relevant information as we could, attempting to build our foundation from which to work from for the rest of the project. This included investigating different converter topologies until we both understood our specific needs as well as determined which converters would be necessary to satisfy our needs. Since the DC-DC converter connected to the PV array only needed to have unidirectional power flow, this could be simply enabled with a Boost converter and was decided on early. Additionally, for the DC-AC conversion we settled very early on Sine PWM inversion since that method came up often in our early research and was simplest. A couple topologies that were ultimately unused for our bidirectional converter design connected to the BES include a SEPIC converter and bi-directional boost-buck converter. The SEPIC converter boasts reduced output voltage ripple and low voltage stress on switches. (Paul et al.) This converter design had specific benefits when used for BES in renewable energy applications, but we ultimately moved on from it since the research we found was only verified in simulation and not experimentally. The bidirectional boost-buck converter showed promise in similar applications using battery arrays, but required multiple cascaded converters that we felt would over-complicate our simulations and experimental tests. (Busquets-Monge et al.) There were many more research paths that we went down in these early stages and they all contributed to our understanding of hybrid converter design.

Most of the research mentioned previously was performed by my partners, while I focused on the control side. As is common with designing circuits with control systems, the first step was to learn how to design the open-loop converters topologies. Open-loop refers to a non-feedback control system, while a closed loop uses the output and "loops" it back into a control system and compares it to a reference so that error can be minimized. The open-loop design was straightforward due to the availability of resources between video tutorials and textbook references. Fundamentally, power converters use switching devices that switch at high frequencies with a specific duty ratio that determines the characteristics of the power conversion. Duty ratio refers to the pulse width in a signal, specifically the ratio of time the switch is considered “ON” to the amount of time the signal is considered “OFF”. Adjustments to the duty ratio, switching frequency and circuit topologies all factor into how a power converter operates. After successfully producing an open-loop design, the next step would be to incorporate a feedback loop as part of the closed-loop design which allows the system to maintain a steady and constant output as disturbances are introduced to the system. This process involves implementing PI (Proportional and Integral) tuning which is key to the system adapting to disturbances. Our research up until this point let us set a general roadmap for the design process as we transitioned our work to the simulations.

Reflecting on the research portion of the project, we could have been more focused and efficient during these early steps. I think this is a very normal experience for students and junior engineers applying themselves to new undertakings. Reflecting on our early team meetings during the research portion, it’s clear that we still didn’t have a comprehensive grasp on the scope of the design process that we were about to initiate. As we progressed into the design and simulation phases of the project, we began to identify more specific areas of continued research. So even though we were moving past the official research phase, we continued to build upon our collective knowledge base as it became necessary during various challenges.

From our initial research, we decided to pursue an overall topology similar to the one shown in **Figure 1**. We needed a converter to boost the PV array output up to that of the DC bus, which would be converted from DC to AC via the Sine PWM inverter. Additionally, to incorporate the BES we needed a converter capable of bidirectional power flow since the BES would act as either source or sink depending on the output of the PV array as well as the needs of the grid. The specifics of each converter design changed as we progressed further in the design process, but the overall topology remained similar to that shown in **Figure 1**.

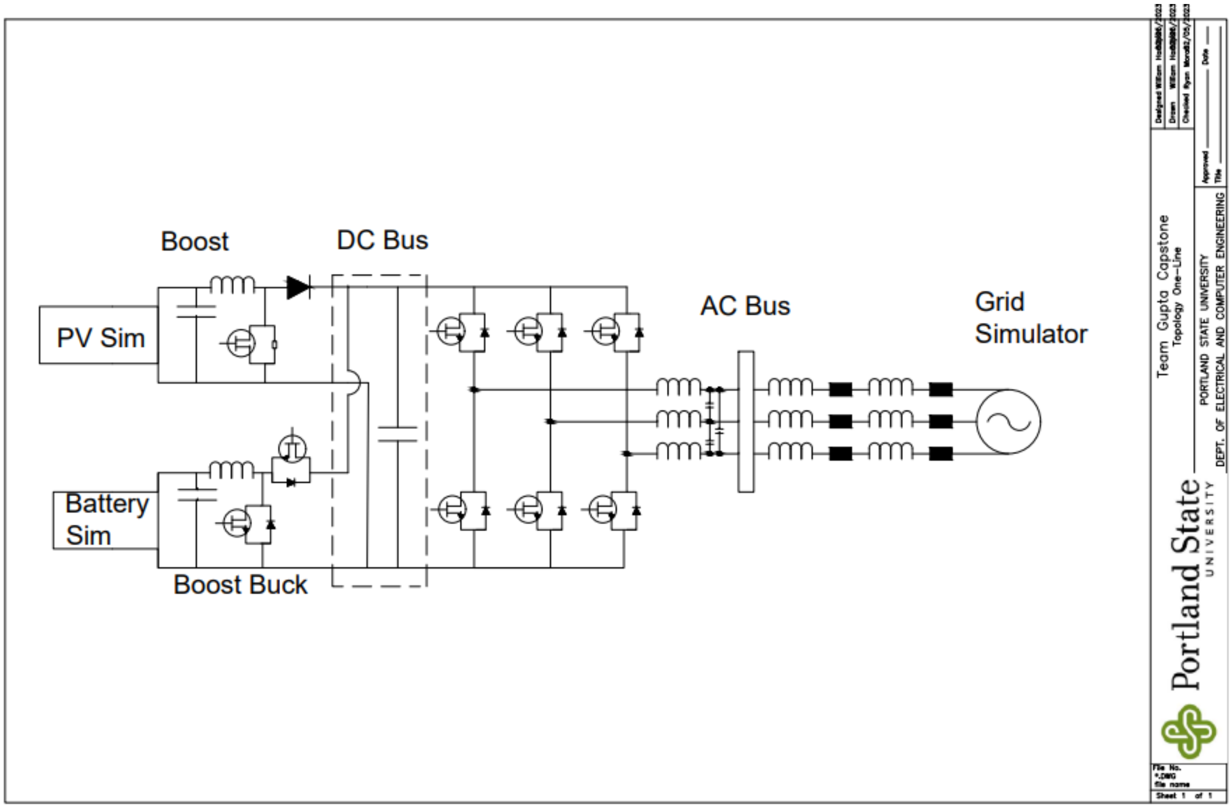


Figure 1: Initial topology proposal

Simulations

With a basic understanding of how we needed to approach the circuit designs, we began our design/simulation phase of the project. As mentioned previously, we first needed to design the open loop versions of the circuits. This was largely performed within LTSpice, a simple and widely used circuit simulation program that we all had experience with from many of our undergraduate courses. LTSpice is a useful tool, but is limited in its ability to design a comprehensive control system and doesn't provide a clear method of transferring our work to a working prototype. After exhausting the usefulness of LTSpice, we needed to design the closed loop versions of these circuits and we chose to do so within Simulink. Simulink is a product within the MATLAB suite of programs and in addition to having a robust set of toolboxes for control design, it also has easy tools that allowed us to convert our simulation to readable code for the microcontroller that would produce the control signals for each of the converters.

