Flyback Converter using PWM Control

Maxwell Stonham

EE442 Course Project

Introduction

The flyback converter is an SMPS that converts an AC input into a DC output. The flyback converter done in this project will implement a PWM control to adjust the duty cycle depending on the output. The requirement of this project is to have an AC input of 100-130Vrms (141-184VAC) at 60Hz to be able to output 5V at 1A for a USB charger. Knowing this, our load will be set to R = 5V/1A = 5ohms. This also gives us the power that is expected to dissipate from our load which is P = 5V*1A = 5W.

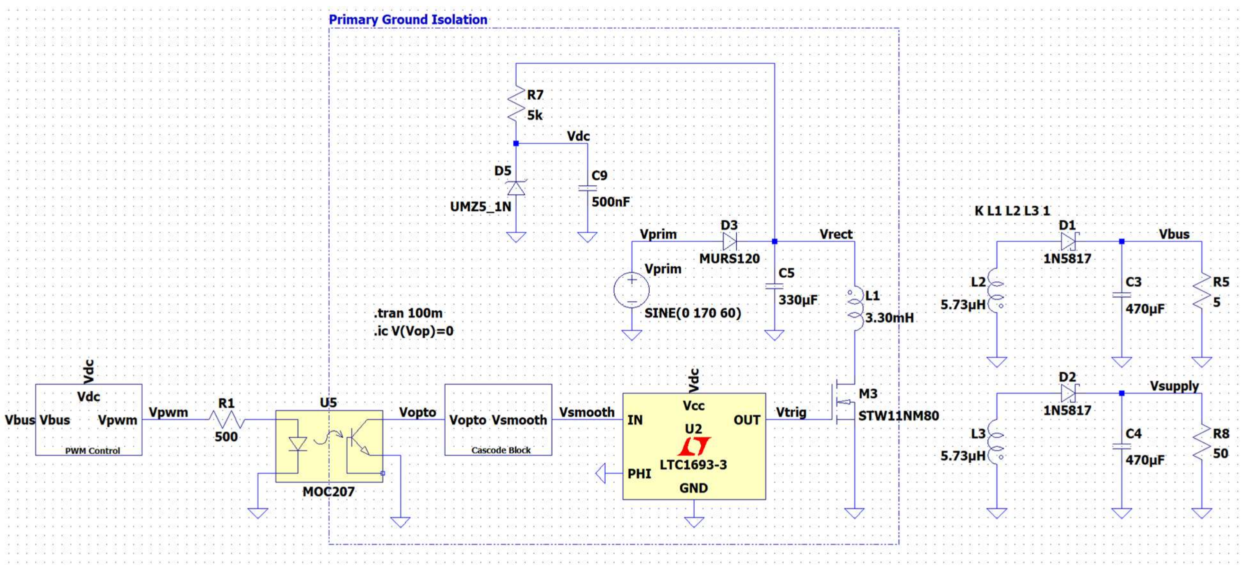

Figure 1. The flyback topology used in this project

Primary Side

Starting with our AC input, we design a half-wave rectifier on our primary side to convert our AC input to DC. Using the basic half wave rectifier topology, we will choose a MURS120 diode and a filter capacitor value of 333uF. The diode will allow current to only flow in one direction (forward-biased) and the filter capacitor is to smoothen out the output. The filter capacitor is chosen using the formula below:

$$\Delta V_o \approx V_m\left(\frac{2\pi}{\omega RC}\right) = \frac{V_m}{fRC}$$

Solving for C in the above equation given f = 60Hz and R = 4.55ohms, we obtain a C value of 366mF. This is much too high, so we will decrease this to 330uF since this can output an adequate DC voltage with only about a 10V ripple.

With our output now rectified, we continue to choose a transformer. For this design, the 750316022 by Wurth Eletronik is chosen because it can handle a 140-380VDC input at a frequency of 100kHz and has two secondary coils of the same ratio which outputs around 5VDC at 1.6ADC. Two secondary coils of the same turns ratio is chosen since the flyback converter will make use of the second secondary output to power up the electronics used in the circuit. This will be further elaborated later. The transformer chosen has a turns ratio of 24:1 with the primary having an inductance of 3.30mH. Solving for the secondary inductance using sqrt(L1/L2) = N1/N2 yields L2 = 5.73uH.

