Boost Converter Design Project

Brendan Grimes, James O'Connell, Cameron Greene

December, 8th, 2021

ECE 2630
Instructor: Prof. Adam Barnes

Contents

1 Introduction **1**

2 Analytical **1**

2.1 Experimental Inductor Resistance . . . 1

2.2 Determining R_3 and C_2 . . . 1

2.3 Energy Consumed by the Load . . . 3

2.4 Determining C_3 . . . 3

2.5 Inductor Energy Supply Verification . . . 4

3 Numerical Verification **5**

3.1 Multisim Verification . . . 5

3.2 Ultiboard Design . . . 7

4 Experimental Verification **8**

5 Summary and Error Analysis **11**

List of Figures

1 Boost Converter Circuit with Unknown Values . . . 1

2 Multisim Design of the Boost Converter . . . 6

3 555 Timer PWM Signal Output . . . 6

4 Boost Converter Total Voltage Output . . . 7

5 The Boost Converter PCB Design Layout . . . 8

6 The PCB Board with all Components in Place . . . 9

7 The Pulse-Width Modulator Output Set to 1 kHz . . . 9

8 Trace Showing V_{PWM} and V_{cap} . . . 10

9 Trace Showing V_{out} . . . 11

1 Introduction

Our task was to design, build and test a boost converter. A boost converter is a DC to DC converter that boosts the incoming supply voltage to a greater output voltage. The boost converter we are designing uses an inductor that transfers its magnetic energy to a capacitor via high-frequency PWM Square-wave. The output voltage is held constant by the charged capacitor and a diode prevents it discharging back into the inductor. The base circuit is shown below:

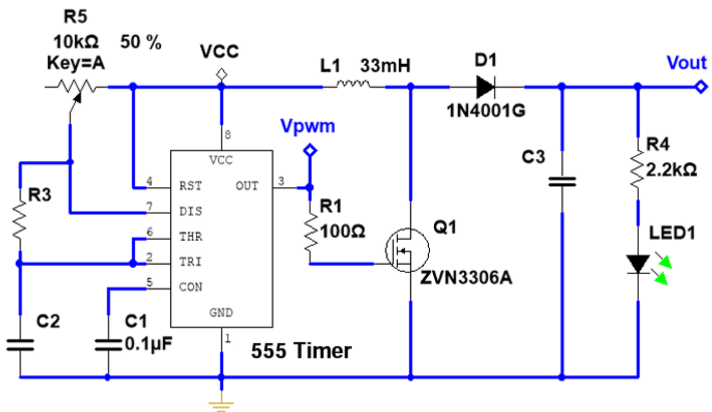

Figure 1: Boost Converter Circuit with Unknown Values

As you can see from the figure there were some values that we needed to calculate. Those values included R3, C2 and C3 the calculations for which appear below.

2 Analytical

2.1 Experimental Inductor Resistance

The experimentally measured value of the inductor's internal resistance was:

$$R_L = 55.9\ \Omega$$

2.2 Determining R_3 and C_2

To determine the value of R_3, we can use the relationships for the duty cycle of a 555 Timer in the Astable Multivibrator configuration along with the requirements that the duty cycle remain between 50% and 80% and that the frequency $f = \frac{1}{T_{on}}$ be on the order of 1 kHz.

For a 555 Timer in the Astable Multivibrator configuration we have:

$$T_{on} = 0.693(R_2 + R_3)C_2$$

$$T_{off} = 0.693R_3C_2$$

$$T_{total} = T_{on} + T_{off}$$

$$\text{Duty Cycle: } D = \frac{T_{on}}{T_{total}} = \frac{R_2 + R_3}{R_2 + 2R_3}$$

We can break the problem down into two cases corresponding to the maximum and minimum resistances of R_2 (the 10k potentiometer).

Lower ($R_2 = 0\ \Omega$):

$$D = \frac{R_2 + R_3}{R_2 + 2R_3}$$

$$0.5 = \frac{(0\ \Omega) + R_3}{(0\ \Omega) + 2R_3}$$

$$0.5 = \frac{R_3}{2R_3} = \frac{1}{2} = 0.5$$

Therefore, because the value of R_3 cancels out at the lower bound, we can safely say that this case implies no restrictions on the value of R_3.

