Two-Phase Boost Converter

A Master Thesis

Presented to

The Faculty of California Polytechnic State University

San Luis Obispo

In Partial Fulfillment

Of the Requirements for the Degree

Master of Science in Electrical Engineering

By

Tadeus Gunawan

December 2009

© 2009
Tadeus Gunawan
ALL RIGHTS RESERVED

COMMITTEE MEMBERSHIP

TITLE:	Two-Phase Boost Converter
AUTHOR:	Tadeus Gunawan
DATE SUBMITTED:	December 2009
COMMITTEE CHAIR:	Taufik, Professor
COMMITTEE MEMBER:	Bill L. Ahlgren, Associate Professor
COMMITTEE MEMBER:	Dale Dolan, Assistant Professor

ABSTRACT

Two-Phase Converter

Tadeus Gunawan

A boost converter is one of the most efficient techniques to step up DC input voltage to a higher needed DC output voltage. The boost converter has many possible applications, such as in a photovoltaic system, hybrid car and battery charger. The proposed prototype in this report is a proof of concept that a Two-Phase Boost Converter is a possible improvement topology to offer higher efficiency without compromising any advantages readily offered by a basic boost. The prototype is designed to be able to handle up to 200 watts of output power with an input of 36 volts and an output of 48 volts. This paper goes through step-by-step the calculation, design, build and test of a Two-Phase Boost Converter. Calculations found in this paper were done on Mathcad and the simulations were done on LTSpice and Pspice. These include converter's efficiency and other measures of converter's performance. Advantages, disadvantages as well as possible improvements of the proposed topology will be presented. Data collected and analyzed from the prototype were done on a bench test, not through an actual application.

Keywords: Two phase boost converter.

ACKNOWLEDGEMENT

It is a pleasure for me to thank those who made this thesis possible. First I would like to give my thanks to Dr. Taufik for all his help and guidance throughout my college career especially during this thesis project. It has been a fun and challenging journey and thank you for being my advisor and teacher. I also want to extend my thanks to my friends and/or fellow students. Arief Hernadi, Kyaw Kyaw Htin, and Jonathan Paolucci thank you for all the help and support during my design processes and late night consultations. Also, I thank Rithy Chhean and Mike McCarty for helping me with the troubleshooting and data gathering. Lastly, I want to thank my family for all their support that helped me through college. I do not know where I would be without them.

TABLE OF CONTENTS

	Page
LIST OF TABLES.....	viii
LIST OF FIGURES	ix
I. Introduction	1
II. Background	3
2.1 Continuous Conduction Mode (CCM).....	8
III. Requirements	18
IV. Design Calculations	20
4.1 Duty Cycle	20
4.2 Inductor	21
4.3 Input Capacitor.....	23
4.4 Output Capacitor	24
4.5 Power MOSFET.....	26
4.6 Voltage and Current Sense Resistors	29
5.1 Output Voltage.....	33
5.2 Output Current	34
5.3 Switching Signals.....	36
5.4 Inductor Current.....	38
5.5 Converter's Efficiency	41
5.6 Load Regulation.....	45
VI. Hardware Design and Results.....	46
6.1 Hardware Components.....	46

6.1.1 Pulse Width Modulator	48
6.1.2 Inductor	50
6.1.3 MOSFET	50
6.1.4 Schottky Diode.....	51
6.1.5 Input and Output Capacitors	51
6.2 Final Hardware Design	52
6.3 Hardware Results and Analysis	53
6.3.1 Output Voltage Ripple	53
6.3.2 Switching Signal	54
6.3.3 Inductor Current Ripple	58
6.3.4 Current Sense	61
6.3.5 Load Test Results.....	63
6.3.6 Load Regulation.....	65
6.3.7 Line Regulation.....	66
6.3.8 Transient Response	67
6.4 Hardware Troubleshooting	68
6.4.1 Output Voltage.....	68
6.4.2 Line Regulation.....	71
VII. Conclusion.....	72
BIBLIOGRAPHY	74

LIST OF TABLES

Table	Page
3-1. Design requirements for Proposed Two-Phase Boost	19
5-1. Simulated Efficiency from 10% - 100% Load.....	44
5-2. Simulated Load Regulation.....	45
5-3. Simulated Line Regulation.....	45
6-1. Hardware Efficiency from 10% - 100% Load.....	64
6-2. Hardware Load Regulation.....	66
6-3. Hardware Line Regulation.....	66

LIST OF FIGURES

Figure	Page
2-1. Boost Convert.....	4
2-2. Two-Phase Boost Converter.....	7
2-3. Switching Signals of Two-Phase Boost.....	7
2-4. Ideal Boost Converter with Conducting Switch.....	8
2-5. I_L vs. Time and V_L vs. Time in CCM with Conducting Switch.....	10
2-6. Ideal Boost Converter with Open Switch.....	11
2-7. I_L vs. Time and V_L vs. Time in CCM with Open Switch in Each Phase.....	12
2-8. Ideal Current Waveform for Input Capacitor.....	15
2-9. Ideal Current Waveform for Output Capacitor.....	15
2-10. Frequency Multiplication in Two-Phase Boost	16
4-1. RMS Ripple Input Current Rating.....	23
4-2. Input Capacitor Current.....	24
4-3. RMS Ripple Output Current Rating.....	25
4-4. Output Capacitor Current.....	26
4-5. ρ vs. Junction Temperature for FDD3682.....	28
4-6. Voltage Divider.....	30
5-1. Two-Phase Boost Schematic Using LTSpice.....	32
5-2. Output Voltage Ripple.....	33
5-3. Overlaid unfiltered Output Current of Phase 1 and Phase 2.....	34
5-4. Filtered Output Current Ripple.....	35
5-5. Phase 1 Switching Signals.....	36

5-6.	Phase 2 Switching Signals.....	37
5-7.	Superimposed Duty Cycles.....	38
5-8.	Inductor Current at Phase 1.....	39
5-9.	Inductor Current at Phase 2.....	40
5-10.	Overlay of Phase 1 and Phase 2 Inductor Current.....	40
5-11.	Low Load Inductor Currents.....	41
5-12.	Input Power Ripple.....	42
5-13.	Output Power Ripple.....	43
5-14.	Efficiency vs. Load from Simulation.....	44
6-1.	Top Layer of the Final PCB Design.....	47
6-2.	Bottom Layer of the Final PCB Design.....	47
6-3.	LT3782A R_{FREQ} vs. Frequency.....	49
6-4.	Output Voltage Ripple.....	54
6-5.	Phase 1 Duty Cycle.....	55
6-6.	Phase 2 Duty Cycle.....	56
6-7.	Overlaid Duty Cycle.....	57
6-8.	First Phase Inductor Current.....	58
6-9.	Second Phase Inductor Current.....	59
6-10.	Overlaid Inductor Current.....	61
6-11.	Phase 1 Current Sense Waveform.....	62
6-12.	Phase 2 Current Sense Waveform.....	62
6-13.	Load vs. Efficiency for Hardware Test.....	65
6-14.	Output Voltage Overshoot from Full Load to Low Load.....	67

6-15.	Output Voltage Overshoot from Low Load to Full Load	68
6-16.	Phase 1 Inductor Current during Low Load.....	70
6-17.	Phase 2 Inductor Current during Low Load.....	70

I. Introduction

DC – DC converters are widely used in today's industrial or commercial electronic devices to manipulate a dc voltage source. As the name implies, DC – DC converters work exclusively to take a dc voltage input and convert it to output a different level of dc voltage. They can either step-up or step-down the input dc voltage while maintaining minimal power loss during the process. There are many different topologies available for use such as Buck (step down), Boost (Step up), Flyback, Push-Pull, etc. This versatility is the reason that DC – DC converters are popular among many current electronic devices.

Many portable devices and even industrial grade machines today depend on a reliable and efficient DC – DC converters. Examples such as hybrid or electric cars, mp3 players, cell phones, laptops, photovoltaic systems to name a few contain sub-systems that require different and constant level of voltage to operate properly. From power electronics' point of view the main issue revolves around the batteries. The charge in the batteries will start to decline as time passes, therefore to provide constant and different level of voltages it will have to use multiple batteries for a single device. However, with DC – DC converters we are able to provide a more efficient solution than extra batteries because of the following two reasons. First, DC – DC converters are designed to have a certain load and line regulations which provides “cushion” in the case of unstable or changing input level and/or load. Secondly, these converters are able to split a single constant source to several different voltage levels by using a multi-winding transformer. Other benefits of using DC – DC converters include the ability to operate the system in

high switching frequency operation. This would further result in a relatively small system crucial especially in portable and vehicle applications.

II. Background

There are two categories of DC – DC converters: Isolated and Non-Isolated DC – DC converters. Isolated DC – DC converters, as the name implies, electrically isolate the output from the input using a high frequency transformer. The transformer's turns ratio dictates the relationship between the input and output voltage. By having multiple secondary windings, isolated converters are able to provide multiple levels of output voltages. The size of the transformer usually dictates the size of isolated converter; hence they generally are large in size. On the other hand the Non-Isolated topologies can vary the dc output voltage and provide different level of output voltages without the use of transformer and require fewer components to implement.

Boost converter is also known as the step-up converter. It is one of the well known topology in the field of power electronics and like any other DC – DC converters it aims to achieve high efficiency conversion. The existence of boost converter relies on the discovery of semiconductor switches. These switches are much faster and reliable than other switches such as vacuum tubes and electromechanical relays. The semiconductor switches are able to operate at high frequency, from about 200 kHz to as high as 2 MHz. The frequency of Boost converters has an inverse relationship with the size of the magnetic components; therefore having a high switching frequency is beneficial to decrease the size of the converter.