The MURS120 diode is chosen because it has the basic specifications needed. With a voltage breakdown of 200V, this is sufficient to have our rectifier operate without exceeding the 200V threshold since our AC input only goes up to 184VDC.

One thing to note is that a half-wave rectifier is chosen because it uses only one diode while a full-wave rectifier uses two for center tap rectification and four for the full-bridge rectifier. Using a full-wave rectifier will allow for a smaller ripple but cost more and take up more space. For our application, the half-wave rectifier is sufficient.

A power MOSFET also needs to be chosen. The STW11NM80 is chosen in this case because it has the basic requirements needed for our converter. This MOSFET has a Vto of 4.5V, and with our PWM switching at 0-5V, this allows the MOSFET to turn on with no issues. It also has a drain to source voltage of up to 800V, which is enough to switch our voltage of 184V.

Secondary Side

For the secondary side of the converter, Schottky diodes are chosen because they are good for switching at high frequencies and also have a lower forward voltage drop. Unlike normal diodes with a forward voltage drop of roughly 0.6-0.7V, Schottky diodes usually have a forward voltage drop of around 0.3V. Schottky diodes also have a fast recovery time which makes it ideal for high-speed switching. The 1N5817 is chosen for its high-speed switching capabilities as well as low voltage drop of 0.45V.

The filter capacitor chosen is based on the formula shown below:

$$C_{\min} = \frac{D}{R \cdot f \cdot (\Delta V_{OUT}/V_{OUT})}$$

To calculate this, we know our resistor value is at 5ohms, but we must also know our duty cycle and frequencies. Let us choose our switching frequency to be 100kHz. This frequency is chosen because it will give us a faster transient response time as well as a lower output ripple. The drawback of using this frequency compared to, say a 10kHz switching frequency would be that at 100kHz, we will have reduced efficiencies and higher power losses which can also result in more heating. For this project, we will stick to using a 100kHz frequency.

Our ideal duty cycle can be calculated by using the formula below:

$$V_{OUT} = V_S \cdot \left(\frac{D}{1-D}\right)\left(\frac{n_2}{n_1}\right)$$

Solving for D using Vout = 5V, Vs = 170VDC (120Vrms), N2 = 1 and N1 = 24, we get a duty cycle of about 0.41. Plugging this into the Cmin formula with f = 100kHz and R = 5ohms at a 1% output ripple gives us a Cmin of 414uF. Choosing a standard capacitor value, we can let C = 470uF. Our ideal duty cycle at 170V would be 0.41, but because we implemented a PWM control to adjust the duty cycle depending on the Vout, this exact capacitor value is not necessary. It is simply a reference to get an idea on what our capacitor value should be for a 1% output ripple.