The open loop designs assume a constant input and therefore only require a constant duty ratio on the switching signal to achieve a steady output. In hindsight this was the simplest part of the capstone project, but we still struggled for longer than expected due to our overall unfamiliarity. After much internal discussion and failed attempts with other topologies, we settled on using a boost converter for the PV-DC bus connection which would take a variable source in the PV array and maintain a constant output voltage of 340V to regulate the DC bus to which our other two circuits would be coupled. A boost converter was chosen since the PV array is always expected to produce less voltage than what we want to maintain at the DC bus and power flow is only necessary in one direction. The BES would be coupled to the DC bus via a bidirectional boost converter such that the battery could be charged and discharged based on the needs of the grid and the availability of the PV array. This design posed the most difficulty early on since we needed bidirectional power flow and there existed many options for this functionality. We chose a bidirectional boost converter with the hopes that the control would be similar enough to the boost converter used for the PV array and we could adapt the same circuit boards that we were using for that design. Finally, to tie the system to the grid we chose to implement a sine Pulse Width Modulation (PWM) Inverter that converts the DC voltage at the bus to the three phase AC voltage at the grid point of connection. Additionally, since we would be maintaining the DC bus with the power converter connected to the PV array we could assume constant input and output for the sine PWM inverter making the control design simple. Below are the LTSpice simulations of these three circuits with their respective output waveforms.

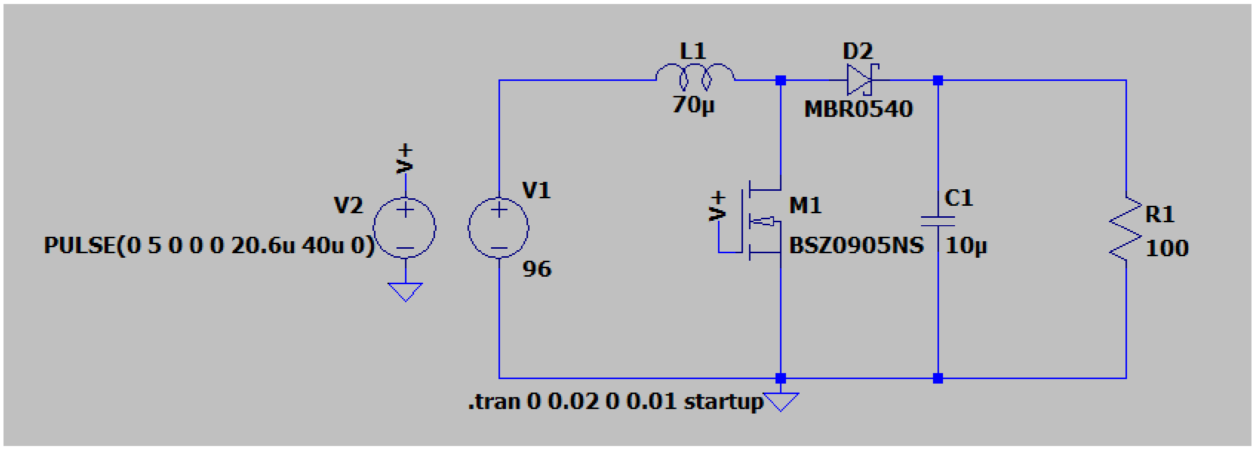


Figure 2: Boost converter, LTSpice topology

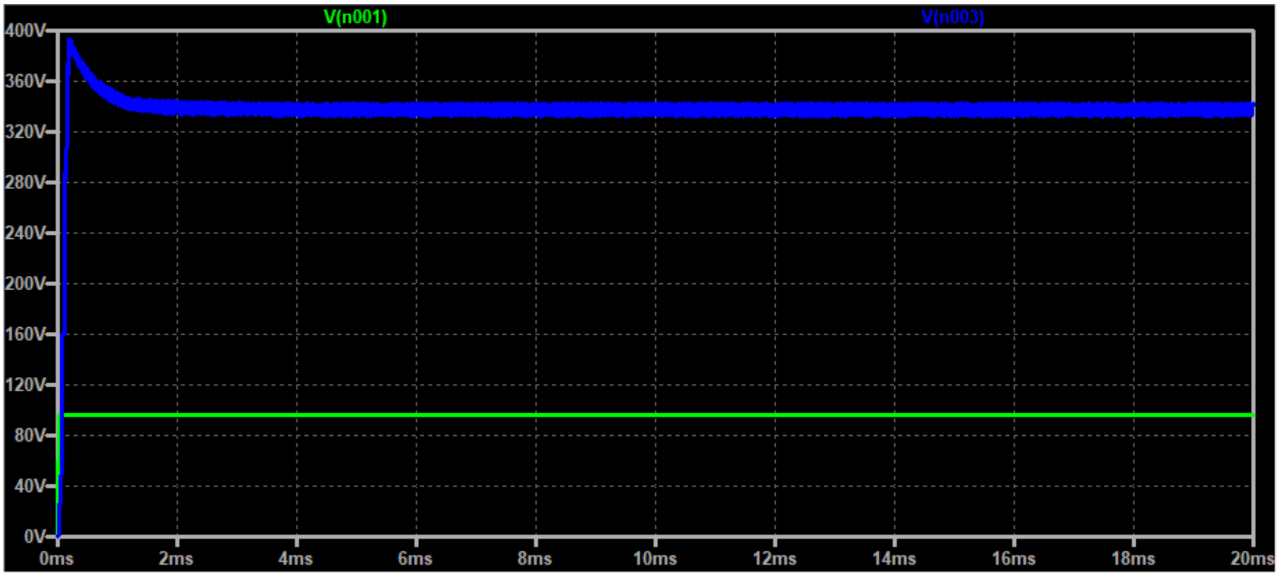


Figure 3: Boost converter, LTSpice simulation output

The Boost converter connecting the PV array and the DC bus was the simplest to design since we didn't need to deviate from standard designs for our purposes. Initially, we chose to use 96V as our maximum expected input to the converter and chose a duty ratio which boosted that voltage up to 340V. We used the following equations to determine the appropriate duty ratio and inductor sizing based on our design parameters (Mohan 47):

$$\frac{V_{out}}{V_{in}} = \frac{1}{1-D}, L = \frac{V_{in}}{\Delta i_L} D T_s$$

Where, D is the duty ratio, L is the inductor, Δi_L is the desired current ripple, and T_s is the switching period. The duty ratio is represented by the pulse function V+ in **Figure 2** and **Figure 3** shows the input voltage in green and output voltage in blue, which settles at 340V.