Figure 2-1: Boost Converter

The proposed Two-Phase Boost Converter is a way to create a more efficient step up converter while at the same time decreasing the strain seen on a basic boost converter. Just like the other topologies of DC – DC converters Two-Phase Boost Converter utilizes semiconductor switches as the foundation of its operation. Semiconductor switches have fast turn on and off times and low on resistances which help to decrease losses during switching and conduction. Two-Phase Boost converter is basically two boost converters placed in parallel, sharing the same input and output as shown in Figure 2-2. In essence it does the same thing as a Boost converter, which is stepping up the fixed dc voltage input. However, unlike paralleled boosts the two phases in the proposed two-phase boost will never draw input current at the same time, rather the path alternates between the first and second phase. Thus each path only carries half of the output power and hence half the strain compared to a single Boost converter with the same output power requirements. This further implies that the switching power has to be equally split into the two phases. Consequently, the maximum duty cycle in each phase is limited to 50%. Figure 2-4 shows the comparison of switching signal for a single boost vs. two-phase boost

converter. Detailed analysis of the two-phase converter will be presented in the next chapter.

Just like the basic boost converter, there are three types of conduction modes for the Two-Phase Boost Converter: Continuous Conduction Mode (CCM), Discontinuous Conduction Mode (DCM) and Boundary Conduction Mode. During Continuous Conduction Mode the inductor current remains positive during switching while in Discontinuous Conduction Mode the inductor current is clipped at zero for a time during the switching. Lastly Boundary Conduction Mode happens when the lowest swing of the inductor current just barely reaches zero. In other words Boundary Conduction Mode happens when the inductor current is between CCM and DCM. In this thesis only continuous conduction mode will be considered in the design, calculation and hardware development of the proposed two-phase boost converter. A more detailed design equations and formula derivations would be covered in section 2.1.

Both Boost and Two-Phase Boost configuration share the same main components. As depicted in Figures 1 and 2, they both employ inductor, switch and diode. The inductor is being utilized as the main storage element that relies on its non instantaneous and linear current change following its Volt-Second Balance. The switch on the other hand depends on the converter's duty cycle. For efficiency purpose and in general, if the switch is conducting more than its idling time, a MOSFET with low $R_{ds(on)}$ is preferred; otherwise a MOSFET with low gate capacitance to minimize switching loss is the preferred choice. As for the diode when quick turns off responses are needed as in DC – DC converters, Schottky diode is commonly used. To increase the converter's efficiency further, a technique called synchronous rectification can be applied to the converter. With

this technique, the diode is replaced with another switch. Depending on the duty cycle, this process would replace the diode with either low $R_{ds_{on}}$ or low switching loss MOSFET for a more efficient process. Nowadays the synchronous rectification technique is prevalent in the market to create more efficient and long lasting products [1, 8, 10]. Quite the contrary, the multiphase topologies including the proposed two-phase converter have only been recently studied. Most of the multiphase topologies have been centered on applications that require a very low step down with output voltage at relatively high current such as those found in microprocessors [5, 2, 3, 6]. However, the potential of using multiphase topology in other topologies has not been extensively explored. One application that has recently gained an increased interest is the renewable energy. Many of the renewable energy sources provide dc output which needs to be stepped up in order to match the voltage level to that of the grid. Naturally, this is where boost converter plays an important role. Hence, this thesis is aimed to study the benefits of implementing a two-phase boost converter that may be used in conjunction with any renewable energy sources that require a step up function. The design and simulation of the proposed converter will be preformed as explained in later chapters, Moreover, to exhibit the potential of the proposed converter, a hardware prototype is built and tested.

Figure 2-2: Two-Phase Boost Converter

Figure 2-3: Switching Signals of Two-Phase Boost

2.1 Continuous Conduction Mode (CCM)

Just like the basic boost converter, the two phase counterpart may operate in a Continuous Conduction Mode (CCM) or Discontinuous Conduction Mode (DCM). Taking into account that the two main switches in the two phases in this design would never turn on simultaneously, it is possible to analyze most of the CCM properties by only considering a single phase i.e. the basic boost. The analysis could be separated into two sections: analysis for closed switch and analysis for open switch.

Figure 2-4: Ideal Boost Converter with Conducting Switch

When the switch or MOSFET conducts the diode will be reverse biased creating the scenario depicted by Figure 2-4. Assuming an ideal converter, the switch loss is negligible and no current is flowing through the diode, creating a singular path for the input current to flow, through the inductor and back to the negative end as represented in Figure 2-4 by the I_L current loop. The voltage across the inductor would then be equal to the voltage source, producing the expression below.

$$V_s = V_{L-ON}$$

$$V_s = L \frac{di_L}{dt}$$

$$\frac{V_s}{L} = \frac{di_L}{dt}$$

Ignoring heat and natural fluctuations, the value of V_s and L can be assumed to remain constant. V_s is a positive DC input and the inductance, L , would also be a positive value. By using these assumptions, when the switch is closed the inductor current's rate of change or the slope of the current curve must therefore be linear and positive in value. By having a positive rate of change it can be hypothesized that the change in the inductor voltage would also be positive when the switch is conducting. Deriving the equation further, the change in the inductor current can be more accurately expressed by,

$$\frac{V_s}{L} = \frac{di_L}{dt} = \frac{\Delta i_L}{\Delta t}$$

Where $\Delta t - on = D * T$,

$$\begin{aligned} \frac{V_s}{L} &= \frac{\Delta i_L}{DT} \\ \Delta i_{L-ON} &= \frac{V_s}{L} DT \end{aligned}$$

D here represents the percentage of the operating frequency when the switch is conducting or known as the duty cycle of the converter. The value of a duty cycle is always bound between 0 and 1 or 0% to 100%. The higher the duty cycle, the more time the inductor has to store energy in the coils. T is the switching period, or the inverse of the operating frequency. For the proposed topology, since energy is split equally and alternately into two boost converters, the maximum duty cycle of each phase would therefore be 50%. All four components: input voltage, duty cycle, switching period and inductance are positive in an ideal Boost converter; therefore the total change in the inductor's current when the switch is conducting would always be a positive. Figure 2-5

depicts a graphical representation of the inductor current versus time during this specific time section. During t_{ON} the total change in the inductor's current is positive with a positive and linear rate of change throughout.

Figure 2-5: I_L vs. Time and V_L vs. Time in CCM with Conducting Switch

When the switch or MOSFET turns off, the diode would be forward biased, creating a current path between the input and output. This condition is shown below in Figure 2-6.

Figure 2-6: Ideal Boost Converter with Open Switch

I_D is equivalent to I_L assuming there is leakage across the open switch. The new voltage across the inductor:

$$V_{L-OFF} = V_s - V_o = L \frac{di_L}{dt}$$

Next, knowing that $\Delta t - Off = (1 - D) * T$ and applying the Volt-Second Balance, the transfer function may now be derived:

$$(\Delta i_{L-ON}) + (\Delta i_{L-OFF}) = 0$$

$$(V_{L-ON} * DT) + (V_{L-OFF} * (1 - D) * T) = 0$$

$$\frac{V_s * DT}{L} + \frac{(V_s - V_o) * (1 - D) * T}{L} = 0$$

$$(V_s * DT) + (V_s - V_o) * (1 - D) * T = 0$$

$$(V_s * D) + (V_s - V_o) * (1 - D) = 0$$

$$V_s D + V_s - V_o - V_s D + V_o D = 0$$

$$V_s - V_o(1 - D) = 0$$

$$V_o = \frac{1}{(1 - D)} V_s$$

The transfer function shows that in boost the output voltage is greater than the input voltage. Returning to our V_{L-OFF} equation:

$$L \frac{di_L}{dt} = V_s - V_o$$

$$\frac{di_L}{dt} = \frac{V_s - V_o}{L} = \frac{\Delta i_L}{\Delta t}$$

$$\frac{\Delta i_L}{(1-D)T} = \frac{V_s - V_o}{L}$$

$$\Delta i_L = \frac{(V_s - V_o)}{L} (1-D)T$$

Since in a boost converter V_o is always greater than V_s , the rate of change and the total change in the inductor current would be negative during t_{OFF} .

Figure 2-7: I_L vs. Time and V_L vs. Time in CCM with Open Switch in Each Phase

Using this relation it is possible to find the duty cycle for a design given the required input and output voltage. Looking back to Figure 2-7, it can be seen that even though the current drops off in value, it never actually hits zero or below zero amps. This is because the Boost converter is being analyzed in CCM. The value of I_{Lmax} and I_{Lmin} can actually

be calculated by deriving the formula for average inductor current and inductor current ripple.

$$P_{IN} = P_{Out}$$

$$V_s * I_s = \frac{\overline{V_o}^2}{R}$$

$$I_s = \overline{I_L} = \frac{\overline{V_o}^2}{R * V_s}$$

Substituting V_o by the transfer function gives,

$$\overline{I_L} = \frac{\left[\frac{V_s}{(1 - D)} \right]^2}{R * V_s}$$

$$\overline{I_L} = \frac{V_s}{(1 - D)^2 * R}$$

For the proposed topology, the above equation still holds although the average inductor current in each phase should now be half of the average of the source current:

$$\overline{I_L} = \frac{I_s}{2}$$

It was found previously that,

$$\Delta i_{L-ON} = \frac{V_s}{L} DT = \frac{V_s D}{Lf}$$

Here f denotes the operating frequency of the converter which is equal to the switching frequency in each phase. So, adding the half of the inductor's peak to peak current ripple to the average current gives the formula of I_{Lmax} ,

$$I_{Lmax} = \overline{I_L} + \frac{\Delta i_L}{2}$$

$$I_{Lmax} = \frac{V_s}{(1 - D)^2 * R} + \frac{V_s D}{2Lf}$$

The value of R in the equation above for the proposed converter must refer to the half of the maximum output current. Hence, if maximum output power is P_{O-MAX} and output voltage is V_O , then R for each phase:

$$R = \frac{V_O^2}{P_{O-MAX} / 2}$$

$$R = 2 * \frac{V_O^2}{P_{O-MAX}}$$

Subtracting half of the current ripple, would instead give the formula to find I_{Lmin} ,

$$I_{Lmin} = \overline{I_L} - \frac{\Delta i_L}{2}$$

$$I_{Lmin} = \frac{V_S}{(1-D)^2 * R} - \frac{V_S D}{2Lf}$$

Where L is known as the critical inductance of the converter whose value is derived from boundary conduction mode (BCM.) For boost, this value is:

$$0 = I_{L-MIN}$$

$$0 = I_{L-ave} - \frac{\Delta i_L}{2}$$

$$0 = \frac{V_S}{(1-D)^2 R} - \frac{V_S DT}{2L}$$

$$L_{crit} = \frac{D(1-D)^2 R}{2f}$$

In the actual design, we normally pick the inductance to be larger than the calculated L_{crit} .