Upper ($R_2 = 10\ \text{k}\Omega$):

$$D = \frac{R_2 + R_3}{R_2 + 2R_3}$$

$$0.8 = \frac{(10\ \text{k}\Omega) + R_3}{(10\ \text{k}\Omega) + 2R_3}$$

$$8\ \text{k}\Omega + 1.6R_3 = 10\ \text{k}\Omega + R_3$$

$$0.6R_3 = 2\ \text{k}\Omega$$

$$R_3 = 3.33\bar{3}\ \text{k}\Omega \approx 3.3\ \text{k}\Omega$$

While $R_3 = 3.3\ \text{k}\Omega$ is a resistor value available in our kit, a choice of $R_3 = 3.9\ \text{k}\Omega$ is more appropriate for our case, which will be demonstrated by the calculations for frequency and C_3. With this value, the upper bound of the duty cycle ($R_2 = 10\ \text{k}\Omega$) is:

$$D = \frac{(10\ \text{k}\Omega) + (3.9\ \text{k}\Omega)}{(10\ \text{k}\Omega) + 2(3.9\ \text{k}\Omega)} = \frac{13.9\ \text{k}\Omega}{17.8\ \text{k}\Omega} = 0.781 = 78.1\% < 80\%$$

The lower bound ($R_2 = 0\ \text{k}\Omega$) is still $D = 50\%$ because the lower bound is independent of the value of R_3.

This choice of $R_3 = 3.9\ \text{k}\Omega$ can be used to find a value for C_2. Letting the frequency $f = 1$ kHz and the potentiometer be set to 50% ($R_2 = 5\ \text{k}\Omega$) to obtain a middle-of-the-road guess for C_2, we have:

$$f = \frac{1}{T_{total}} = \frac{1}{0.693C_2(R_2 + 2R_3)}$$

$$C_2 = \frac{1}{0.693f(R_2 + 2R_3)} = \frac{1}{0.693(1\ \text{kHz})((5\ \text{k}\Omega) + 2(3.9\ \text{k}\Omega))} = 0.112\ \mu\text{F} \approx 0.1\ \mu\text{F}$$

which is a capacitor value at our disposal. Then, if we let $C_2 = 0.1\ \mu\text{F}$, f ranges from:

$$f_{lower} = \frac{1}{0.693C_2(R_2 + 2R_3)} = \frac{1}{0.693(0.1\ \mu\text{F})((10\ \text{k}\Omega) + 2(3.9\ \text{k}\Omega))} = 0.811\ \text{kHz}$$

when $R_2 = 10\ \text{k}\Omega$ to

$$f_{upper} = \frac{1}{0.693C_2(R_2 + 2R_3)} = \frac{1}{0.693(0.1\ \mu\text{F})((0\ \text{k}\Omega) + 2(3.9\ \text{k}\Omega))} = 1.850\ \text{kHz}$$

when $R_2 = 0\ \text{k}\Omega$. This is an acceptable range for the frequency. Thus, we have:

$$R_3 = 3.9\ \text{k}\Omega \qquad C_2 = 0.1\ \mu\text{F}$$

2.3 Energy Consumed by the Load

To find the total Energy consumed by the load, it is useful to first find the current through the load assuming a voltage drop across the load of $V_{load} = 13$ V and a voltage drop across the LED of $V_{LED} = 2$ V:

$$i_{load} = \frac{V_{R_4}}{R_4} = \frac{V_{load} - V_{LED}}{R_4} = \frac{11\ \text{V}}{2.2\ \text{k}\Omega} = 5\ \text{mA}$$

Then, we can determine the power absorbed by the load:

$$P_{LED} = (2\ \text{V})(5\ \text{mA}) = 10\ \text{mW}$$

$$P_{R_4} = (11\ \text{V})(5\ \text{mA}) = 55\ \text{mW}$$

$$P_{load} = P_{LED} + P_{load} = 65\ \text{mW}$$

Having obtained P_{load}, we can divide this number by the frequency to obtain the energy absorbed by the load. Since the frequency ranges from 0.811 kHz to 1.850 kHz, there is once again a range of values for the energy consumption.

Lower ($f = 1.850$ kHz):

$$w_{lower} = \frac{65\ \text{mW}}{1.850\ \text{kHz}} = 35.135\ \mu\text{J}$$

Upper ($f = 0.810$ kHz):

$$w_{upper} = \frac{65\ \text{mW}}{0.810\ \text{kHz}} = 80.247\ \mu\text{J}$$

However, we are primarily concerned with calculating the energy for when the signal is operating at 1 kHz because this is the frequency at which we will be doing our testing. For this case, the energy absorbed by the load is:

$$w_{load} = \frac{65\,\text{mW}}{1\,\text{kHz}} = 65\,\mu\text{J}$$

2.4 Determining C_3

Since we know that the load current $I_{load} = 5$ mA, to find the charge of the capacitor, we can use the relationship:

$$Q_{C_3} = I_{load} \cdot \Delta t$$

Since we know that C_3 must discharge for the entirety of $T_{on} = \Delta t = T_{total} - T_{off} = 1\ \text{ms} - 0.693(R_3)C_2 = 1\ \text{ms} - 0.693(3.9\ \text{k}\Omega)(0.1\ \mu\text{F}) = 729\ \mu\text{s}$, we can substitute the requisite values into our expression for charge to obtain:

$$\Delta q = (5\ \text{mA})(729\ \mu\text{s}) = 3.649\ \mu\text{C}$$

With this value firmly in place, it is a trivial matter to derive the capacitor's size. Because we know that the voltage must not fall below 0.3% of the standard value 13V, $\Delta V = 13(0.003) = 0.039$V

$$C = \frac{\Delta q}{\Delta V} = \frac{(3.649\ \mu\text{C})}{0.039\text{V}} = 93.555\ \mu\text{F}$$

Therefore, a choice of $C_3 = 100\ \mu$F is sufficient.

2.5 Inductor Energy Supply Verification

The energy in the inductor is calculated using the following formula :

$$w(t) = Li^2$$

However, in the case of a boost converter, the current through the inductor is not constant, so this quantity must now involve calculus:

$$w(t) = L\int_0^{T_{on}} i(t)di$$

It now remains to find an expression for the current i(t) in the inductor. This is done by first using KVL to evaluate all currents in the circuit:

$$-V_{cc} + V_L + iR_L + V_{DS} = 0$$

However, the voltage across the inductor is not constant either. It is expressed in terms of the current rate of change below:

$$V_L = L\frac{di}{dt}$$

Finally, the above equations can be rearranged to produce a differential equation that will be solved for i(t):

$$\frac{L}{R_L}\frac{di(t)}{dt} + i(t) = \frac{V_{cc} - V_{DS}}{R_L}$$

This is a first-order linear differential equation. Before solving it, however, all quantities involved must be well defined. The drain-source voltage V_{DS} is derived from the triode region of a MOSFET, shown below:

$$V_{DS} = (V_{GS} - v_t) + \frac{1}{R_D K_n} \pm \sqrt{[(V_{GS} - v_t) + \frac{1}{R_D K_n}]^2 - \frac{2V_{DD}}{R_D K_n}}$$

The solution to the differential equation will comprise a homogeneous solution and a particular solution. First, it is solved for the homogeneous part below.

$$\frac{L}{R_L}\frac{di_c(t)}{dt} + i_c(t) = 0$$

$$\frac{di(t)}{i_c(t)} = -\frac{R_L}{L}dt$$

$$i_c(t) = K_1 e^{-\frac{tR_L}{L}}$$

Next, the process to get the particular solution is shown below. Taking the guess of the solution to be a constant C:

$$\frac{L}{R_L}\frac{di(t)}{dt} + i(t) = \frac{V_{cc} - V_{DS}}{R_L}$$

$$0 + i_p(t) = \frac{V_{cc} - V_{DS}}{R_L} \rightarrow i_p(t) = \frac{V_{cc} - V_{DS}}{R_L}$$

By combining solutions, we have:

$$i(t) = i_c(t) + i_p(t) = K_1 e^{-\frac{tR_L}{L}} + \frac{V_{cc} - V_{DS}}{R_L}$$

Next, a boundary condition must be applied. From the outset, the current through the inductor can not change instantaneously, so it must be zero, that is, i(0) = 0:

$$0 = K_1 + \frac{V_{cc} - V_{DS}}{R_L} \rightarrow K_1 = -\frac{V_{cc} - V_{DS}}{R_L}$$

$$i(t) = \frac{V_{cc} - V_{DS}}{R_L}(1 - e^{-\frac{tR_L}{L}})$$

Finally, the power is determined with a definite integral:

$$w(t) = L\int_0^{T_{on}} i(t)di$$

$$w(t) = L\frac{i(t)^2}{2}$$

$$w(t) = L(\frac{i(T_{on})^2 - i(0)^2}{2})$$

$$w(t) = \frac{L}{2}((\frac{V_{cc} - V_{DS}}{R_L}(1 - e^{-\frac{T_{on}R_L}{L}}))^2 - 0)$$

$$w(t) = \frac{L}{2}((\frac{V_{cc} - ((V_{GS} - v_t) + \frac{1}{R_D K_n} \pm \sqrt{[(V_{GS} - v_t) + \frac{1}{R_D K_n}]^2 - \frac{2V_{DD}}{R_D K_n}})}{R_L}(1 - e^{-\frac{T_{on}R_L}{L}}))^2)$$