PWM Control

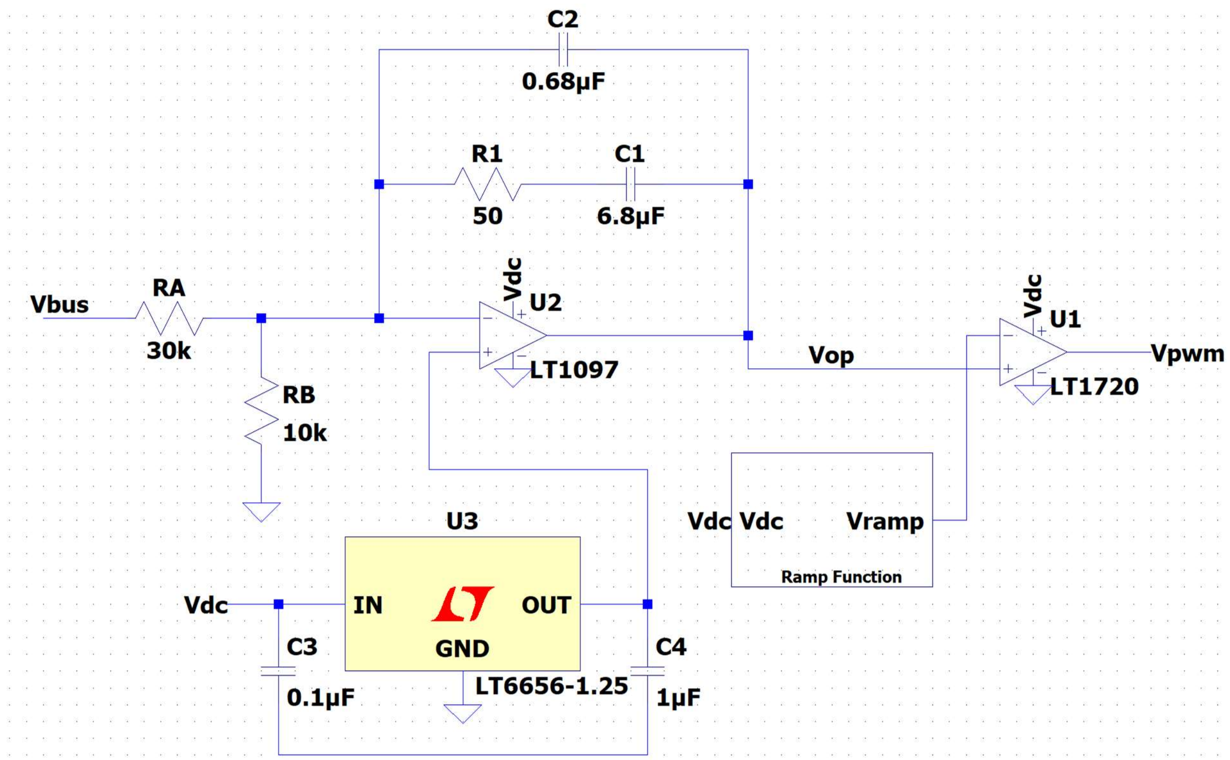

Figure 2. The PWM Control Circuit

In this flyback converter, a PWM control circuit is chosen to regulate the output voltage at different loads or AC inputs. The alternative to the PWM control circuit would be to use hysteresis control which is a much simpler option in terms of design and would also give faster response times, but the PWM was chosen since hysteretic control can have more noise as well as higher ripple.

The basic topology of a standard PWM control circuit is shown below. A standard bandgap of 1.25V is chosen for the non-inverting terminal of our op-amp. We use a voltage reference to achieve this using the LT6656-1.25 chip. This chip can handle a Vin between 2.51-18V, which is sufficient for our 5V power supply. We want our Vout to be about 5V, so making our RA = 3k and RB = 1k sets our voltage divider to output 1.25V in the inverting terminal of the op amp. The op amp chosen in this circuit is the LT1097 because it is a basic op amp that has the specifications needed to function the circuit properly. This op amp can handle a supply of 20V (we would only need to supply 5V).

To use the PWM control circuit, frequency guidelines need to be met. Starting off with our resonant frequency, we can calculate this by using the formula:

$$f_0 = \frac{1}{2\pi\sqrt{LC}}$$

Solving for our resonant frequency using the secondary inductance and filter capacitor (L = 5.73uH, C = 470uH) yields f = 3066.9Hz. Knowing this, we need to follow the two guidelines. The first one being that our zero frequency needs to be less than or equal to our resonant frequency. Our zero frequency can be calculated using the formula:

$$f_z = \frac{1}{2\pi R_2 C_1}$$

Choosing a zero frequency of 500Hz, this frequency would satisfy the first guideline. Setting our R2 = 50ohms, we solve for C1 = 6.37uF. We will set C1 = 6.8uF as it is a standard capacitor value.

Next, we need to satisfy the second guideline which would be to have the unity gain frequency less than or equal to a tenth of our zero frequency. This means that our unity gain frequency needs to be less than or equal to 50Hz. Our unity gain frequency can be calculated using:

$$f_{un} \approx \frac{1}{2\pi R_1 C_1} \qquad\qquad R_A || R_B = R_1$$

Solving for R1 using RA = 30k and RB = 10k, we see that R1 = 7500. Plugging this into our unity gain frequency, yields fun = 3.12Hz which is much lower than the guideline requirement of 50Hz but still satisfying it. Initially, a 3k and 1k pair of resistors were chosen, but this caused an overshoot in the output voltage to shoot up to around 12V. Increasing these resistor values but keeping the voltage divider ratio the same successfully lowered this overshoot to only about 8.5V making it more stable but respond slower. This will be seen in the output section of this report.