To find the new capacitance for the input and output capacitors it is important to remember one of the benefits from a two phase design, frequency multiplications.

Frequency multiplication increases the frequency of the noises, making them easier to filter out with smaller capacitance. With two phases the operating frequency of the input

and output of the converter would be doubled to 1MHz and it will affect the charge needed for the capacitors. The equation to find the capacitance is:

$$C = q * \Delta V$$

C is the required capacitance; q is the required charge of the capacitors and ΔV is the change of voltage through the capacitor during normal operation. Figure 2-8 and Figure 2-9 depict ideal current waveform for the input and output capacitors, respectively.

Figure 2-8: Ideal Current Waveform for Input Capacitor

Figure 2-9: Ideal Current Waveform for Output Capacitor

T represents the period found in each phase of the converter. With an operating frequency of 500kHz per phase the value of T comes out to be $2\mu\text{s}$. Notice that the current waveform in both figures has a period half of T due to the frequency multiplication, hence the value of q will decrease. This shows that frequency multiplication does help drop the capacitance needed to filter the converter.

To summarize, Figure 2-10 shows the critical waveforms in the proposed two-phase boost converter of Figure 2-2.

Figure 2-10: Frequency Multiplication in Two-Phase Boost

Figure 2-10 exhibits the benefits of proposed two-phase topology. Firstly, as mentioned above, although each individual phase has an operating frequency of 500kHz the frequency multiplication effect causes the output current to have double the frequency. Secondly, the frequency multiplication also doubles the input current frequency. With less change in the current there would be less noise and less filtering required at both the input and output of the proposed two-phase topology.

III. Requirements

The proposed Two-Phase Boost Converter was designed to fit within the given set of parameters in Table 3-1. The parameters were set to achieve the goal of creating a two-phase converter that would have the potential to improve the efficiency of a regular Boost converter. Since many boost converters are being used in renewable energy applications to step up battery output voltage, hence the proposed converter will be designed to have input and output voltages at the level that would be achievable or usable by battery.

In terms of the output power level, since the objective of this thesis is to build the first prototype, then 200 W is chosen such that the prototype could be tested with the actual photovoltaic panels currently available in the power electronics lab. To achieve the efficiency at this power level while minimizing the physical size of the converter, switching frequency of 500 kHz is selected. This frequency seems to be the value that is commonly used in multiphase dc-dc converters. For a two-phase converter, at this frequency the input and output stages of the converter will actually be seeing twice as much (1 MHz) due to frequency multiplication property of multiphase converter. Careful considerations in the design and in selecting the components have to be taken since operating the converter at this frequency may worsen the efficiency. Therefore, efficiency of 90% at full load should be a good target for the converter.

To ensure reliable and good quality output, the proposed two-phase converter should have output voltage peak-to-peak ripple and load regulation of less than 1%. In addition, to account for variation at the input voltage, the proposed converter should be able to maintain line regulation within 1%.

Table 3-1: Design Requirements for Proposed Two-Phase Boost.

Parameters	Requirements
Rated Power	200 Watts
Input Voltage	36 V _{DC}
Output Voltage	48 V _{DC}
Input Current Max	5.56 Amps
Output Current Max	4.167 Amps
Operating Frequency	500 KHz
Line Regulation	< 1%
Load Regulation	< 1%
Output Voltage Ripple	< 1% V _{P-P}
Efficiency	> 90%

IV. Design Calculations

This chapter explains the design calculations for selecting the components used in the proposed two-phase boost converter.

4.1 Duty Cycle

Recall that duty cycle is the percentage of a switching period when the main switch of the converter in each phase is conducting. Previously it was derived that the duty cycle for both single and two phase Boost can be found by using the same formula given the input and output voltage. Looking at chapter III it can be seen that this converter needs to have input voltage of 36 volts and output voltage of 48 volts. So plugging in the values into the equation gives an ideal duty cycle of 25%.

$$V_o = \frac{1}{(1-D)} V_s$$

$$48 = \frac{1}{(1-D)} * 36$$

$$\frac{4}{3} * (1-D) = 1$$

$$1-D = \frac{3}{4}$$

$$D = \frac{1}{4} = 25\%$$

4.2 Inductor

In boost topologies the inductor is a crucial energy storing element, thus it is very important that the inductor is sized correctly. To determine the minimum inductance needed for the given design requirements, we would first need to calculate the maximum load resistance. Using Ohm's law it can be seen that maximum load resistance would be achieved if the output current is at its minimum. For this design the minimum current has been specified as 5% of the maximum output current. Assuming maximum output power is during a perfect conversion, and then the three values can be calculated as follows.

$$\begin{aligned} I_{O-Max} &= \frac{P_{Out}}{V_{Out}} = \frac{200W}{48V} = 4.167A \\ I_{O-Min} &= I_{O-Max} * 5\% = 0.20835A \\ R_{Load-Max} &= \frac{V_{Out}}{I_{O-Min}} = \frac{48V}{0.20835A} = 230.38\Omega \end{aligned}$$

However, in the proposed two-phase converter, the output power is divided equally into the two phases. Hence, $R_{load-max}$ for each phase:

$$\begin{aligned} R_{Max-phase} &= \frac{R_{Load-max}}{2} \\ R_{Max-phase} &= \frac{230.38\Omega}{2} = 115.19\Omega \end{aligned}$$

The minimum inductance can then be found using:

$$\begin{aligned} L_{Min} &= \frac{[D * (1-D)^2 * R_{max-phase}]}{2f} \\ L_{Min} &= \frac{[0.25 * (1-0.25)^2 * 115.19]}{2 * 500000} = 16.2\mu H \end{aligned}$$

Taking into account losses, how heat and high current lower inductance, the inductors being used in this design were chosen to have slightly higher inductance than $22\mu H$. The maximum current limit of the inductor is also an important factor so as not to saturate it

at maximum output power. The absolute maximum current that could go through the inductor in each phase is equal to half of the maximum input current:

$$I_{in-ave-max} = \frac{P_{in}}{V_s} = \frac{200W}{36V} = 5.56A$$

$$I_{Lpk} = \frac{I_{in-ave-max}}{2} + \frac{\Delta I_L}{2} = \frac{I_{in-ave-max}}{2} + \frac{\left(\frac{V_s}{L} * DT\right)}{2}$$

$$I_{Lpk} = \frac{5.56}{2} + \frac{\left(\frac{36V}{22\mu H} * 0.25 * \frac{1}{500000Hz}\right)}{2} = 3.19A$$

Copper loss of each inductor can be calculated from the equation below when an actual inductor has been acquired. This equation is taken from the LT3782A datasheet [4]:

$$P_{Winding} = \left(\frac{\frac{I_{O(Max)}}{2}}{1 - D_{Max}} \right)^2 * R_w$$

Another important aspect for selecting the inductor is the self-resonant frequency of the inductor. Each phase will operate with a frequency of 500 kHz, so the inductor needs to have a self-resonant frequency much larger than 500 kHz. This is because as the frequency increases, the parasitic capacitance of the inductor will increase and start to cancel out the inductance. If the inductor has a lower or nearly equal self-resonant frequency compared to the operating frequency, the inductor will act as a short and will no longer act as an energy storage element.

4.3 Input Capacitor

The input capacitor is placed to help filter out any possible noise from the DC input source. This component is not as crucial as the output capacitor because in Boost topologies, the inductor is placed in series with the source. The inductor would force the input current to be a continuous and triangle shaped waveform. However, it is still important to find the maximum possible rms current going through the input capacitor to ensure that the actual capacitors chosen will be able to handle loss due to ESR. For the two-phase converter each phase will operate at 500 kHz, but the capacitor will actually see twice the value. Figure 4-1 below is given in the LT3782A datasheet to find the rms ripple for the input capacitor current which comes out to be around 0.175A.

Figure 4-1: RMS Ripple Input Current Rating [7]

Figure 4-2 depicts the input capacitor current where the value of I_{C-peak} is equal to $I_{L-peak} - I_{L-ave}$ and with frequency multiplication the period of the waveform is $T/2$ or $1\mu s$. With q

being the area under the triangle and the input voltage ripple is 0.48V, the input capacitance can be calculated as:

$$C_{in} = q * \Delta V$$

$$C_{in} = 0.205 \mu C * 0.48V = 0.09 \mu F$$

Figure 4-2: Input Capacitor Current

Two different capacitors were chosen as the input capacitors: 47 μF capacitors and 0.22 μF capacitors. The 47 μF capacitors are electrolytic and are mainly being used as a storage element for the converter. Meanwhile, the 0.22 μF capacitors are ceramic and have low ESR, therefore they are placed in parallel with the 47 μF capacitors to reduce the equivalent series resistance of the input capacitors. These input capacitors see an average of 36V input voltage, so capacitors with voltage rating higher than 36V are needed for the hardware design.

4.4 Output Capacitor

To limit the peak to peak ripple of the output voltage, capacitor filter is needed at the output. Figure 4-4 is the unfiltered waveform of the output capacitor current waveform. The capacitance of the output filter is function of the duty cycle, frequency

and minimum load resistance during maximum load. Plugging in the values into the output capacitance equation gives:

$$C_{out} = \frac{D}{\left(\frac{\Delta V}{V_o}\right) * R_{min} * 2f}$$

$$C_{out} = \frac{0.25}{(1\%) * 11.52\Omega * 2 * 500kHz} = 2.17\mu F$$

Similar to the input current, it is important to find the rms current for the output capacitor.