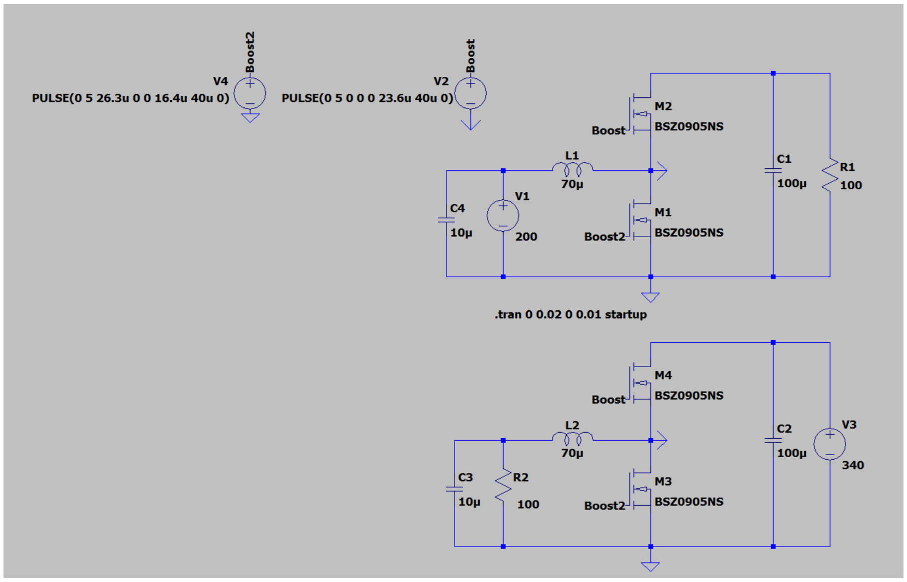


Figure 4: Bidirectional Boost converter, LTSpice topology

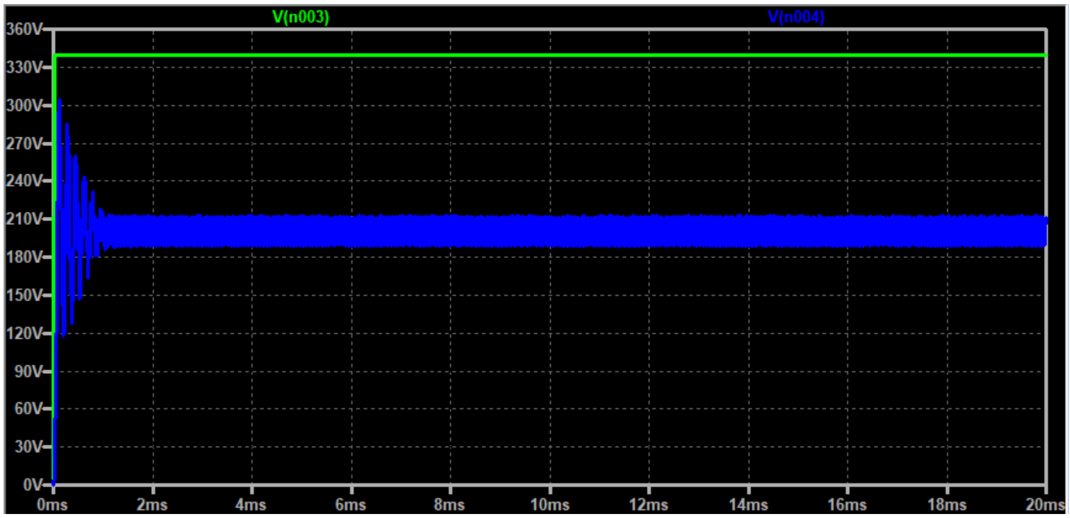


Figure 5: Bidirectional Boost converter buck mode, LTSpice simulation output

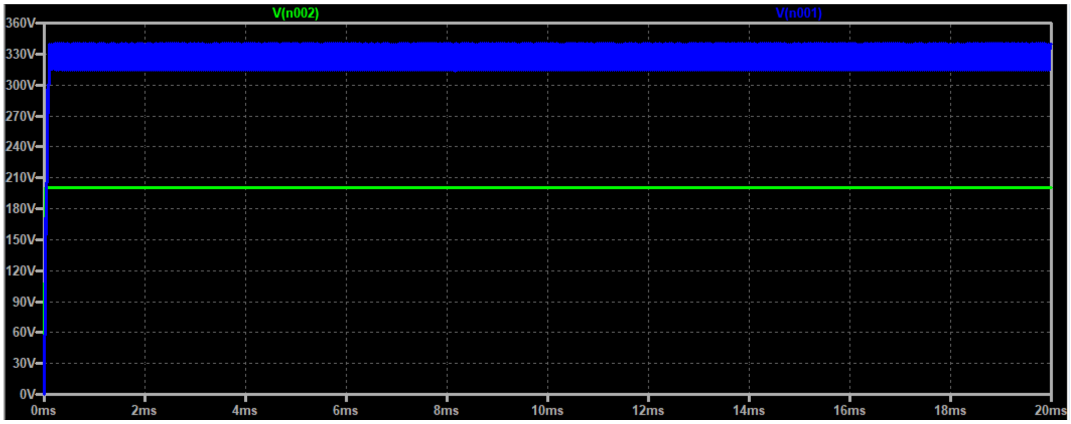


Figure 6: Bidirectional Boost converter boost mode, LTSpice simulation output

Since we were using another Boost converter for the bidirectional DC-DC conversion, we used the same equations to determine our duty ratios and inductor sizing. **Figure 4** shows our initial design for the bidirectional boost converter, which was necessary to represent as two separate circuits for the purposes of LTSpice simulation. The only difference between each mode is the inversion of the switching signals to each MOSFET, represented as Boost and Boost2. **Figure 4** shows the simulation of the buck mode, which would involve power flow from the DC bus (340V) to the BES (200V). **Figure 5** shows the simulation of the opposite mode, with power flow going from the BES to the DC bus. Both modes can be differentiated by identifying the constant voltage input in each.