Now, a comparator is added to adjust the duty cycle depending on the output being fed into the system. We choose the LT1720 as out comparator because it is a low power, high speed comparator. The supply voltage ranges from 2.7-6V (we are using 5V to supply) and a maximum toggle frequency of 62.5MHz which is much larger than what we need it for (100kHz).

In the non-inverting input of the comparator is what we connect our op amp output to and on the inverting input is where we connect our ramp function. This is to ensure that the op amp output is properly compared to the ramp function to adjust the PWM signal accordingly.

<u>Ramp</u>

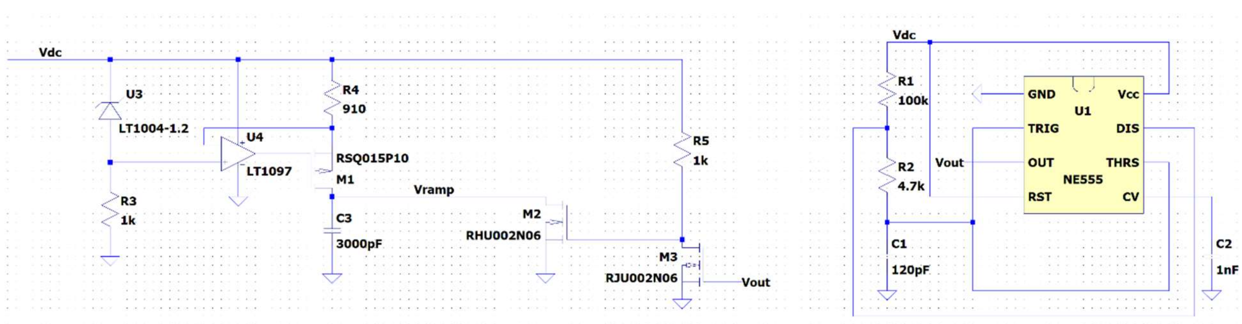

Figure 3. The ramp function generated by a 555 timer

The ramp function is generated by connecting a 555 timer as our oscillator to a current source. Our current source is designed using an op amp and voltage reference. The op amp chosen is the same one previously used while we choose an LT1004-1.2 as our voltage reference. We choose this voltage because initially, using a 1.25 voltage reference with a 1.25k resistor to supply 1mA as our current source did not give a sharp ramp, so to do something different, a 1.2V voltage reference was chosen with a 910ohm resistor. This supplied a 1.31mA current as our source that is charging up the capacitor that produces a ramp. Knowing that I/C = dv/dt, we have our I =1.31mA source. Let us choose a rate of 1V/2.25us for our ramp to have it go up to 4V in 9us. Solving for C produces around 2.9nF. Let us round this to 3nF or 3000pF since this is a standard used capacitor value.

Our oscillation is going to be supplied by a 555 timer. To generate this timer, we need to first calculate the resistor values. To calculate this, let us choose a duty cycle of 95%. Having chosen a frequency of 100kHz, our period is 10us. A 95% duty cycle means that our oscillator should be on for 9.5us and off for 0.5us. Using the formulas for the 555 timer:

$$t_{\text{high}} = \ln(2) \cdot (R_1 + R_2) \cdot C \qquad\qquad t_{\text{low}} = \ln(2) \cdot R_2 \cdot C$$

Let us choose R2 = 5k. Knowing t_low to be 0.5us, solving for C yields 144pF. Plugging this C value into our t_high formula, we can solve for an estimated R1 value of 90k. However, using these exact values produced a slightly longer t_high, so adjusting these values accordingly and considering standard resistor and capacitor values, we finalize our 555 timer values as R1 = 100k, R2 = 4.7k and C = 120pF. This produces an oscillation with a 95% duty cycle. Connecting this to an NMOS parallel to our 3000pF capacitor produces a ramp function that is high when our oscillator is low. To fix this, we add an inverter by attaching another NMOS so that the ramp is high when the oscillator is high to produce a sawtooth waveform.