According to Figure 4-3 its rms current should be rated around 0.3A.

Figure 4-3: RMS Ripple Output Current Rating [7]

Figure 4-4: Output Capacitor Current

Rounding up the calculated value, the output capacitor is chosen to be low ESR, 2.2 μ F capacitor. As with the input capacitor, 47 μ F capacitors are also placed in parallel with these low ESR capacitors. Thus, the larger capacitance will perform as storage elements to help supply energy to load and to decrease the output voltage ripple. The low ESR capacitors will reduce the total equivalent resistance of the filter and will serve as a good high frequency filter. The output capacitors will continuously see a peak value of 48V plus $\frac{1}{2}$ of the ΔV_o across them during normal operation. So the chosen capacitors need to be able to handle above 48V.

4.5 Power MOSFET

The power MOSFET is another main component in the boost converter. Several important parameters when choosing a power MOSFET are the drain-to-source breakdown voltage (BV_{DSS}), threshold voltage ($V_{GS(TH)}$), on resistance (R_{DS-ON}), gate-to-

source charge (Q_{GS}), gate-to-drain charges (Q_{GD}), maximum drain current (I_{D-Max}) and the thermal resistances ($R_{TH(JC)}$ and $R_{TH(JA)}$).

Choosing a MOSFET can be started by calculating the approximate power dissipated by the MOSFET. In a two-phase boost that value can be calculated from the equation given in the LT3782A datasheet below:

$$P_{FET} = \frac{\left(\frac{I_{O-Max}}{2}\right)^2}{(1-D)} * R_{DS-ON} * D * \rho_T + k * V_O^2 * \frac{\left(\frac{I_{O-MAX}}{2}\right)}{(1-D)} * C_{RSS} * f$$

The power loss is dependent on I_{O-Max} , duty cycle, R_{DS-ON} , V_O , C_{RSS} , frequency and ρ_T . C_{RSS} is the reverse transfer capacitance of the MOSFET and ρ_T is the normalized drain to source on resistance. It can be considered that the on I_{O-Max} , duty cycle, R_{DS-ON} , V_O , C_{RSS} and frequency are constant for this calculation. So the power loss value will change in accordance to ρ which is the normalized drain-to-source on resistance. The value of ρ is dependent on the temperature of the MOSFET and the relationship between the two can be found in the datasheet unique to each MOSFET.

The main MOSFET will be conducting equal to the duty cycle which is 25% of the time. With the short conducting time, a MOSFET with small switching loss is preferred. This means that the MOSFET should have low output capacitance. After some research FDD3682 power MOSFET was chosen to be the main MOSFET. It has drain-to-source voltage of 100 VDC, continuous drain current of 32Amps and drain-to-source resistance of 36m Ω . For this calculation let's assume the MOSFET is warm from continuous operation and has a temperature of 120°C. Looking at Figure 8, 120°C corresponds to ρ equals to about 1.75.

Figure 4-5: ρ vs. Junction Temperature for FDD3682 [4]

Assuming that the variable k is an empirical constant of 1.7 and C_{RSS} given in the datasheet to be 45pF, the power loss from the main MOSFET can then be estimated as:

$$P_{FET} = \frac{\left(\frac{I_{O-Max}}{2}\right)^2}{(1-D)} * R_{DS-ON} * D * \rho_T + k * V_o^2 * \frac{\left(\frac{I_{O-MAX}}{2}\right)}{(1-D)} * C_{RSS} * f$$

$$P_{FET} = \frac{\left(\frac{4.167A}{2}\right)^2}{(1-0.25)} * 36m\Omega * 0.25 * 1.75 + 1.7 * (48V)^2 * \frac{\left(\frac{4.167A}{2}\right)}{(1-0.25)} * 45pF * 500kHz$$

$$P_{FET} = 0.1215 + 0.2448$$

$$P_{FET} = 0.3663Watts$$

With the calculated power dissipation in the MOSFET, the junction temperature can be calculated with:

$$T_J = T_A + P_{FET} * R_{TH(JA)}$$

$$T_J = 25^\circ C + 0.3663Watts * 52^\circ \frac{C}{W}$$

$$T_J = 44.05^\circ C$$

The value of $R_{TH(JA)}$ is unique to the MOSFETs and can be found in their respective datasheet.

Notice that the power MOSFET chosen has have drain-to-source voltage and continuous drain current that exceed the required specifications. This was done for safety and reliability reasons. By having an increased threshold voltage and current requirement, it can be guaranteed that the MOSFET will not overload nor be strained beyond its limit during typical operations of the converter.

4.6 Voltage and Current Sense Resistors

The pulse width modulator is able to maintain a constant output voltage by adjusting the duty cycle of the converter continuously through the operation. To decide whether it needs to increase or decrease the duty cycle, the pulse width modulator needs feedbacks for both the current and voltage during the operation. This type of controller is called the current mode controller. The monitoring is done through two feedback loops: one from the output voltage and the other from the MOSFET current the main MOSFET.

The voltage sensor is placed in parallel with the load with large resistors by means of a voltage divider. The values of these resistors were chosen so that its midpoint voltage V_x is the same voltage as the internal reference voltage of the controller, see Figure 4-6. The large resistances R_{top} and R_{bottom} are needed to prevent significant amount of current being diverted towards the feedback loop. The LT3782A pulse width modulator will try to maintain its reference 2.4Volts input to the feedback pin. Because the required output for the converter is 48Volts, it means the voltage divider has to be designed to step down to V_x to 2.4V. A voltage divider is the perfect choice for this type of sensing because it

has 100% line regulation. This means any fluctuation in the output voltage would be fully reflected to its input hence to the controller.

Figure 4-6: Voltage Divider

The ratio of the two resistors for the voltage divider can be calculated as the follows:

$$V_o = 2.44Volts \left(1 + \frac{R_{Top}}{R_{Bottom}} \right)$$

$$48Volts = 2.44Volts \left(1 + \frac{R_{Top}}{R_{Bottom}} \right)$$

$$19.672 = 1 + \frac{R_{Top}}{R_{Bottom}}$$

$$18.672 = \frac{R_{Top}}{R_{Bottom}}$$

For this case R_{Top} is chosen to be $2.8M\Omega$, which yields.

$$18.672 = \frac{2.8M\Omega}{R_{Bottom}}$$

$$R_{Bottom} = 149.96k\Omega$$

R_{Bottom} is then rounded to $150k\Omega$, both resistors has 1% tolerance. These are ideal choices because both values are common values for resistors. The combined series resistance for

them is 2.95M Ω , which means with 48 Volts it would only draws about 16.3 μ Amps from 4.167Amps, less than 0.00039% of the output current.

With increasing concern to minimize energy usage the need to measure the flow of current through an electronic system becomes a common practice. The current feedback loop is taken, as mentioned above, from the current going through the main MOSFET. To do this a resistor, called current sense resistor, is placed between the source pin of MOSFET and the ground. The value of this resistor can be calculated from the equation given in the LT3782A datasheet where $V_{Sense(Max)}$ into the controller is 63 mV.

$$R \leq V_{Sense(Max)} * \frac{1 - D_{Max}}{1.2 * \frac{I_{O-Max}}{2}}$$

$$R \leq 63mV * \frac{1 - 0.25}{1.2 * \frac{4.167A}{2}}$$

$$R \leq 18.89m\Omega$$

For the Two-phase Boost Converter it uses two 20m Ω resistors in parallel. As recommended by Linear Technology an RC filter is placed in parallel with the resistors before going into the sense pins of the pulse width modulator to clean up the current sense feedback from any noise spikes. The power loss from the sense resistor can be found through the equation given from the LT3782A datasheet:

$$P_{R(sense)} = \left(\frac{\frac{I_{O(Max)}}{2}}{1 - D_{Max}} \right)^2 * R * D_{Max}$$

$$P_{R(sense)} = \left(\frac{4.167A}{1 - 0.25} \right)^2 * 10m\Omega * 0.25 = 0.0193 Watts$$

V. Simulations

LTSpice, a simulator made by Linear Technology, offers a wide range of Linear Technology chips for simulation, including the LT3782A. The inductors, resistors, diodes and capacitors specifications are based on the previously values and chosen from the list of parts provided by LTSpice. Figure 5-1 below is the screenshot of the LTSpice schematic.

Figure 5-1: Two-Phase Boost Schematic Using LTSpice.

The simulation needs to run until the output voltage and current settle down into steady state before any accurate readings can be taken. Starting the simulation from zero state, that is, the input starts at 0V and ramps up to 36V, takes about 20ms to reach steady state. An in depth full load analysis is shown in the sections below, followed by an overview analysis with different level of loads.

5.1 Output Voltage

Figure 5-2 is a snapshot of the output voltage ripple during steady state. It ripples between 48.001Volts and 47.994V with an average value of 47.964V, approximately 99.9% of the required value of 48V. Using the same values, the output voltage ripple is calculated to be 0.007V_{P-P} or 0.015% which is well below the required 1% ripple.

Figure 5-2: Output Voltage Ripple

Secondly it is important to check the output voltage frequency. The required operating frequency of each phase is 500kHz, thus creating an expected output frequency of 1MHz due to frequency multiplication effect of the Two-Phase topology. Looking at Figure 5-2, a full period happens approximately between 19.242075ms and 19.241075ms.

$$Freq_{Output} = \frac{1}{19.242075ms - 19.241075ms} = 1MHz$$

This is the expected 1MHz output frequency.

5.2 Output Current

The output current for this project can be divided into two different categories, filtered and unfiltered. The unfiltered output currents are recorded by measuring the current through the schottky diodes individually. The resulting plots have distinct characteristics, they are discontinuous and trapezoid shaped as expected, see Figure 5-3.

Using the plot below, the frequency of a single phase is calculated out to be:

$$Freq_{I_{out-unfiltered}} = \frac{1}{19.2225ms - 19.2205ms} = 500kHz$$

Notice that when the plot from both phases are overlaid, phase 1 (green) and 2 (blue), the phase are shifted by 180° . This is because the main MOSFET switched alternately and evenly between the two phases, creating the phase shift.