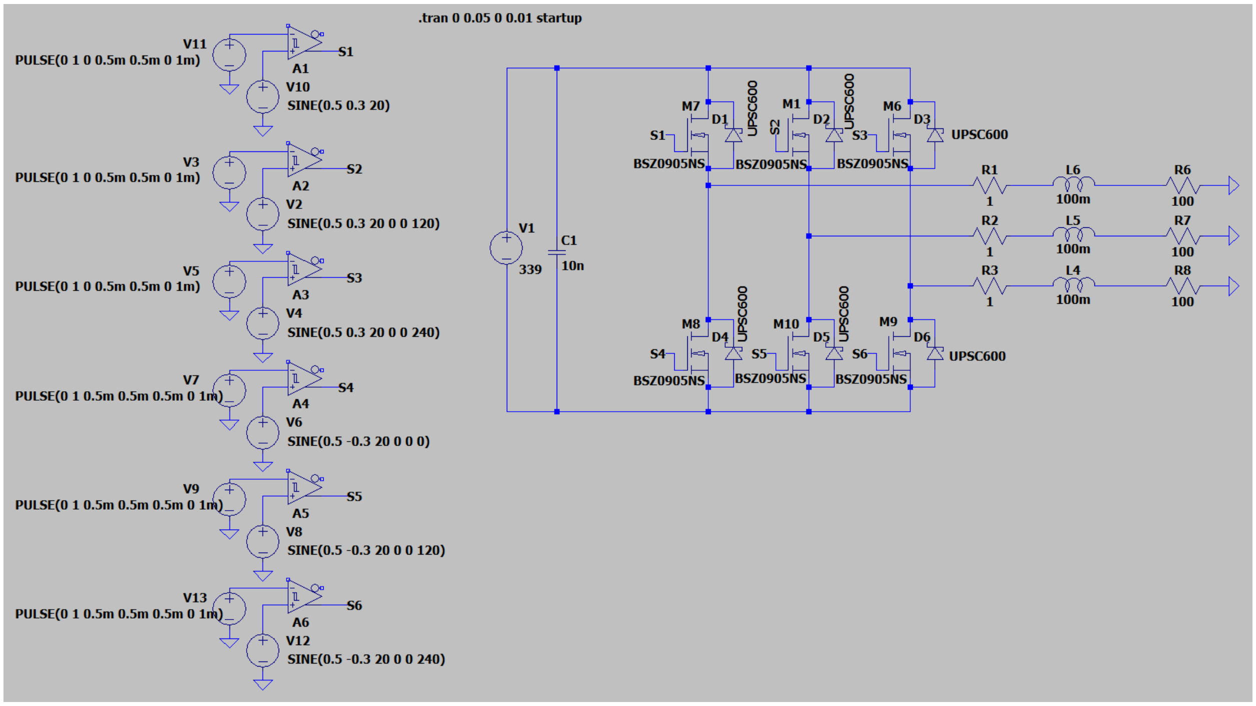


Figure 7: Sine PWM inverter, LTSpice topology

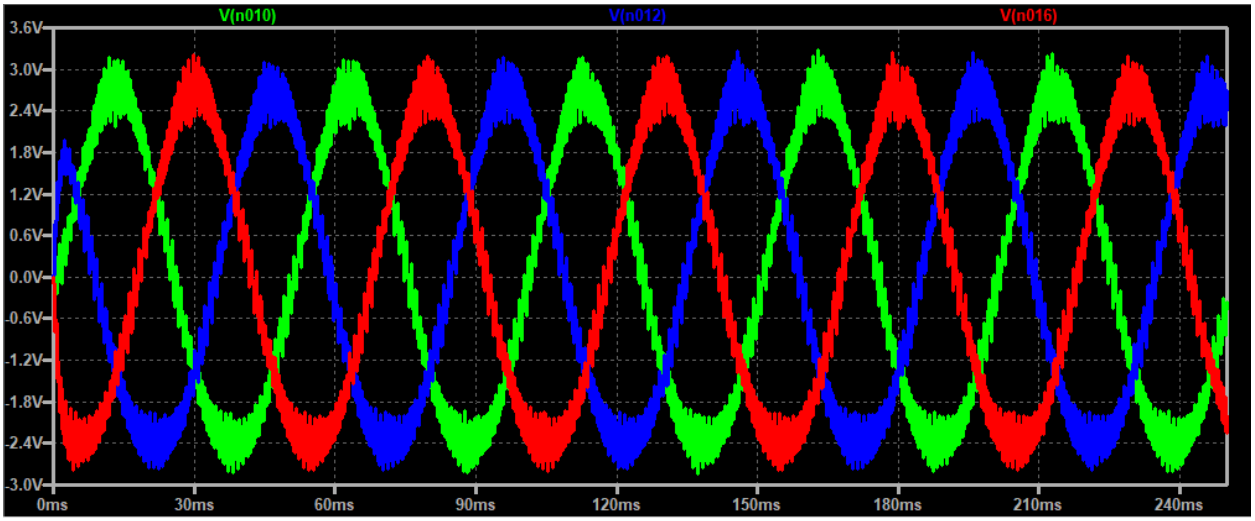


Figure 8: Sine PWM inverter, LTSpice simulation output

Figure 7 and **Figure 8** show the LTSpice topology and simulation results for the Sine PWM inverter, which takes a constant DC input from the DC bus and converts it to 3 phase AC which feeds to the grid. Note that the amplitude of the output voltage waveforms is around 3V when it should be operating around half of the input voltage (~170V). We couldn't resolve this bug, but our parameters were verified by our capstone advisor so we are confident that this was just an issue within LTSpice.

With our open loop designs finished, we were ready to move onto designing the closed loop control systems for our circuits. The main idea of this part of the design was to add an error calculation to the output that dictates the duty ratio of the PWM signal that drives the switching devices of the circuit and therefore the output. This is the fundamental principle behind closed-loop control and hinges on determining the Proportional (P) and Integral (I) constants that are used to reduce the error of the output as compared to a fixed reference value. This process was painstaking as we tried multiple approaches with little success. Any of the methods likely could have worked well, but we still lacked a foundation of experience that made implementing these methods confusing and difficult.