Opto-isolator

The opto-isolator is used to isolate the two grounds: one from the primary and the other from the secondary. This is done for safety reasons so that if there happens to be a short from the secondary side, the user will not get shocked and have 170V surging through them. This is also good to separate the two sections so that if a short occurs in the electronics on the primary side and for some reason destroy the electronics, only one side of the converter is affected.

The opto-isolator separates the two grounds but allows a signal to pass through using an LED and BJT. The LED activates when a voltage is applied to it, in this case, the PWM voltage that is coming out of the comparator will pulse the LED on and off. This LED pulse is detected by the BJT and turns the BJT on and off this way. The opto-isolator that was chosen is the MOC207. This is also connected to a cascode circuit which is useful to overcome the parasitic capacitance, also known as the Miller effect.

MOSFET Driver

The MOSFET driver is used to increase drive strength and reduces switching losses. The one chosen (LTC1693-3) is sufficient to do this since it has a supply voltage range between 4.5-13.2V (which we will be supplying with 5V). This driver is also chosen since it has a PHASE pin which is utilized in this circuit to invert the output for the PWM to switch the Power MOSFET.

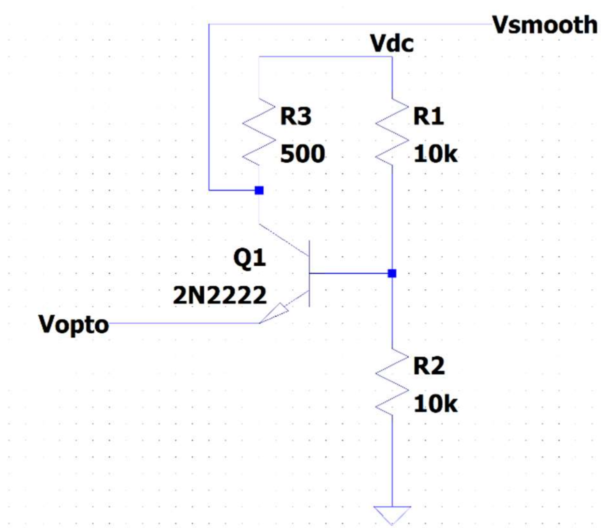

Figure 4. The cascode block

5V Power Supply

This design implements two alternative circuits. An ideal 5V supply (using a 5V voltage source) as well as a non-ideal 5V supply. Ideally, the 5V for our electronics would be powered by a second transformer on the secondary side, but due to issues with startup, the alternative was to use a Zener diode on the primary side to drop the voltage down to 5V. This dissipates lots of power on the resistor connected to it and greatly affects efficiency.

Results

In this results section, a few design differences were simulated:

1. Design with an ideal 5V power supply, but no opto-isolator\
2. Design with a non-ideal 5V power supply, with opto-isolator

Note that in these designs, leakage inductance is neglected. However, the designs are based on a real transformer available to buy but is simply simulated with perfect coupling.

Design with a ideal 5V power supply, with no opto-isolator
At Vin = 140V, at t = 100ms

R (Load)	Vout	Iout	Power (load)	Power (source)	Efficiency
5	5.05V	1.01A	4.98W	7.52W	66.2%