By taking the practical values $V_{cc} = 5$V, $V_{GS} = 5$V, $R_D = R_L = 55.9\Omega$, $K_n = 0.1233\frac{\text{A}}{\text{V}^2}$, $v_t = 1.824$V, $V_{DD} = 5$V, $T_{on} = 729\mu$s and $L = 0.033$H, this expression yields the following:

$$w(T_{on}) = 60.51\,\mu\text{J}$$

Note that this is not sufficient to supply the energy needed by the load, w_{load}:

$$(w_{load} = 65\,\mu\text{J}) \geq (w(T_{on}) = 60.51\,\mu\text{J})$$

This will likely result in a slightly lower output voltage than we would like, more specifically:

$$P_{load} = (1\,\text{kHz})(60.51\,\mu\text{J}) = 60.51\,\text{mW}$$

$$V_{out} = \frac{P_{load}}{i_{load}} = \frac{60.51\,\text{mW}}{5\,\text{mA}} = 12.102\,\text{V}$$

3 Numerical Verification

3.1 Multisim Verification

Next, the circuit is constructed and its behavior is analyzed in Multisim. Below is an image of the design:

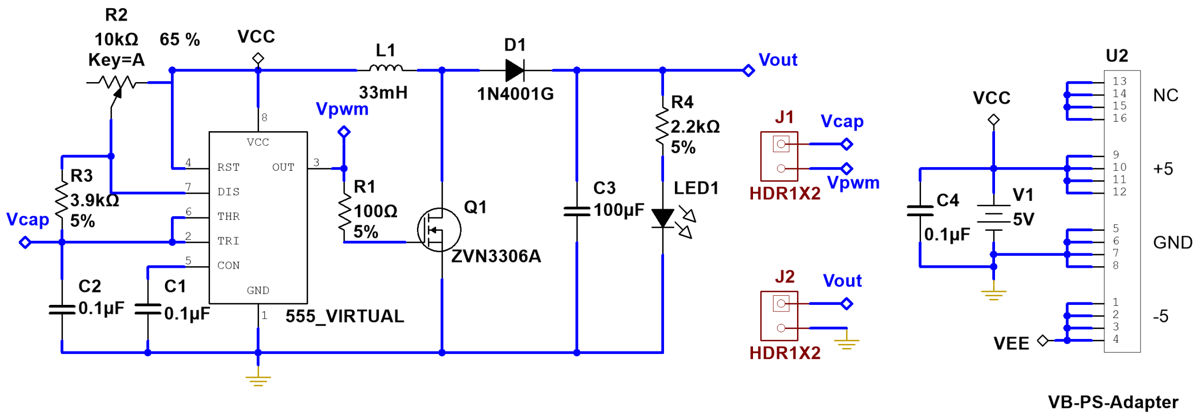

Figure 2: Multisim Design of the Boost Converter

Now, the behavior of the circuit is analyzed. First, the 555 timer output is taken over a few milliseconds to verify the charging procedure:

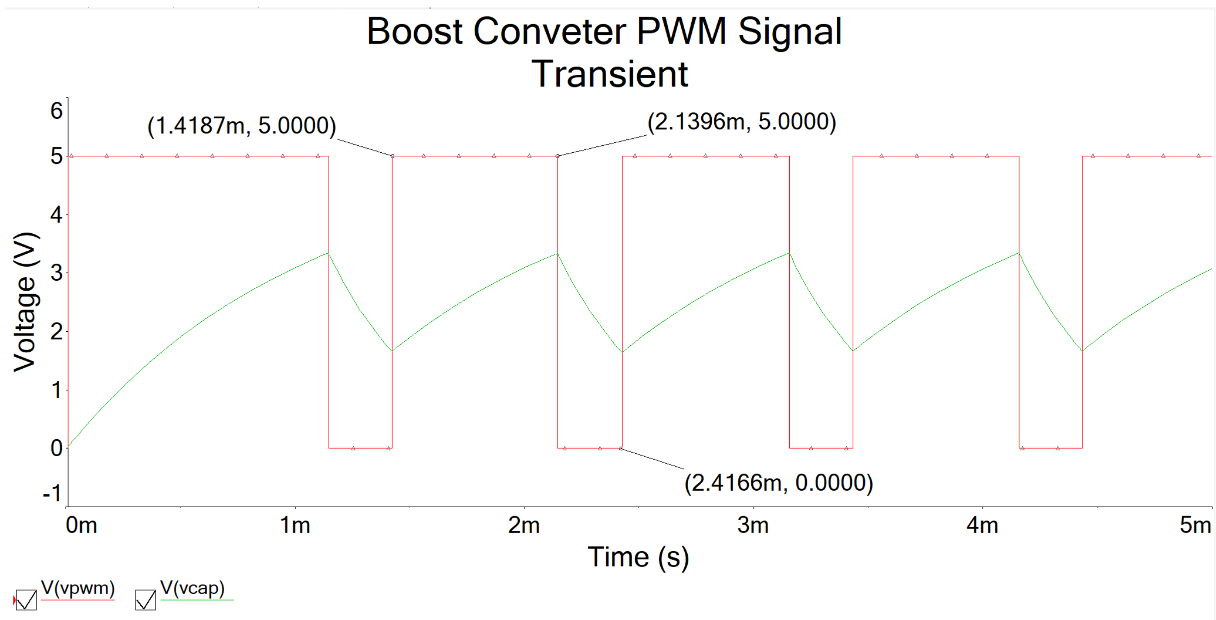

Figure 3: 555 Timer PWM Signal Output

By taking the period of the timer to be 2.4166s-1.4187s = 0.9979s, we verify that the frequency is $\frac{1}{T} = \frac{1}{0.9979\text{s}}$ = 1.002 kHz. This is virtually equivalent to our desired frequency, 1kHz. Moreover, we verify that the duty cycle is D = $(100)\frac{T_{on}}{T_{on}+T_{off}} = (100)\frac{2.1396\text{s}-1.4187\text{s}}{2.4166\text{s}-1.4187\text{s}} = 72.24\%$. This is also well within the constraints of the design.

Finally, the behavior is analyzed over a few seconds to verify the total voltage output:

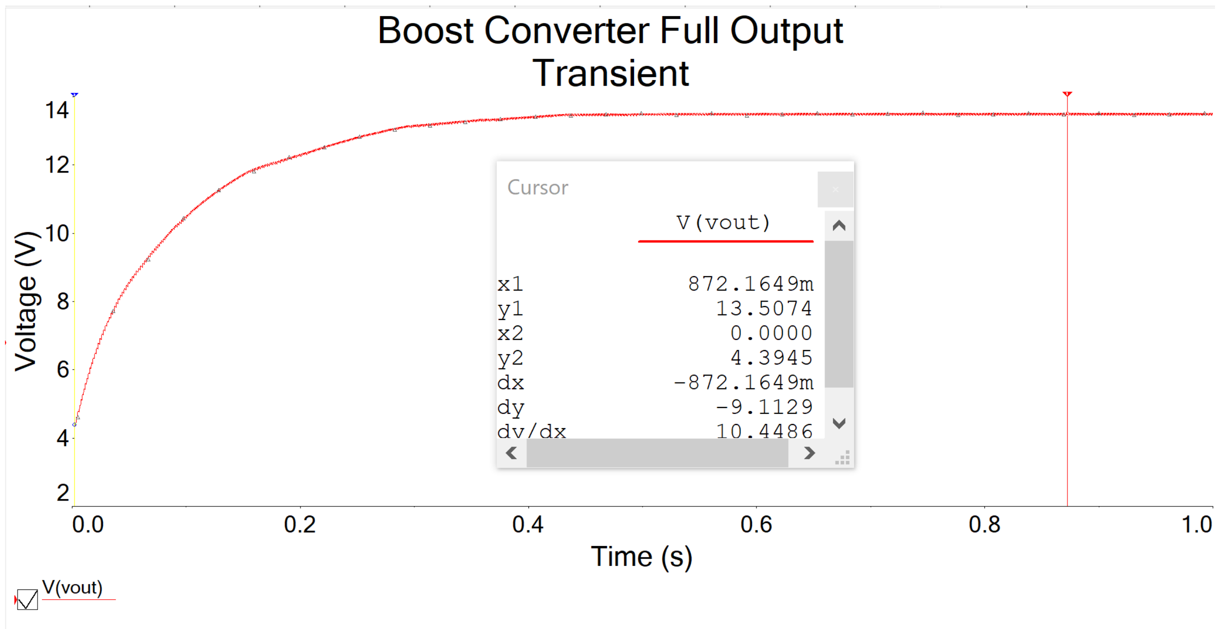

Figure 4: Boost Converter Total Voltage Output

The total voltage output is close to 13.5V, well within the constraints of the design.