The first method we spent months trying to adapt for our purposes was a theory-based approach detailed in Mahima Gupta's thesis. (Gupta) Since Dr. Gupta was our capstone advisor, this method seemed promising because we had a willing teacher ready to help walk us through the process. However, this process was unintuitive since we didn't have a fundamental understanding of the theory behind the converters. Although similar to our project, the thesis was different enough that we had to adapt it for our purposes. This proved very difficult and we decided to abandon it in favor of making headway with other PI tuning methodologies.

The second approach taken was to use Simulink's built-in PI tuning feature, but this required another process where we needed to help Simulink build a linearized version of the circuit model so that the program could process the data. We were able to progress through most of this process but became hung up on a step that needed us to adjust a frequency response plot of the linearized system according to our desired design criteria. However, we couldn't figure out how to translate our desired design goals to the frequency criteria which was presented via this frequency response plot. Once again, an issue with us not fully understanding the theory behind the design that we were attempting.

From the research portion conducted earlier, we found that the most common method of tuning PI control was through an educated trial and error, where an engineer slowly adjusts the different constants until a desired output is achieved. We chose to pursue other methods first since we assumed that we didn't have the experience to make informed adjustments. However, after failing to implement the two other methods explained above, we decided to try this manual approach to PI tuning. I first used the Ziegler-Nichols method of PI tuning, which involved slowly incrementing or decrementing the Proportional constant until a steady output was achieved and then using this final value in conjunction with an equation to determine both of our PI constants. The Ziegler-Nichols method didn't fully translate for our purposes, but the mechanical trial and error process helped establish an understanding of the system we were tuning. Although we were met with challenges and multiple failures along the way, throughout the design process we began to build an intuitive basis. We were able to successfully tune the Boost converter (**Figure 2**) so that the system had a stable and constant output, even when disturbances to the input voltage and load were introduced. At first it seemed we had unwittingly stumbled onto the solution, but upon reflection it's clear to me and my group members that the months spent with the simulation while trying different tuning methods had incrementally built to this solution.

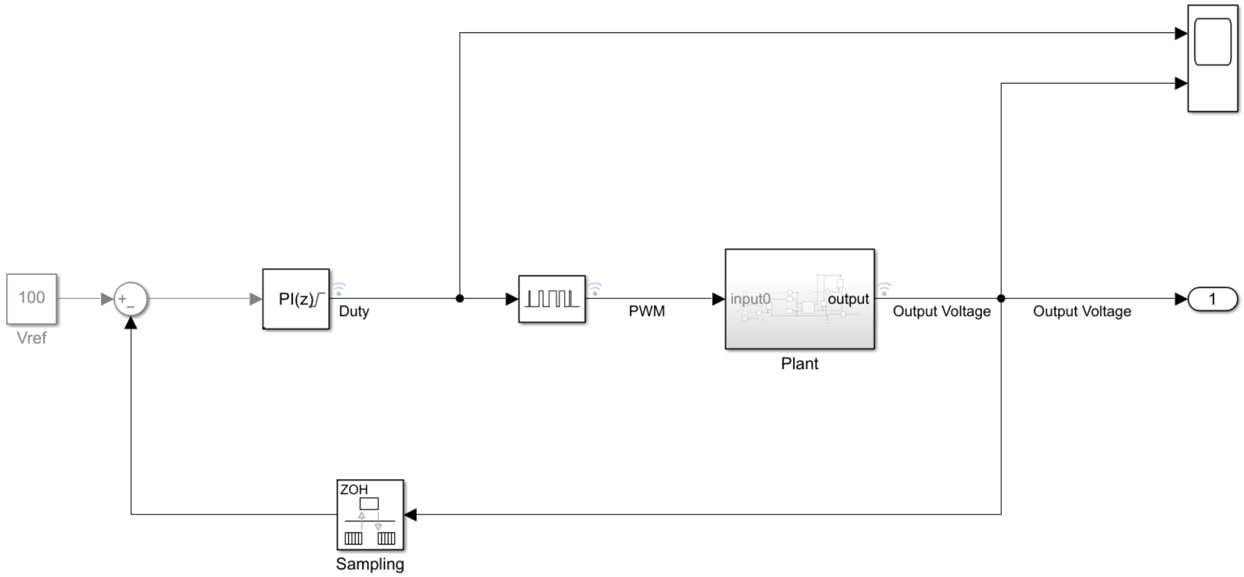


Figure 9: Boost converter closed loop control design, Simulink

Figure 10: Boost converter closed loop output with input voltage and load disturbances

Figure 9 shows the closed loop feedback design, where the output voltage is fed back and compared to a voltage reference (Vref) and then manipulated by the PI block to get a duty ratio based on the current output. **Figure 10** has the output of the simulation, with disturbances to the input voltage and load resistance, which is observed by the spikes after the simulation starts. The system has steady output equal to the reference, indicating a successful control. The thick lines mean that there are oscillations around the steady state, but these are within the design tolerance of +/- 5%.

To maintain simplicity in our designs, we chose to not design a control loop for the DC-AC conversion performed by the Sine PWM Inverter. Since we were maintaining a constant DC bus voltage with our control on the other two converters and the grid is assumed to be constant, both input and output to the inverter would not change. A control system could be implemented to maintain aspects like output current or account for faults elsewhere in the topology.

Implementation and Testing

Unfortunately, by the time we progressed to the Testing phase of the project, we were about a month away from the end of the term. With collaboration from our Capstone advisor, we decided to cut parts of the final deliverable so that we could still produce something that was adequately tested instead of trying to do everything and having little to show for it when the deadline finally arrived. We agreed to move forward with low voltage testing for the DC-AC functionality, which includes the Boost converter and the Sine PWM inverter but not the bidirectional converter on the BES.

For the implementation of both the Boost converter and the Sine PWM inverter, we repurposed high voltage (rated for use up to 650V) half-bridge circuit boards from the Portland State University power lab and used Arduino Duo controllers to produce our constant switching signals during the open-loop testing. Closed-loop testing would integrate a Texas Instruments microcontroller to supply the switching signals for all of the converters.