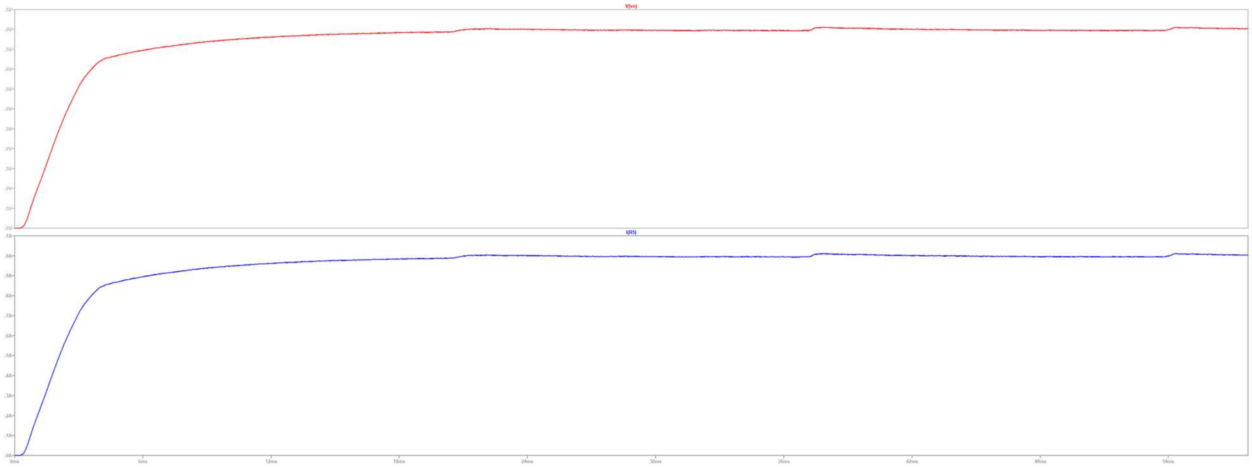

Figure 5. Simulation at Vin = 140V for our ideal 5V source

At Vin = 170V

R (Load)	Vout	Iout	Power (Load)	Power (source)	Efficiency
0	13.75V	N/A	N/A	4.1W	N/A
5	5.017V	1.003A	5.025W	7.204W	69.8%
50	8.69V	180mA	1.47W	3.41W	43.1%