Figure 5-3: Overlaid Unfiltered Output Current of Phase 1 and Phase 2.

Figure 5-4 shows the filtered output current seen during full load. The filtered waveform is once again continuous and possesses both AC and DC component, similar to the input current. The required output current level for this converter is specified to be

4.167A. Taking the average of the output current waveform gives the value of 3.997A or 95.98% of the expected output current. The output current ripple is 0.59mA, or 0.014%, well below the required 1% ripple. As for the operating frequency, just like the output voltage, it can be hypothesized that the output current should operate at a higher frequency. Because both of the phases are connected to the same load it forces the filtered output current to have a frequency of 1MHz. With both phases operating close to 500kHz, the filtered output should operate at a frequency close to 1MHz. Using the period of the graph in Figure 5-4 the frequency comes out to be:

$$Freq_{I_{out-Filtered}} = \frac{1}{19.8414ms - 19.8404ms} = 1MHz$$

This is the expected value of 1MHz and hence proves the frequency multiplication effect of the two-phase boost converter.

Figure 5-4: Filtered Output Current Ripple

5.3 Switching Signals

Switching signals are the signals sent by the Pulse width modulator to drive the MOSFETs. According to the specification, the pulse width modulator is able to supply 10V of gate drive at a maximum operating frequency of 500kHz per phase. The duty cycle, or the “on time”, for this simulation is calculated to be 25%. By analyzing the switching signals in Figure 5-5 we can see that the full cycle happens between 6.5842ms and 6.5862ms with a period of 2μs and a calculated frequency of:

$$Freq_{DutyCycle} = \frac{1}{6.5862ms - 6.5842ms} = 500kHz$$

The signal has 11V gate drive, slightly over the specification, and lasted for 0.6μs for a duty cycle of 28.6%. The higher duty cycle than expected value should make sense since the circuit use real model of the components which introduce losses.

Figure 5-5: Phase 1 Switching Signals

The duty cycle from the second phase is shown in Figure 5-6. A full cycle lasts between 6.0242ms and 6.0262ms with a period of 2μs and an operating frequency of:

$$Freq_{DutyCycle2} = \frac{1}{6.0262ms - 6.0242ms} = 500kHz$$

Similar to the first phase, the gate drive operates at 11V and lasts approximately for 0.6μs for a duty cycle of 30%.

Figure 5-6: Phase 2 Switching Signals

A two phase converter is required to have a staggered switching signal. In other words, in one period there would be two turn on signals, one from each phase, spaced evenly by half of a period as seen in Figure 5-7. The overlaid figure is a great graphical representation of why the output and input frequency is doubled that of the phase operating frequency. Both output and input are connected to phases 1 and 2, so they “see” the overlaid switching signal instead of a single phase signal, causing them to have an operating frequency of 1MHz. This can be proven by finding the period of the wave while superimposed. The new waveform finished a complete cycle between 6.5786ms and 6.5796ms. Calculating the frequency using these values gives:

$$Freq_{NewDutyCycle} = \frac{1}{6.5796ms - 6.5786ms} = 1MHz$$

Figure 5-7: Superimposed Duty Cycles

5.4 Inductor Current

For a Two-Phase Boost Converter, the inductor current in each phase was expected to be half of its single phase counterpart with 180° phase shift between the two phases. This hypothesis was drawn from the fact that the two phases are in parallel, dividing the current evenly, and it employs staggered switching between the two phases as discussed in section 5.3.

Figure 5-8: Inductor Current at Phase 1

Figure 5-8 shows that the inductor current of phase 1 swings between 3.2A and 2.35A or 0.85A_{P-P}. These average out to 2.775A which is about half of the input current at 5.56A with a ripple of 30.63% and slightly less than the calculated 1.11A or 40% ripple. The frequency of the signal can be calculated using the time it took to finish one period.

$$Freq_{Inductor} = \frac{1}{19.4265ms - 19.4245ms} = 500kHz$$

For comparison, Figure 5-9 shows the inductor current from the second phase of the converter. The second phase inductor current swings between 3.18Amps and 2.33Amps, which average out to about 2.755Amps, a mere 20mA less than the first phase's inductor current. The current ripple is 0.85A_{P-P} or 30.85% which is still less than the expected 40%. The frequency can similarly be found using the period of the plot, for example the full cycle happens between 19.221ms and 19.223ms gave:

$$Freq_{L2} = \frac{1}{19.223ms - 19.221ms} = 500kHz$$

Figure 5-9: Inductor Current at Phase 2.

One of the easiest ways to see the phase shift between the inductor currents is to overlay the output plot on the same axis. Figure 5-10 has both phase 1 (green) and 2 (blue) inductor current and observing the x-axis it can be seen that one of the phase is shifted by about 1ms. So with a period of 2ms the 1ms shift, half of a period, introduces the 180° phase shift.

Figure 5-10: Overlay of Phase 1 and Phase 2 Inductor Current

To be thorough it is important to check that the converter operates in continuous conduction mode through out the load range. The most susceptible condition for discontinuous conduction mode is during low load or 5% load. At low load there is less current being drawn by the load so the average current through the inductors will drop. With the same current ripple in the inductor, if the average current drops there is a chance that the converter will enter discontinuous conduction mode. Figure 5-11 shows that the converter is actually operating in discontinuous conduction mode because the current barely drops below 0A. This problem can be solved by increasing the inductance of the design.

Figure 5-11: Low Load Inductor Currents

5.5 Converter's Efficiency

One of the main focuses in power electronics is efficiency. Board layout, temperature and component tolerances all affect the efficiency of a DC – DC converter. In the simulation those losses cannot be simulated. So it is apparent that the power

efficiency calculations done based on the simulation result will not be accurate, yet it gives a broad sense of the efficiency of the design.

The input power in Figure 5-12 is obtained by multiplying the average input current with the average input voltage. The input power swung between -209W and -190.5W, the negative indicates that the source was supplying power. According to the simulation the average power supplied by the input is 197.62W, slightly off from the 200W specified in the requirement table.

Figure 5-12: Input Power Ripple

To find the output power the same technique was used, multiply the average output current with the average output voltage. Figure 5-13 of the output power ripple shows that the output power rippled between 191.92W and 191.975W. The simulation estimated the average to be around 191.71W. In the case of output power, the graph shows that it existed in the positive Y-axis, meaning that 191.95W was being consumed by the load.

Figure 5-13: Output Power Ripple

Using the average power of both the output and input power, we can now calculate the simulated efficiency of the converter.

$$Efficiency = \frac{P_{out}}{P_{in}} * 100\% = \frac{191.71W}{197.62W} * 100\%$$

$$Efficiency = 97.01\%$$

This efficiency is comparable to the simulated efficiency done by Linear Technology for a slightly different 200W Two-Phase Boost Converter. Linear Technology's design of two-phase boost with LT3782A states that it can reach 98% efficiency. However, it is not specified whether the efficiency is calculated during a full load or no load simulation.

Table 5-1: Simulated Efficiency from 10% - 100% Load

Load [%]	R_{Load} [Ω]	Vout [V]	Iout [A]	Vin [V]	Iin [A]	Pout [W]	Pin [W]	Efficiency [%]
10%	120	47.985	0.399	36	0.5563	19.188	21.836	95.81
20%	60	47.98	0.7997	36	1.1546	38.369	41.564	92.31
30%	40	47.98	1.1995	36	1.6908	57.551	60.87	94.55
40%	30	47.985	1.5995	36	2.2589	76.752	81.319	94.38
50%	24	47.974	1.9989	36	2.7876	95.896	100.35	95.56
60%	20	47.976	2.3988	36	3.3185	115.09	119.47	96.33
70%	17.14	47.974	2.799	36	3.8704	134.28	139.33	96.38
80%	15	47.971	3.1981	36	4.417	153.41	158.93	96.53
90%	13.3	47.975	3.599	36	4.9597	172.66	178.55	96.70
100%	12	47.964	3.997	36	5.4894	191.71	197.62	97.01

Figure 5-14: Efficiency vs. Load from Simulation

5.6 Load Regulation

Load regulation refers to the ability of the converter to maintain the required output voltage with changing loads. Using Table 5-1 above the load regulation can be calculated using:

$$\text{Load Regulation} = \frac{V_{O-\text{Lowload}} - V_{O-\text{Fullload}}}{V_{\text{Lowload}}} * 100\%$$
$$\text{Load Regulation} = \frac{47.985V - 47.964V}{47.985V} * 100\% = 0.052\%$$

Table 5-2: Simulated Load Regulation

V_{Lowload} [V]	V_{Fullload} [V]	Load Regulation [%]
47.99	47.964	0.052%

5.7 Line Regulation

Line regulation is the converter's ability to maintain the required output voltage when the input voltage fluctuates. The highest input this converter is required to handle is 38V and the lowest input required is 34V. Table 5-3 holds the data gathered during full load with input voltage of 34V, 36V and 38V. Using these data the line regulation for the simulation can be calculated to be:

$$\text{Line Regulation} = \frac{V_{O-\text{highestinput}} - V_{O-\text{lowestinput}}}{V_{O-\text{Nominal}}} * 100\%$$
$$\text{Line Regulation} = \frac{47.973V - 47.965V}{48V} * 100\% = 0.0167\%$$

Table 5-3: Simulated Line Regulation

$V_{O-\text{Lowestinput}}$ [V]	$V_{O-\text{Highestinput}}$ [V]	V_{Nominal} [V]	Line Regulation [%]
47.965	47.973	48	0.0167%

VI. Hardware Design and Results

6.1 Hardware Components

When creating the first prototype for the design, it was originally thought that it could have been done on a breadboard or protoboard to save money. The protoboard was the first choice because it was the cheaper of the two. After purchasing and designing the circuit for the protoboard, it was then realized that the protoboard could not easily be adapted to the small controllers. Protoboard is also a poor choice to design a high current controller, because it could not safely isolate and carry such large current. This also rules out using breadboard for the prototype for the same two reasons. The first prototype ends up being done on a double sided PCB board. It costs considerably more to prototype with a manufactured PCB, but it is a much safer and reliable choice.