With this development, we outlined our testing procedure as follows:

1. Open-loop DC-DC test (boost converter)
2. Sine PWM inverter test
3. Open-loop DC-AC test (Sine PWM inverter + boost converter)
4. Open-loop DC-AC test using the AC Load Simulator
5. Closed-loop DC-DC test (boost converter using a microcontroller for control)
6. Closed-loop DC-AC test
7. Closed-loop DC-AC test using the AC Load Simulator

Open-loop DC-DC Test

While using probes to measure the output and monitoring the input with the power supply, we slowly increased the input voltage from 1V to 50V with a constant duty ratio of 50% applied to the switches. Using the test data, we were able to make an efficiency map (**Figure 11**) for the Boost converter. Our test setup for this part is shown in **Figure 12**. Each half-bridge board has two switches that are controlled via a single USB-B connection, connections at each corner for the V+ and V- rails, and a connection in series with an on-board inductor at Vm. For the purposes of the Boost converter testing, we added a 1mH inductor in series with the input (Vm) and connected a 100 Ohm load on our output (in parallel to the V+ and V- connections). We collected data from the power supply for our input voltage and current and used oscilloscope probes on the load for our output voltage and current.

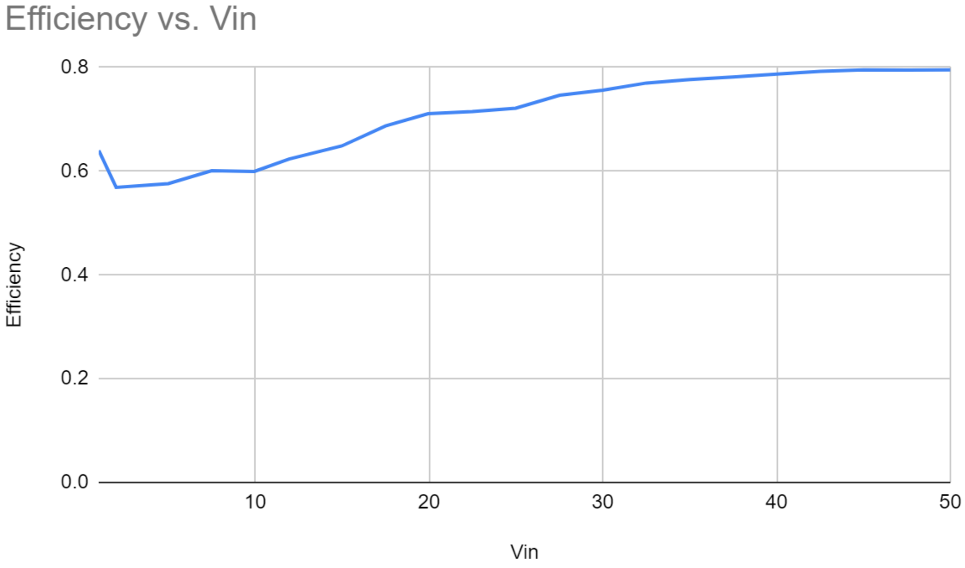


Figure 11: Open-loop Boost converter efficiency plot

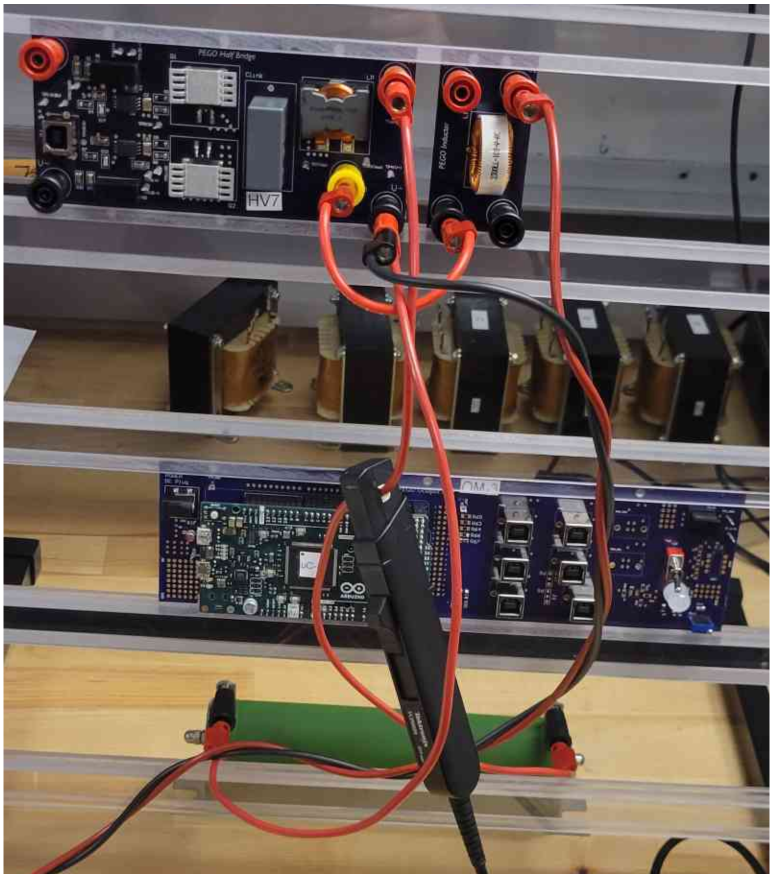


Figure 12: Open-loop Boost converter test setup

Sine PWM Inverter Test

We only verified that the DC-AC conversion was operating as expected for this testing and didn't take measurements. In hindsight, it would have been useful to take output and input measurements at this stage to compare the efficiencies between each of the converters on their own versus the efficiency with them in operation together. Additionally, I don't have a picture to show our test setup for this, but you can see part of it in **Figure 14** below when we integrated the Boost converter and the Sine PWM inverter together for testing. Our 3-phase loads were connected in series to each of the phase outputs of the inverter, using 56 Ohm resistor banks alongside two 1mH inductors for each phase. Finally, three half-bridge boards were used for each of the switch pairs in the Sine PWM inverter design.

Open-loop DC-AC Test

Next we tested both the Boost converter and Sine PWM inverter together. **Figure 14** shows our test setup, which involves 4 total half-bridge boards and two Arduinos. We used a separate Arduino Duo for the inverter switching signals since we encountered issues trying to implement the code for each purpose on the same board. Additionally, we added a DC link capacitor in between the two converters to help smooth out some of the noisy EMI we encountered during the testing process. **Figure 13** is an efficiency plot for the combined circuits.