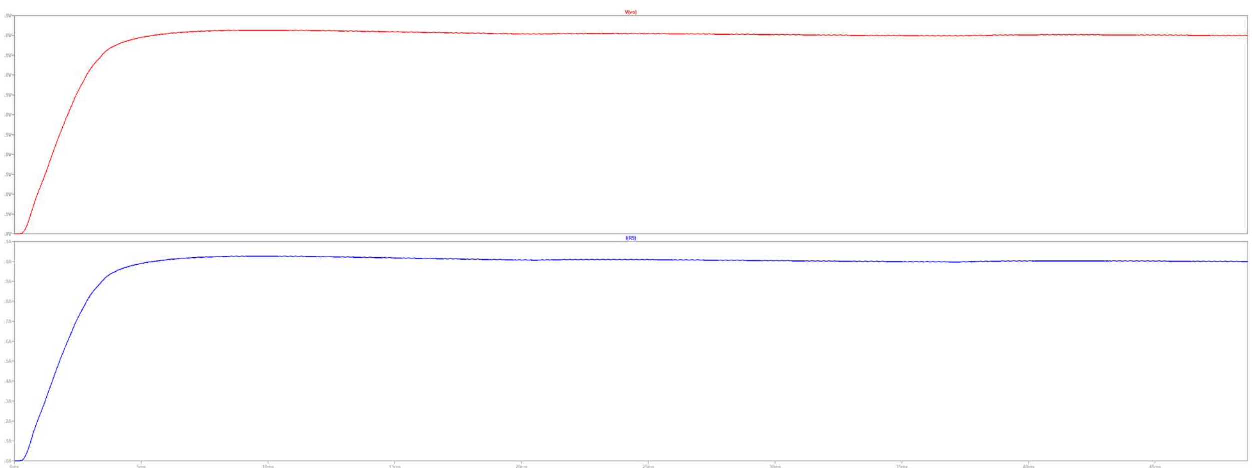

Figure 6. Simulation at Vin = 170V for our ideal 5V source

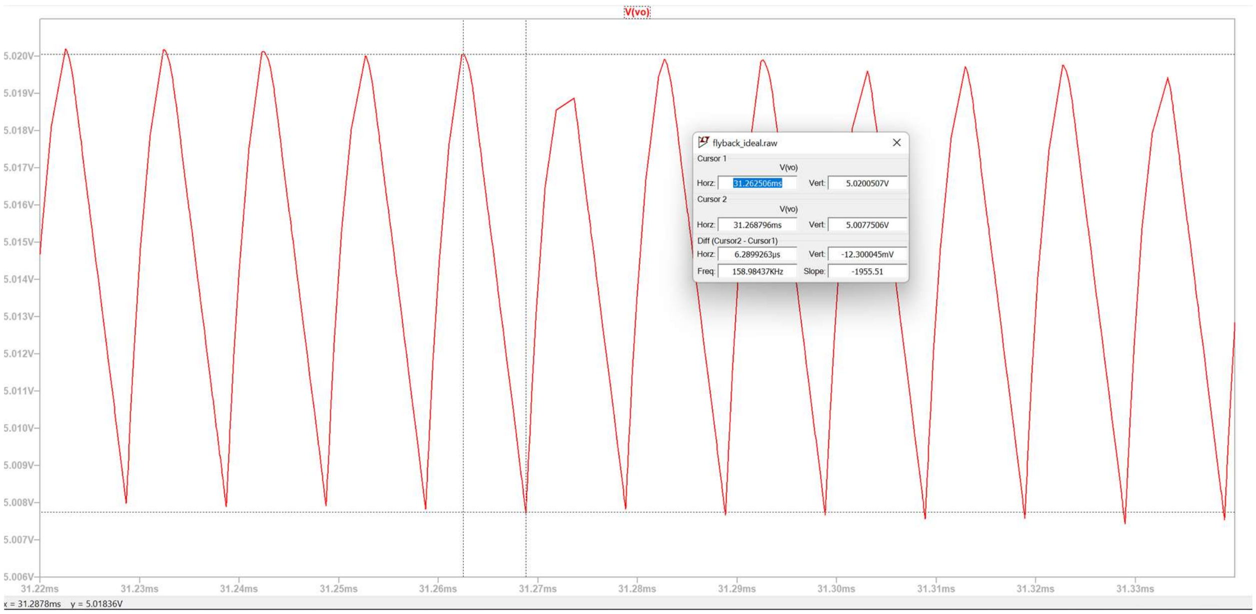

Figure 7. Simulation at Vin = 170V for our ideal 5V source showing the output ripple

At Vin = 185V

R (Load)	Vout	Iout	Power	Power (source)	Efficiency
5	5.06V	1.01A	5.125W	9.69W	52.9%

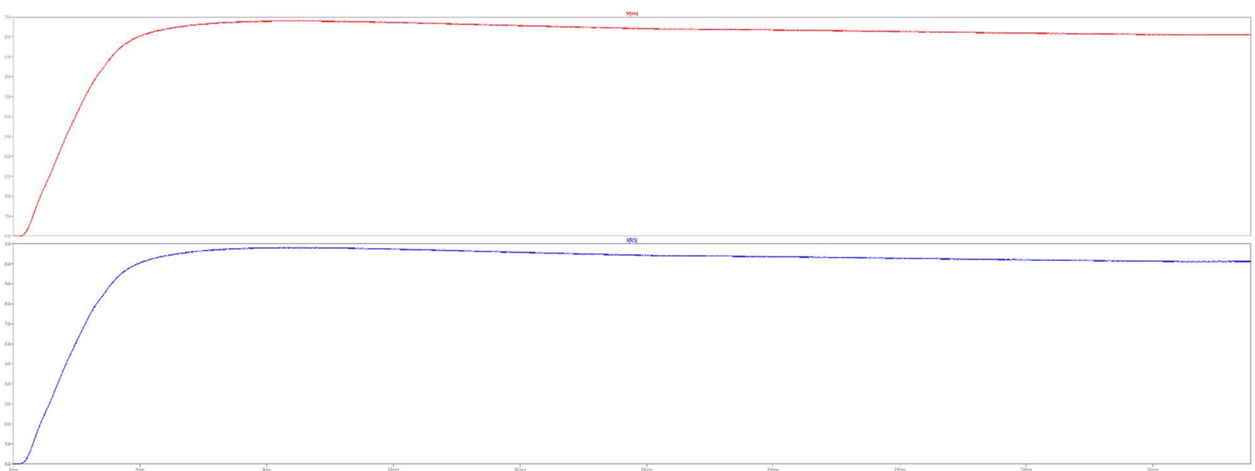
Figure 8. Simulation at Vin = 185V for our ideal 5V source

<u>Design with a non-ideal 5V power supply, with opto-isolator</u>

This design should be the final design in this project since it is a design without any ideal components and purely COTS parts (aside from the perfectly coupled transformers previously mentioned).