The first prototype design ends up to be a reliable and correct design that only little tweaks were made to create the final board design. Compared to the first board the final design in Figure 6-1 has closer, thicker traces and larger ground plane. The large metal plate in Figure 6-1 is the ground trace for the converter. It is made specifically so that the ground traces only exist on the top of the board and close to the controller to provide good grounding and thermal conductivity. On the bottom layer there is only a few inches of ground trace to connect the remaining parts that does not fit on the top layer. The main focus of the PCB design is to separate the control nodes and switching nodes as far away as possible to reduce noise and ripple. On the top layer is where the pulse width modulator can be found. All of the control nodes are also found to the left and on the top layer of the board. To place the switching nodes as far away as possible it is designed so they are found to the right and on the bottom layer of the board.

Figure 6-1: Top Layer of the Final PCB Design

Figure 6-2: Bottom Layer of the Final PCB Design

6.1.1 Pulse Width Modulator

A starting point for this design lies in the pulse width modulator controller, also known as PWM. This project will require one PWM controller which should be able to control the main switches with correct timing. Prominent companies such as Texas Instrument, Maxim and Linear Technology have various selections of efficient two phase controllers. After researches, a Linear Technology DC/DC controller chip, LT3782A, is chosen for the two phase boost design. It is an efficient two phase controller that also accommodates two high side drivers for synchronous rectification if need be.

The LT3782A comes in FE 28-leads plastic TSSOP packaging with the 29th pin on the bottom to provide good thermal conduction. LT3782A has an input range between 6 volts to 40 volts which includes the desired input of 36 volts. The range of programmable switching frequency of the controller goes up to 500 kHz which is the required speed. With the 10 volts and high current gate drive this controller could handle most of the industrial MOSFETs, which helps widen the component options for the power MOSFETs. LT3782A has other safety features for the circuit such as under voltage lockout, soft-start, current limit, DCL (Duty Cycle Limit) and slope compensation.

To set limit and program the pulse width modulator, the design merely follows the information given in the datasheet. To set the operating frequency, 40k Ω is attached to pin 11 for R_{SET}. The value of 40k Ω is determined by reading Figure 6-3 which was taken out of LT3782A datasheet.

Figure 6-3: LT3782A R_{Freq} vs. Frequency [7]

The soft start capacitance value is calculated from:

$$t = \frac{C * 2.44V}{10\mu A}$$

Where t is the typical time it takes the output to reach the programmed level. The time is set to 2.5ms, so the soft start capacitance comes out to be:

$$C = \frac{t * 10\mu A}{2.44V}$$

$$C = \frac{2.5ms * 10\mu A}{2.44V} = 10.25 pF$$

The soft start capacitor is chosen to be 10pF and it is attached to pin 14. The other resistor values and capacitors are set according to the recommended value stated in the datasheet.

6.1.2 Inductor

Using the calculated values above and taking into accounts the inductor requirements, Murata high current 1422311C inductor is the perfect candidate. It has an inductance of 22 μ H, maximum rms current rating 11 Amps, maximum resistance of 11 $\mu\Omega$ and self-resonance frequency of 9.3 MHz. It has larger inductance than the calculated value to provide cushion against inductance drop due to thermal and frequency during normal operation. The power loss through the inductor winding can be calculated by:

$$P_{Winding} = \left(\frac{\frac{I_{O(Max)}}{2}}{1 - D_{Max}} \right)^2 * R_W$$
$$P_{Winding} = \left(\frac{\frac{4.167A}{2}}{1 - 0.25} \right)^2 * 11\mu\Omega = 84.89\mu Watts$$

6.1.3 MOSFET

The MOSFET chosen for the hardware prototype is a power MOSFET by Fairchild Semiconductor FDP3682. The FDP3682 model has exactly the same parameters and rated power dissipation as FDD3682, analyzed in section 4.5. It has maximum V_{DS} of 100V and maximum forward current rating of 32A. They merely differ in the packaging; FDP3682 has TO-220 packaging whereas FDD3682 has TO-252 packaging. FDP3682 is chosen for the final hardware because it offers better heat dissipation compared to its surface mount counterpart. TO-220 has an option that allows a heat sink to be attached to the back of the MOSFET.

6.1.4 Schottky Diode

To increase efficiency a schottky diode is used for output stage of each phase. With a fast diode more power could be transferred to the output from the converter during the discharging time of the inductor. SS210 Multicomp schottky diode is an ideal choice with its maximum forward current of 2A with maximum reverse voltage of 100V. The power loss of each diode can be approximated through:

$$P_{Diode} = \frac{I_{O-max}}{2} * V_D$$
$$P_{Diode} = \frac{4.167A}{2} * 0.85V = 1.77Watts$$

6.1.5 Input and Output Capacitors

There are two types of capacitors being used in both the input and output filter, ceramic and electrolytic capacitors. Ceramic capacitors are known to have low ESR, so they are ideal for filtering high frequency noises. The electrolytic capacitors, on the other hand, have higher ESR value and capacitance, which are ideal for energy storage. To gain the benefit of both the ceramic and electrolytic capacitors, they are placed in parallel. This increases the total capacitance, granting the filter a higher energy storage capacity to decrease the input and output voltage ripple. At the same time the total ESR of the filter decreases, decreasing total power loss from the filter.

For safety purposes and reliability, the current and voltage ratings on the chosen capacitors are about double of the values found in Section 4.3 and 4.4. The input capacitors have maximum voltage rating of at least 63V. The output capacitors have maximum voltage rating of at least 100V.

6.2 Final Hardware Design

In the final hardware design, two SS210 Schottky diodes are used in parallel for each phase. The objectives are two-folds. First at full load each phase carries a maximum current around 2.1A, larger than the rated maximum forward current of the SS210 Schottky diodes. This, by having two diodes in parallel in each phase, the amount of current is now down to 1.05A per diode. Secondly, the decrease in current per diode also means reduced power loss, hence increasing the overall efficiency of the converter. The power loss in each of the diode is now half of that calculated in section 6.1.4. The new power lost in each diode would approximately be 0.885W.

The schematic and layout designs were done using Express PCB Schematic and Express PCB Layout. Both programs are free software developed and distributed by Express PCB. Both the schematic and layout program are linked together to simplify the board design process. First the schematic is drawn according to the specification of the converter. Next, Express PCB Layout allows the users to link the footprints for each part in the schematic to their footprint database. It also has the flexibility to allow the users to create custom footprints for any irregular parts. By linking the schematic to the layout, the programs guide the user during the layout connection process, minimizing the chance of mistake in the final PCB board layout.

To reduce noise and feedback interference the control node and the switching node of the converter needs to be separated. To isolate the switching node from the control node the final design of the board is a double sided PCB as shown in Figure 6-1 and 6-2. The top layer holds the control node and input signals whereas the bottom layer holds the switching node and the output signals.

6.3 Hardware Results and Analysis

During the hardware test, it is decided that the pulse width modulator's input should be separated from the input power. With this, it is possible to monitor if the chip has been damaged during the test. If the chip does burn out, the voltage drawn by the chip will drop and it will instead draw a large current from the power supply. If the power supplies are turned off as soon as any anomalies are observed, it helps prevent any damage from being done to the whole converter. This also works vice versa for the chip, were there any disturbance to the converter's input it helps to protect the costly pulse width modulator. Another benefit from this technique is the ability to monitor the power dissipated from the chip throughout the test.

The plots and calculations analyzed are all done during full load test. During a full load test the converter is operating at its maximum rated current of 4.2A and rated input voltage of 36V. A nearly constant 200W of power flows through the board, creating a great condition to analyze the design's capabilities and limits. Through out this section, phase 1 and 2 are merely arbitrary notations to distinguish data gathered from the two phases. For the sake of consistency and accuracy, phase 1 and 2 will always represent the same path through out the analysis.

6.3.1 Output Voltage Ripple

Figure 6-4 displays the output voltage ripple during a full load test. The ripple height is about 2.5 divisions with 1V/division setting, resulting in 2.68V peak-to-peak ripple measured using the oscilloscope. With an output voltage rated at 42.545V during full load, from Table 6-1, the output voltage ripple could be calculated to be 6.3%

Compared to the required 1% output voltage ripple this converter performs outside the parameter.

Figure 6-4: Output Voltage Ripple

It is apparent that the output voltage level is well below the required parameter of 48V.

Discussions and troubleshooting techniques for this issue will be covered in Section 6.4.

6.3.2 Switching Signal

To drive the power MOSFETs the pulse width modulator sends out two switching signals, one for each phase. Using the LT3782A datasheet, the switching signal has been programmed to operate at 500KHz per phase at 25% duty cycle. The switching signal can be observed by placing the scope probe on pin 20 and 23 of the pulse width modulator [7]. Figure 6-5 is taken from pin 23 which drives the MOSFET in the path that has been considered as “phase 1” in this section. The signals heights in the oscilloscope reading are approximately at 2.5 divisions at 5V/division. Multiplying the two values out gives the signal strength to be around 12.5V, enough to drive the 10V gate drive required by

FDP3682. Horizontally, the graph is rated at 500ns/division with a full cycle of the switching signal being approximately 4.2 divisions, 2.1ms in length. The signal itself lasted for 1 division which is 500ns long. At 2.1ms period the duty cycle of the first phase is calculated to be 25% and measured to be 23.81% by the oscilloscope. The operating frequency of the first phase can then be calculated as the following:

$$Freq_{phase1} = \frac{1}{Period_{Phase1}}$$

$$Freq_{phase1} = \frac{1}{2.1ms} = 476.2kHz$$

Figure 6-5: Phase 1 Duty Cycle

Switching the scope probe to pin 23 of the pulse width modulator allows the oscilloscope to capture Figure 6-6. Approximating with the noise the signal strength is about 2.25 divisions. The vertical scale of this graph is still at 5V/division resulting in signal strength of 11.25V which is still enough to drive the FDP3682 power MOSFET.