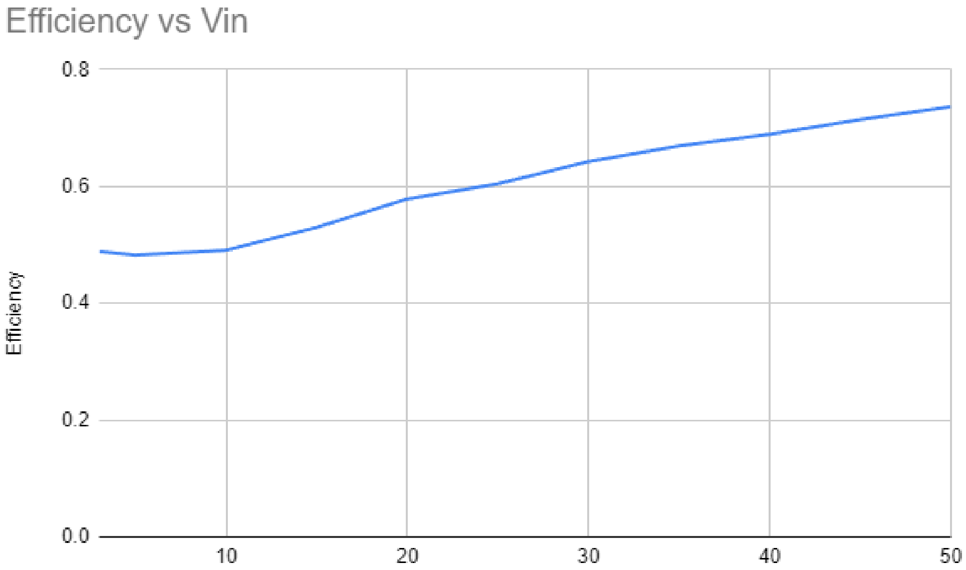


Figure 13: Open-loop DC-AC efficiency plot

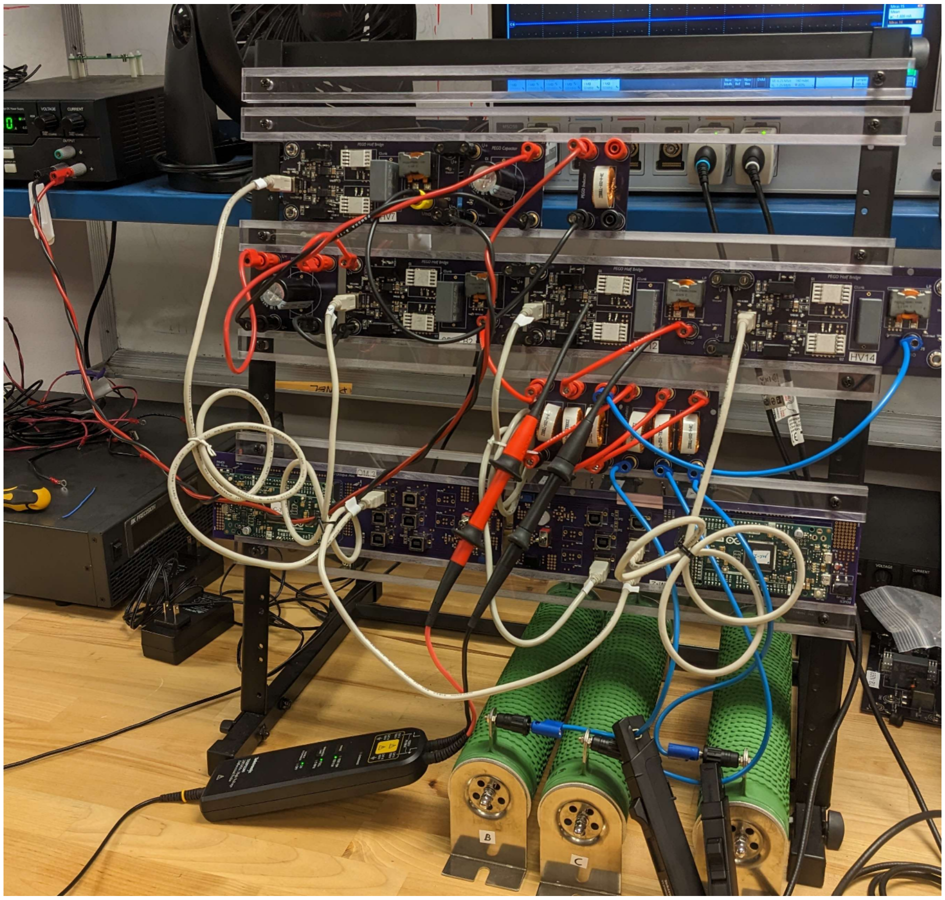
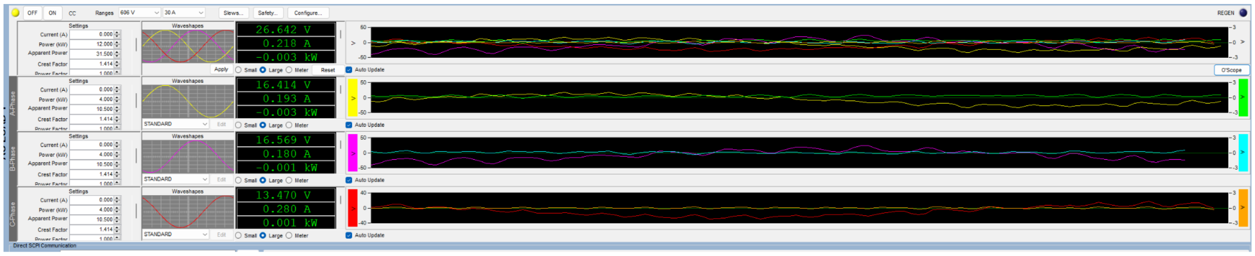


Figure 14: Open-loop DC-AC test setup

Open-loop DC-AC Test with AC Load Sim

One of our biggest goals during the testing phase was to successfully use the NHR AC Load Simulator with our converters. However, we didn't get to this part until the very last week to still run experiments. At the time of writing my final revisions to this paper, we were still planning on performing last minute debugging and gathering better documentation of how we connected our system. The back of the Load Sim has connections for each of the three phases and neutral, which we connected to by customizing banana cables and connecting to our system above the inductors on our 3-phase loads. **Figure 15** and **16** shows the output from our converter represented in the software of the Load Sim. Note that Figure **16** has a negative DC offset (the center of the sinusoids is below 0V) and this will be the primary focus of our debugging steps. Despite our lack of results reflected in this paper, we are satisfied that we were able to progress to this step at all given the time frame we had available for testing.