At Vin = 170V

R (Load)	Vout	Iout	Power (Load)	Power (source)	Efficiency
5	5.25V	1.05A	5.671W	13.665W	41.5%
50	9.33V	187mA	1.907W	9.753W	19.6%

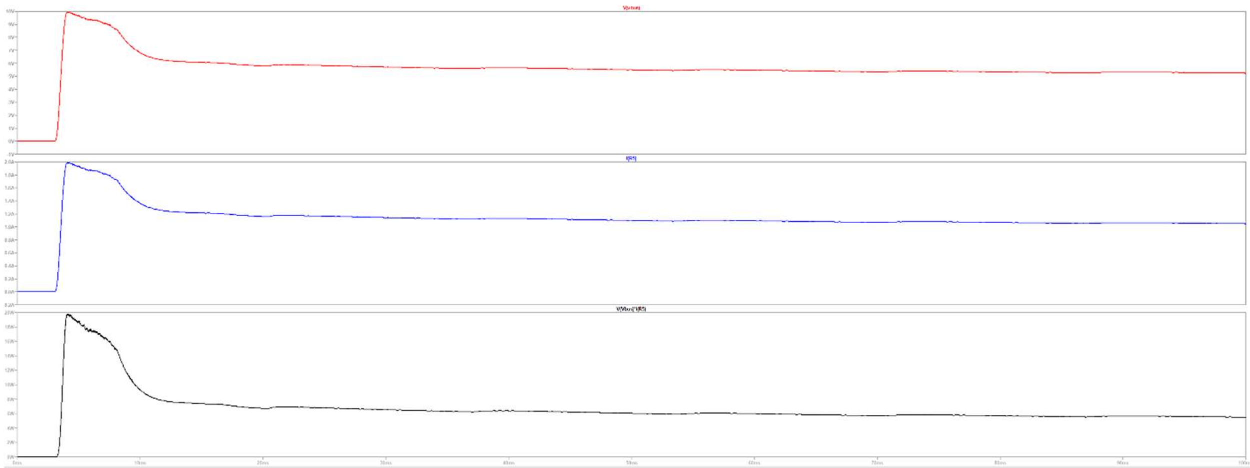

Figure 9. Simulation at Vin = 170V for our non-ideal 5V source

Size Estimate

From the datasheets of each major component in the circuit, we see that:

Component	Length	Width	Area
555 Timer	9.33mm	6mm	55.98mm^2
Transformer	17.75mm	13.46mm	238.9mm^2
LT6656	10.16mm	6.808mm	69.2mm^2
LTC1693	5mm	3.1mm	15.5mm^2
LT1720	6.2mm	5mm	31mm^2
USB	15mm	10mm	150mm^2

As a rough estimate, we could assume that we would need at least about 560mm^2 of area. This is not including the resistors, capacitors and diodes.

Weaknesses and Future Work

The drawback to this converter would be the 5V power supply. For one, the 5V supply is taken directly from the input by using a Zener diode to drop the voltage to 5V. Connecting this to a resistor and capacitor, we can output 5V consistently. However, the power dissipated by the 5k resistor is at about 6W. This can definitely get hot and the designer would need to take into consideration this heat dissipation and whether a chosen resistor part can handle high power.

On top of heat and power rating consideration, this would also affect the overall efficiency since this one resistor is dissipating so much power. Another thing to note would be the ground isolation: by taking the voltage directly from the AC input on the primary side, supplying this voltage to the feedback control electronics would connect the two grounds together which would render the opto-isolator useless since there isn't a way to shut off this supply after the electronics power up.

In terms of an ideal 5V source, the simulations above show that at the common 120Vrms input, we see that our Vout values settle between 5-5.1V with a ripple of about 12mV. This shows that the flyback design works using an ideal 5V supply and no opto isolator. However, this method proves to be very power inefficient due to the Zener diode dropping the voltage from the input to 5V thus dissipating lots of heat in the 5k resistor (around 5Watts).