For the signal to complete a full cycle it takes 4 divisions at 500ns/division. Multiplying the two values gives the period of the signal which is 2ms. The signal itself lasts for about 0.75 divisions or 375ns; the duty cycle is calculated to be about 25% and measured to be 18.75% by the oscilloscope. The calculated and measure duty cycle for the second phase differs extremely due to noise in the switching signals. Figure 6-6 and 6-7 show that the duty cycle of the second phase does last for approximately a quarter of each period, but notice that the peak of the signals has a lot of noise. The noise reduces the switching signals height to below 10V which is less than the gate turn on voltage. This will reduce the duty cycle and disturb the output voltage level. The operating frequency of the second phase is calculated to be:

$$Freq_{Phase2} = \frac{1}{Period_{Phase2}}$$

$$Freq_{Phase2} = \frac{1}{2ms} = 500kHz$$

Figure 6-6: Phase 2 Duty Cycle

To increase efficiency, a two-phase pulse width modulator phase shifts one of its signal by 180°. To prove the two switching signals turns on alternately, start by superimposing Figure 6-5 and Figure 6-6 resulting in Figure 6-7. Phase 2 switching is shifted by about 2 divisions from phase 1 and vice versa. This creates 1ms, or half a period shift, which can be translated to be 180° phase shift. The period of the new waveform is about 2.1 divisions at 500ns/division, about 1.05ms. The overlaid signal frequency is the operating frequency seen by the input and output of the converter. The new frequency can be calculated using the new period through the following equation:

$$Freq_{new} = \frac{1}{Period_{new}}$$

$$Freq_{new} = \frac{1}{1.05ms} = 952.4kHz$$

Figure 6-7: Overlaid Duty Cycle

6.3.3 Inductor Current Ripple

One of the requirements for the two-phase boost is that both phases need to operate in continuous conduction mode. To determine the mode of operation for both inductors a current loop is placed between the input plane and each of the inductors. Figure 6-8 is the captured inductor current waveform from the first phase using the current probe. The peak-to-peak ripple of the first phase current is approximately 2 divisions. With the setting of 50mV/division on the oscilloscope and 0.1A/10mV on the current probe, the first phase inductor current has a ripple of 1A. Figure 6-8 also shows that the first phase is operating in continuous conduction mode during full load. The period of a full cycle in phase 1 takes an approximate 2 division lengths at 1μs/division which is 2ms. Using the period, the frequency of the first phase comes out to be:

$$Freq_{phase1-inductor} = \frac{1}{Period_{phase1-inductor}}$$

$$Freq_{phase1-inductor} = \frac{1}{2ms} = 500kHz$$

Figure 6-8: First Phase Inductor Current

Figure 6-9 is the captured waveform of the inductor current in the second phase. The peak-to-peak ripple of this current is noticeable smaller, around 0.5 divisions. This measurement is taken using the same vertical axis settings as the first phase. Thus, using the same conversion rates the ripples come out to be 0.05A. The difference in the current ripple can be traced to the noise found in the switching signal in phase 2 (see Section 6.3.2.) Notice in section 6.3.2 that the duty cycle of the second phase is about 18.75% whereas the ideal duty cycle is 25%. A shorter effective duty cycle, as mentioned in section 2.1, decreases the amount of time the inductor is allowed to store up energy from the input. With a constant voltage drop across the inductor, the shorter duty cycle will drop the amount of current being charged in the inductor. Hence, the smaller inductor current ripples seen in Figure 6-9.

Figure 6-9: Second Phase Inductor Current

The period of the second phase inductor current is slightly larger than phase 1 at 4.2 divisions, 500ns/division scale. With 2.2ms period, the frequency of the second phase inductor current is:

$$Freq_{Phase\,2-Inductor} = \frac{1}{Period_{Phase\,2-Inductor}}$$

$$Freq_{Phase\,2-Inductor} = \frac{1}{2.2ms} = 454.5kHz$$

Like the switching signals, section 6.3.2, when the inductor currents are superimposed it will show the frequency and phase shift seen by the input. It was predicted that because of the staggered switching signals of the pulse width modulator the input and output of the converter will see an operating frequency of 1MHz. Finding the new period in Figure 6-10 gives a period of about 1.05ms, 2.1 divisions at 500ns/division.

$$Freq_{new} = \frac{1}{Period_{new}}$$

$$Freq_{new} = \frac{1}{1.05ms} = 952.4kHz$$

The doubling in frequency also means that the new waveform is a combination of two waveforms where they are shifted by 180° from each other. The requirement also stated that the converter must operate in the continuous conduction mode. In continuous conduction mode the lowest peak of the inductor current ripple is larger than 0A. Figure 6-10 shows that both phases are indeed operating in continuous conduction mode from the triangular shape of the current. Notice that the currents are able to hit the bottom peak without being clipped at 0A, just like the example graph in Figure 2-7.

Figure 6-10: Overlaid Inductor Current

6.3.4 Current Sense

Each phase has current sense resistors connected to an RC filter as a current mode control. This control is connected to the source pin of the power MOSFET to allow the pulse width modulator to monitor the current going through the MOSFET during operation. In this design the two $20\text{m}\Omega$ resistors in parallel act as the current sense resistor. Figure 6-11 shows the voltage across the current sense resistors. With 500mV/division scale, the current through the resistors in the first phase starts to swing from about 750mV and -1V and settles around 500mV . The same figure can be seen on the second phase's current sense resistors as seen in Figure 6-12. For the second phase, the voltage through the resistors swings between 500mV and -500mV .

Figure 6-11: Phase 1 Current Sense Waveform

Figure 6-12: Phase 2 Current Sense Waveform

6.3.5 Load Test Results

The first test done on the converter is a no load test. During no load test the output pins are left open with only a multimeter connected to them. No load test exerts the least amount of stress on the converter because it pulls a miniscule amount of current. Hence, during no load test the output current and input current are read as 0A by the multimeter. The output voltage during no load test comes out to be 47.985V. With the unreliable current readings it is impossible to calculate the efficiency of the converter during no load condition.

The load data covers from 10% to full load with 10% steps. Due to the large power rating of the converter, resistor box cannot be used as the load. Instead, electronic load is used to simulate load at the output end of the converter. Table 6-1 shows a compilation of the signals measured and calculated during the load tests. The values of V_{Out} , I_{Out} , V_{In} and I_{in} are measured through the multimeters. Both input and output powers are calculated by multiplying the voltage and current measured through respective ports. The output powers are determined from V_{Out} and I_{Out} from each load steps. To calculate the total input powers, the total sum of the input power to the board and the input power to the chip is considered. The powers supplied to the board are calculated using V_{In} and I_{in} from each load steps. The values of P_{Chip} are calculated by multiplying voltage and current supplied to the pulse width modulator. During no load test the chip pulls 11.9V and a miniscule amount of current which is interpreted as 0A by the multimeter, resulting in an incalculable P_{Chip} . Starting from 10% to full load the pulse width modulator pulls a constant 0.05A and 11.9V. Calculating P_{Chip} for all the load steps gives a constant value of 0.595W being supplied to the pulse width modulator.

Table 6-1: Hardware Efficiency from 10%-100% Load

Load [%]	Vout [V]	Iout [A]	Vin [V]	Iin [A]	Pout [W]	Pchip [W]	Pboard [W]	Pin [W]	Efficiency [%]
5%	43.46	0.2	36	0.249	8.692	0.595	8.97	9.56	90.93
10%	43.03	0.412	35.95	0.524	17.73	0.595	18.84	19.43	91.23
20%	43.05	0.76	35.98	0.934	32.72	0.595	33.61	34.20	95.67
30%	43.119	1.17	36.09	1.421	50.45	0.595	51.28	51.88	97.24
40%	42.707	1.627	35.95	1.975	69.48	0.595	70.99	71.59	97.06
50%	42.665	2.065	36.01	2.511	88.10	0.595	90.42	91.02	96.80
60%	42.592	2.36	35.99	2.852	100.52	0.595	102.66	103.26	97.35
70%	42.612	2.76	36.00	3.341	117.61	0.595	120.28	120.88	97.30
80%	42.622	3.16	36.01	3.822	134.69	0.595	137.63	138.23	97.44
90%	42.63	3.56	36.01	4.312	151.76	0.595	155.28	155.87	97.36
100%	42.545	4.16	36.01	5.053	176.99	0.595	181.96	182.55	96.95

The efficiencies listed in Table 6-1 are calculated using the equation shown in the sample calculation below:

$$Efficiency_{Fullload} = \frac{P_{out}}{P_{In}} * 100\%$$

$$Efficiency_{Fullload} = \frac{181.96}{182.55} * 100\% = 96.95\%$$

Compared to the simulated full load efficiency, the values are almost identical. The hardware efficiency is 0.06% lower than the simulated efficiency, but it is above the required 90% or better efficiency. Compared with those available in the market, single phase boost converter boasts around 95%-97% efficiency. To compare against other two-phase boost, this design is comparable to the Linear Technology's two-phase boost design with 98% efficiency. This design can probably match or be more efficient than the Linear Technology design with synchronous rectification and/or other modification. It is difficult to accurately compare against single and multi phase converters that are designed for lower or much different specification. Figure 6-13 shows that as the load

increases the efficiency of the converter increases along until about 30% load. With a load at 30% or higher the efficiency of the converter maintains an average of 97%.