Ranges: 606 V, 30 A

Settings: Current (A) 0.000; Power (kW) 12.000; Apparent Power 31.500; Crest Factor 1.414; Power Factor 1.000

26.642 V
0.218 A
-0.003 kW

A-Phase — Settings: Current (A) 0.000; Power (kW) 4.000; Apparent Power 10.500; Crest Factor 1.414; Power Factor 1.000; Waveshape: STANDARD

16.414 V
0.193 A
-0.003 kW

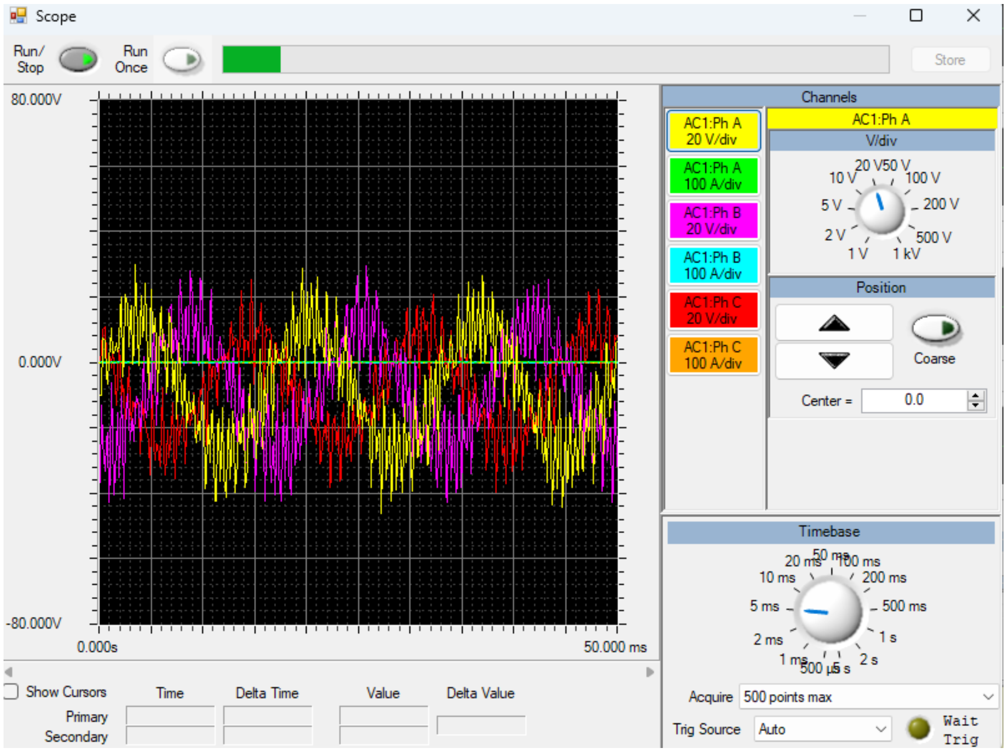
B-Phase — Settings: Current (A) 0.000; Power (kW) 4.000; Apparent Power 10.500; Crest Factor 1.414; Power Factor 1.000; Waveshape: STANDARD

16.569 V
0.180 A
-0.001 kW

C-Phase — Settings: Current (A) 0.000; Power (kW) 4.000; Apparent Power 10.500; Crest Factor 1.414; Power Factor 1.000; Waveshape: STANDARD

13.470 V
0.280 A
0.001 kW

Figure 15: Load Sim Phase Results



Channels: AC1:Ph A 20 V/div; AC1:Ph A 100 A/div; AC1:Ph B 20 V/div; AC1:Ph B 100 A/div; AC1:Ph C 20 V/div; AC1:Ph C 100 A/div

Center = 0.0

Acquire: 500 points max

Trig Source: Auto

Figure 16: Load Sim Oscilloscope results

Results

Overall, our results are disappointing considering the goals we set at the beginning of the project. We simply ran out of time to perform all the tests that we wanted to complete. The overall efficiency of our converter (**Figure 13**) is decent considering the equipment we had access to and the testing environment. All of our testing occurred in the back of the Power Lab, right next to the Grid Sim that was being used concurrently for another capstone project. We experienced a lot of noise throughout our testing and we think a large source of it was the Grid Sim that was in operation at most points. Some of the noise was mitigated by shortening the conductors between components and braiding longer conductors.

If the capstone project didn't have a hard deadline of the end of the term, we would have continued with our original testing plan. We were unable to incorporate the TI microcontroller which meant we couldn't verify the control design that I had spent most of my time creating. As previously mentioned, we never even tried to test the bidirectional converter due to running out of time. While considering all of our shortcomings, we did still verify some of our designs including the open-loop Boost converter and the Sine PWM inverter as well as integrating with Load Sim.

Reflection

I've already touched on our difficulties early in the project during the Research phase. Due to our inexperience, we spent too long not asking the right questions. I am confident that if we attempted this project again with the knowledge we have now, we could have delivered on all of our initial goals in the 6 month time period. But that isn't how school projects go and you have to move on from them at some point.

One aspect of the project that I didn't fully explore in this paper was the iterative process we employed for our organization of the capstone group. We held multiple meetings where the focus was to identify administrative processes that we did well and those we did poorly, and then suggest changes to address those we did poorly. This allowed us to adjust to issues before they got out of hand. I want to credit these focused meetings with us being able to produce any semblance of results since without these improvements we likely would have been bogged down by issues not related to technical shortcomings, such as ensuring a member was taking notes during meetings and having assigned tasks that individuals were responsible for completing before a specific date.

I learned much from the experience working on this Capstone. I enjoyed the logical puzzle that the controls design posed, especially after having established a fundamental basis. Adjusting values a little here and there to finally get the response you are looking for is immensely satisfying. Prior to this project, I thought I was only going to apply for classic power engineering positions post-graduation. Now I am considering more and have expanded my job search to include controls engineering positions. If nothing else, this project has given me the confidence that I can succeed in fields that I lack experience.

Works Cited

Busquets-Monge, Sergio, et al. “A Bidirectional Multilevel Boost–Buck DC–DC Converter.” *IEEE Transactions on Power Electronics*, vol. 26, no. 8, Aug. 2011, pp. 2172–2183, https://doi.org/10.1109/tpel.2011.2105508. Accessed 9 Apr. 2022.

Gupta, Mahima. *Cascaded Power Converters with Reduced Link Capacitors*. University of Wisconsin-Madison, 10 May 2019.

Mohan, Ned. *Power Electronics : A First Course*. Hoboken, N.J., Wiley, 2012.

Paul, Prof, et al. “A High Gain Bidirectional SEPIC Converter.” *International Journal of Engineering Research and Applications Www.ijera.com*, vol. 11, 2021, pp. 36–43, https://doi.org/10.9790/9622-1103023643. Accessed 10 June 2023.