3.2 Ultiboard Design

After verfiying that our design was effective using Multisim, we then used Ultiboard from National Instruments to design a PCB board layout for the circuit. An image for this design appears in Figure 5

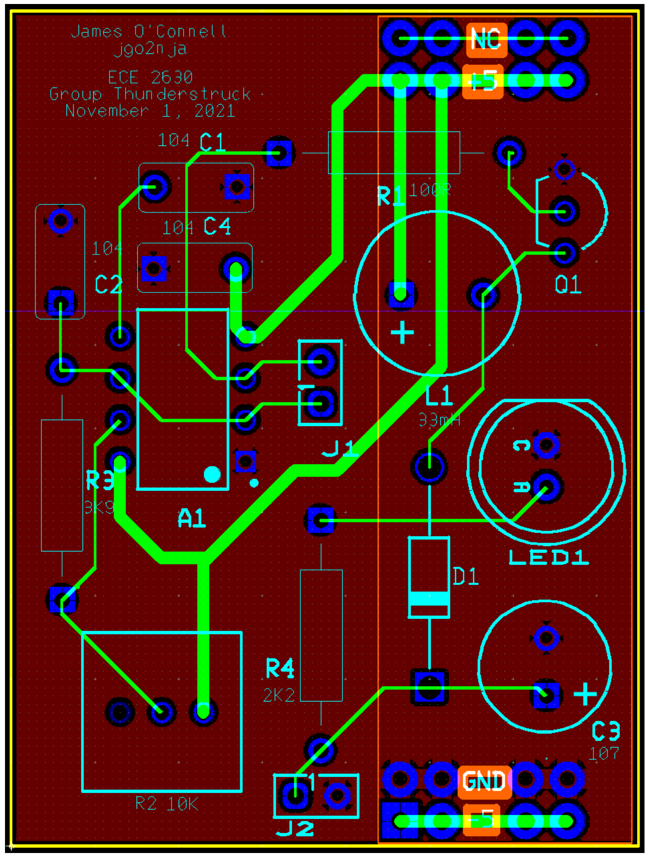

Figure 5: The Boost Converter PCB Design Layout

We subsequently had this PCB board design sent to a manufacturer to obtain a physical version of this with which to experiment.

4 Experimental Verification

After obtaining the physical PCB board manufactured according to our layout from Figure 5, we soldered in the necessary components to form the circuit, using the values specified during analytical design for the capacitors and resistors. An image of the completed PCB board appears in Figure 6.

Figure 6: The PCB Board with all Components in Place

We designed this board to be compatible with our Analog Discovery 2 (AD2) unit such that we could use standoff pins running from the +5 V, -5 V, and GND holes on our PCB to the corresponding rails from the breadboard connected to our AD2. This gave us a convenient power supply system and also provided a stable base on which our Boost Converter circuit board could rest.

Using the AD2, we tested our circuit to verify that it met all design specifications. By tuning the potentiometer R_2, we set the PWM signal to 1 kHz, as shown in Figure 7.

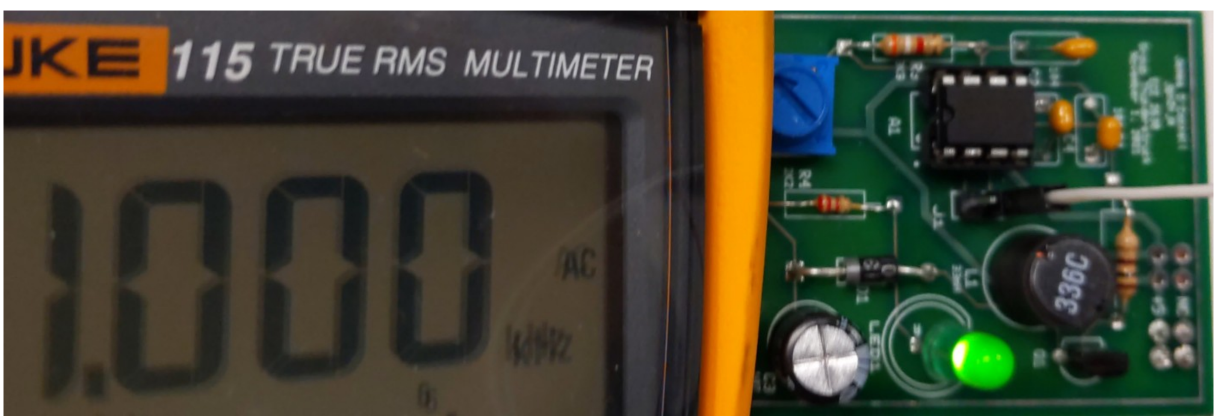

Figure 7: The Pulse-Width Modulator Output Set to 1 kHz

We then used the Oscilloscope function of the AD2 to obtain traces of V_{PWM} and V_{cap} as shown in Figure 8.