Figure 6-13: Load vs. Efficiency for Hardware Test

6.3.6 Load Regulation

To calculate the load regulation the value of $V_{O-LowLoad}$, $V_{O-HighLoad}$ and $V_{O-nominal}$ are required. The nominal output voltage is determined to be the voltage at full load and the other two values can be found from Table 6-1. Low load voltage is referring to the output voltage during 5% load. The high load voltage, on the other hand, refers to the full load output voltage which is now equal to the nominal output voltage. Using these three values the load regulation can then be calculated through:

$$Load \ Regulation = \frac{V_{O-Lowload} - V_{O-Fullload}}{V_{O-Lowload}} * 100\%$$

$$Load \ Regulation = \frac{43.46V - 42.545V}{43.46V} * 100\% = 2.11\%$$

Table 6-2: Hardware Load Regulation

$V_{\text{Lowload}} [\text{V}]$	$V_{\text{Fullload}} [\text{V}]$	Load Regulation [%]
43.03	42.545	2.15%

The load regulation is double higher than the required 1% value, but still within an acceptable range.

6.3.7 Line Regulation

To calculate the line regulation the equation requires two extra output voltages, from the lowest input voltage and the highest input voltage. The lowest input to calculate the line regulation for this design is 34V and the highest input is at 38V. With 34V input it yields 40.073V and with 38V input it yields 43.76V. To maintain consistency in the data and calculations, both readings were done with full load.

$$\text{Line Regulation} = \frac{V_{O-\text{highestinput}} - V_{O-\text{lowestinput}}}{V_{O-\text{Nominal}}} * 100\%$$

$$\text{Line Regulation} = \frac{43.76\text{V} - 40.073\text{V}}{42.545\text{V}} * 100\% = 8.67\%$$

Table 6-3: Hardware Line Regulation

$V_{O-\text{LowestInput}} [\text{V}]$	$V_{O-\text{HighestInput}} [\text{V}]$	$V_{\text{nominal}} [\text{V}]$	Line Regulation [%]
40.073	43.76	48	8.67%

The line regulation for the hardware test falls far from the required 1% regulation.

Attempts were made to improve the line regulation and they are discussed in the next section 6.4.

6.3.8 Transient Response

Transient response test check the ability of they proposed converter to handle abrupt and large change in the load. With AC coupling and single trigger setting on the oscilloscope it is able to capture the overshoot and ripple time. Two tests were done to check the transient response of the converter. The first test changes the load from full load to 10% load. Figure 6-14 is the captured waveform of the transient response of the converter. The vertical overshoot is about 6V and it takes 3 division length at 50ms/division or 150ms for the signal to settle. The positive overshoot is caused by the large and sudden current drop in the load.

Figure 6-14: Output Voltage Overshoot from Full Load to Low Load

The second test changes the load from 10% load to full load. Notice that Figure 6-15 has a negative overshoot instead due to the large current being drawn suddenly by the load. It has an overshoot of about 7V and lasts for 125ms before settling into a steady state.

Figure 6-15: Output Voltage Overshoot from Low Load to Full Load

6.4 Hardware Troubleshooting

6.4.1 Output Voltage

Comparing Table 6-1 with Table 5-1 it is obvious that the hardware's output voltages are well below the required level. There were three theories of why this might have happened. One of the theories hypothesized that by decreasing the voltage feedback level it increase the output voltage level. This hypothesis is based by the ability of the pulse width modulator to increase the output voltage when the feedback signal drops below 2.44V. To test this theory, the value of the output voltage divider was dropped to 30k Ω and 700k Ω . During testing it is observed that the no load output voltage increased to 53.12V and the loaded output to an average of 44V. This proves that dropping the voltage feedback ratio does indeed increase the output voltage level, but it does not solve

the large voltage drop between the no load and loaded test. All of the resistors used have a 1% tolerance.

The second theory is that the converter is not operating in continuous conduction mode when loaded. To check validity of this theory a current probe is used to obtain the inductor current through the current test loops. The two crucial points in the load test are the two extreme ends of the load test, 5%, and full load. During full load it was already discussed that the converter is indeed operating in the continuous conduction mode. In the 5% load test the converter is most susceptible to being in discontinuous conduction mode. At this point a small amount of current is being pulled by the load so the average inductor current also drops. If the average inductor current drop low enough, the lowest peak of the inductor current ripple can dip below zero. To check if at 5% load the converter operate in continuous conduction mode a current waveform is taken from each of the current loop. Figure 6-13 shows the first phase inductor current during low load test and Figure 6-16 shows the second phase. Both figures have full triangular ripples without having the bottom peak dipping below 0A meaning phase 1 and 2 are in continuous conduction mode. These two graphs prove that the drop in output voltage is not due to the converter operating in discontinuous or boundary conduction mode.

Figure 6-16: Phase 1 Inductor Current during Low Load

Figure 6-17: Phase 2 Inductor Current during Low Load

Third, the issue might have come from the noisy switching signals. As mentioned before, the noise at the peak of the switching signal drops the signals value below 10V. When this happen the MOSFETs gates are receiving below the required gate turn on voltage. This could cause the second phase MOSFET to not conduct as long as the

controller signaled it to be, reducing the output voltage. However, because this theory has not yet been thoroughly tested no results could be reported regarding this hypothesis.

6.4.2 Line Regulation

The first idea is to change the current sense resistor size from $20\text{m}\Omega$ to $10\text{m}\Omega$. By lowering the current sense resistor it would make the converter more sensitive to any change in the voltage. This modification increased the $V_{O\text{-LowestInput}}$ by a few mV, a barely noticeable change. The second idea is to increase the output capacitance. The idea is that by increasing the amount of stored energy it might help to bump the input power, creating an overall higher output voltage. With the extra capacitance the line regulation still has the same general value. So, this idea does not effect the converter in the way it was planned either.

VII. Conclusion

Efficient, cheap and fast switching converters are in high demand in the high-tech world. With varying needs and specifications, engineers are expected to design unique and reliable solutions. The size, cost and safety of each design is also top priority in today's industrialist economy. The two-phase boost converter discussed in this report was designed and built under the same pretense of the real world expectancy. The converter is designed to be safe, efficient, low cost, and small.

After gathering and comparing the different results from the simulation and hardware, the values from both tests are consistent in general. In both cases, the converter successfully completes the full load tests while operating in continuous conduction mode. The voltage ripple and load regulation from both results are also below or equal to the required 1%. It is also impressive to observe that the hardware efficiency reaches 96.95%, merely 0.06% less than the simulated efficiency. This efficiency can be improved, by following some of the recommended improvements suggested above. The largest difference is found in the line regulation where in simulation it is 0.0167%, but the hardware data reads 7.68%.

Fixing the output voltage so it reaches the designed 48V is a crucial fix that this design needs. More tests, trials and calculations need to be done to find the root of the problem. Some of the things that could be done are resizing some of the energy storage components, e.g. the inductors and capacitors or change in the pulse width modulator settings and sensing components.

The efficiency of this design can be improved further by adding synchronous rectification into the design. In the current design two parallel schottky diodes are

controlling the high side switching in each phase. Synchronous rectification will replace those diodes with MOSFETs. By placing MOSFETs with low R_{DS-on} it will decrease the amount of loss through the high side switch especially at low duty cycle. Controlled MOSFETs will also decrease the dead time between switching to a more efficient length. LT3782A has two outputs that will accommodate synchronous rectification for the high side switching; it just needs to be implemented. A total redesign or some tweaks can be done to the current high side driver design to allow it to interface better with the converter.

Paralleling MOSFETs will also increase the efficiency of the converter. Each MOSFETs has a certain value of R_{DS-on} , and hence by paralleling them it will decrease the total resistance across the switches.

Even with the satisfactory results, the final hardware project does not output the required 48V with <1% line regulation. Many troubleshooting ideas were tested although none solved the problems at hand. Despite this issue, the project successfully proves that a two-phase boost concept can actually be implemented. The challenges this design presented has taught me that design is not all in the numbers and papers. Component placements and board design also play out a major part in creating electrical equipments. This project has also taught me invaluable troubleshooting and design skills that will continue to improve and grow with experience.

BIBLIOGRAPHY

- [1] Alou, Pedro, Jose Cobos, Oscar Garcia, Roberto Prieto, and Javier Uceda. "A New Driving Scheme for Synchronous Rectifiers: Single Winding Self-Driven Synchronous Rectification." *IEEE Transactions on Power Electronics*, 16.6 (2001): 803-811.
- [2] Axelrod, B, Y Berkovich, S Tapuchi, and A Ioinovici. "Single-Stage Single-Switch Switched-Capacitor Buck/Buck-Boost-Type Converter." *IEEE Transactions on Aerospace & Electronic Systems*, 45.2 (2009): 419-430.
- [3] Du, Xiong, Luowei Zhou, and Heng-Ming Tai. "Double-Frequency Buck Converter." *IEEE Transactions on Industrial Electronics*, 56.5 (2009): 1690-1698.
- [4] Fairchild Semiconductor. FDD3682 datasheet.
<http://www.fairchildsemi.com/ds/FD/FDD3682.pdf>
- [5] Florian, Paul. "Buck Converter Powers 5-V Circuits from 12-V Solar-Cell Array." *Electronic Design*, 57.19 (2009): 45-46.
- [6] Garcia, Oscar, Pablo Zumel, Angel de Castro, Pedro Alou, and Jose Cobos. "Current Self-Balance Mechanism in Multiphase Buck Converter." *IEEE Transactions on Power Electronics*, 24.6 (2009): 1600-1606.
- [7] Linear Technology. LT3782A datasheet.
<http://cds.linear.com/docs/Datasheet/3782af.pdf>
- [8] Miura, Hidekazu, Shinsuke Arai, Fumihiro Sato, Hidetoshi Matsuki, and Tadakuni Sato. "A Synchronous Rectification Using a Digital PLL Technique for Contactless Power Supplies." *IEEE Transactions on Magnetics*, 41.10 (2005): 3997-3999.
- [9] Rashid, M. H. Power electronics circuits, devices, and applications. Upper Saddle River, N.J: Pearson/Prentice Hall, 2004.
- [10] Selders, Robert. "Synchronous Rectification in High-Performance Power Converter Design." *EDN*, 51.12 (2006): 1-7.
- [11] Taufik, "Introduction to Power Electronics," Lecture Notes, 6th revision 2008.
- [12] Taufik, "Advance Power Electronics," Lecture Notes, 3rd revision 2009.