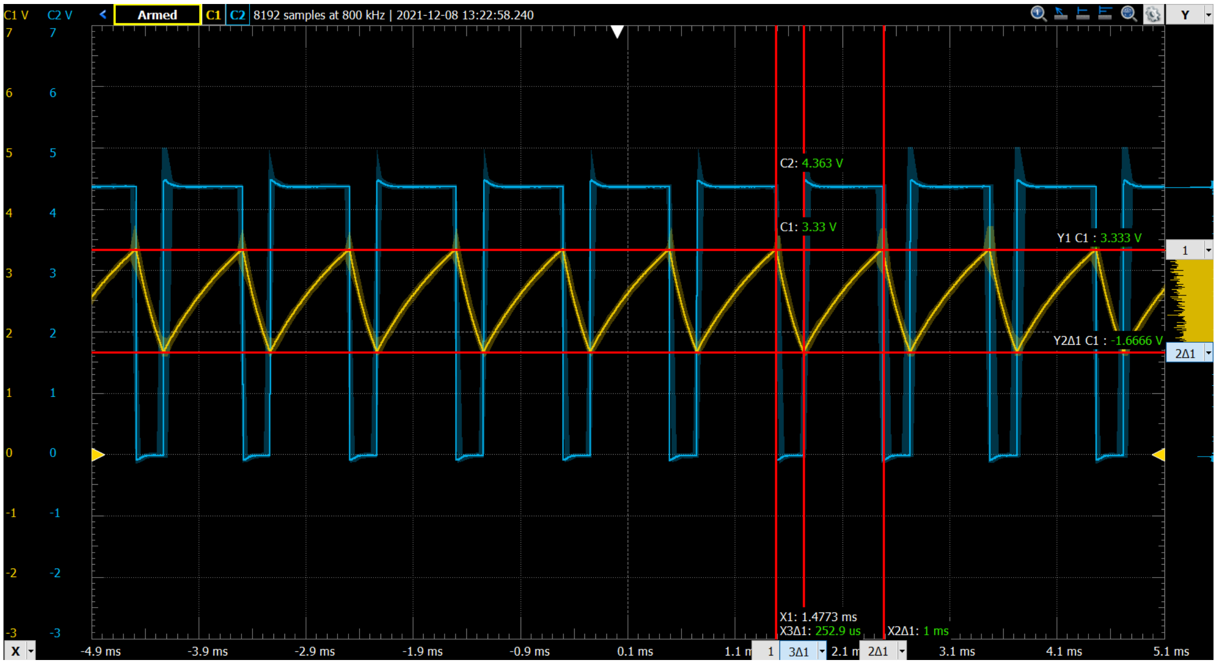

Figure 8: Trace Showing V_{PWM} and V_{cap}

From this we can see that with the signal set to 1 kHz, the V_{PWM} is a little below 5 V, coming in at 4.363 V, which is likely due to imperfections in the 555 Timer. However, the value of V_{cap} is exactly what we would have expected as the high value of 3.333 V is $\frac{2}{3} \times 5\,\text{V}$ and a drop of 1.666 V to the low value is a drop to $\frac{1}{3} \times 5\,\text{V}$. The trace also shows that the duty cycle is:

$$D = \frac{T_{on}}{T_{total}} = \frac{1\,\text{ms} - 0.2529\,\text{ms}}{1\,\text{ms}} = 74.71\%$$

which is within the specification of 50% to 80%. This is also acceptably close to the numerical prediction.

We then measured the output voltage V_{out} using the oscilloscope as appears in Figure 9.

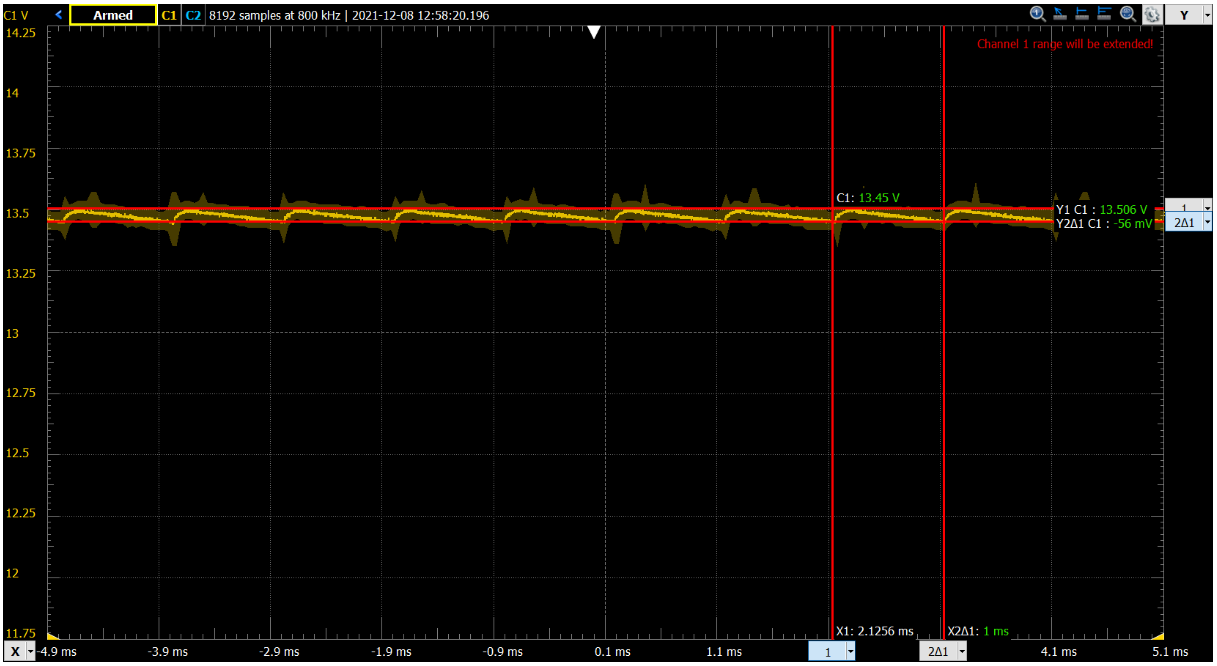

Figure 9: Trace Showing V_{out}

The output stays well above 13 V as was specified by the design requirement. However, the output drops from the maximum by

$$\frac{V_{hi} - V_{low}}{V_{hi}} = \frac{13.506\,\text{V} - 13.45\,\text{V}}{13.506\,\text{V}} = 0.4146\% > 0.3\%$$

Which is still a very small drop, however, the design requirements specified that the output should be no smaller than 13 V within 0.3%, which we assume means that our output should be stable and not vary by more than 0.3% throughout the discharge cycle regardless of the maximum voltage.

5 Summary and Error Analysis

Our design meets the requirements that the voltage output be above 13 V, but depending on the interpretation of the requirements, it does not meet the need to stay constant within 0.3% of its maximum value. There are several reasons for this, the first that comes to mind is that when we made our calculations for the needed size of the capacitor, we assumed the output voltage that needed to be sustained was 13 V, but the actual output voltage was 13.506 V. However, if we recalculate the capacitor size using the experimental output voltage, we have:

$$I_{load} = \frac{13.506\,\text{V} - 2\,\text{V}}{2.2\,\text{k}\Omega} = 5.23\,\text{mA}$$

$$\Delta q = (5.23\text{ mA})(729\ \mu\text{s}) = 3.813\ \mu\text{C}$$

$$C = \frac{\Delta q}{\Delta V} = \frac{(3.813\ \mu\text{C})}{0.003(13.506\text{V})} = 94.1063\ \mu\text{F}$$

which still is less than 100 μF. Thus, the capacitor's size is not the issue assuming the component is close to the nominal value. However, it has a tolerance of $\pm 20\%$, which means that it could easily fall substantially below the needed capacitance and still remain in tolerance. This is the most likely cause of the overly large drop.

As to the magnitude of the output voltage, in our analytical predictions, we found that the energy supplied by the inductor should be insufficient to provide 13 V when operating at 1 kHz. However, Multisim predicted that the result would be closer to 14 V. The experimental circuit matched neither of these, and instead fell at 13.506 V. There are several factors which could have contributed to this, including but not limited to variations in the components from their nominal values and insufficiencies in the model we used for the MOSFET. This latter case is highly likely, as MOSFETs are extremely complex. Furthermore, because Multisim also uses possibly imperfect models, a difference in models for the MOSFET is likely an issue for Multisim as well.

Were we to make another iteration of this design, we would first be sure to measure all of our components to make sure they are close enough to the nominal values to be usable. We would also like to study more accurate modeling for MOSFETS, as this would likely give us a more detailed understanding of the Boost Converter circuit and allows us to make better predictions. This would in turn allow us to fine tune the performance of our system to maximize its effectiveness and stability.