Multi-Stage Power Conversion

Joe Driscoll, Paul Hemberger, David Yamnitsky

1. Introduction (Joseph Driscoll)

MSPC is a three-stage power converter system where each stage not only supports a useful application, but also powers its subsequent stage. The first stage, a bench power supply, takes AC from the wall and converts it to DC ranging between 1 and 12 volts with up to 2 amps. The second stage, the wireless battery charger, transfers that power wirelessly using a magnetic field, and then charges store-bought rechargeable batteries ranging in voltage from 1.5 to 9 volts. Finally our third stage, the portable smartphone charger, uses those rechargeable batteries to charge a smartphone. This requires that it be able to convert the batteries voltage, ranging from 1.5 to 14 volts, to the 5 volts required to charge smartphones. Figure 1.1 shows the block diagram for MSPC. David Yamnitsky was responsible for the bench power supply, Joseph Driscoll was responsible for the wireless battery charger, and Paul Hemberger was responsible for the portable smartphone charger.

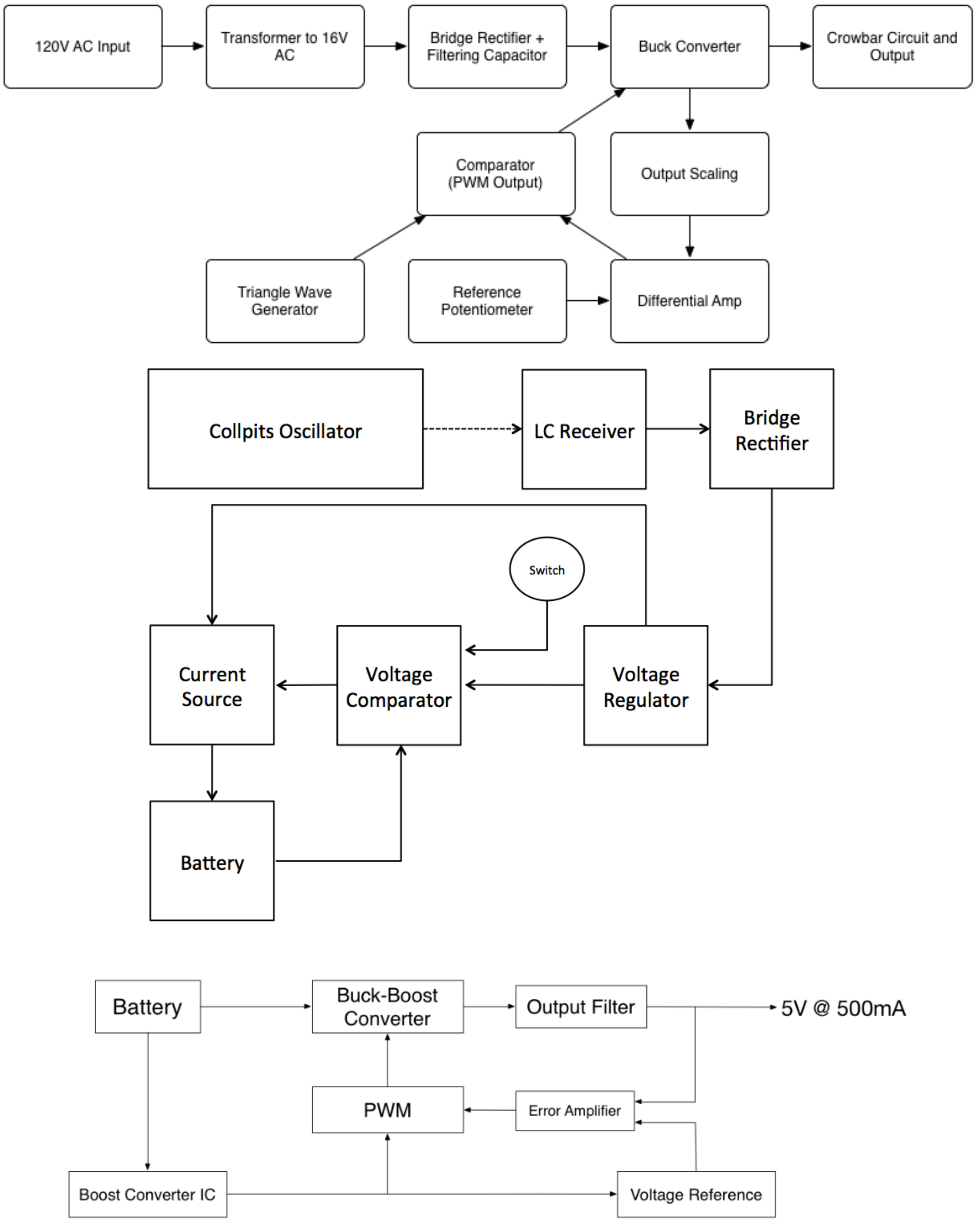


Figure 1.1: Block diagram of each module: power supply, wireless charger, and USB charger

2. Adjustable Bench Power Supply (David Yamnitsky)

The adjustable bench power supply was designed and built by David Yamnitsky. It takes input from an AC-AC wall wart transformer, and has an adjustable DC output. Figure 2.1 is a block diagram showing the components of the system. The circuit takes the 16V AC from the transformer and uses a bridge rectifier and large filtering capacitor to produce a DC output of approximately 16V. Then, a feedback controlled buck converter steps down the voltage for the output, which is adjustable with a potentiometer. For the feedback path, the buck converter output is passed to a voltage divider to scale it to the 0-12V logic range, then a differential amplifier compares it with a reference set with a potentiometer, and applies a gain. The gain is compared with a triangle wave to produce the PWM signal for the buck converter.

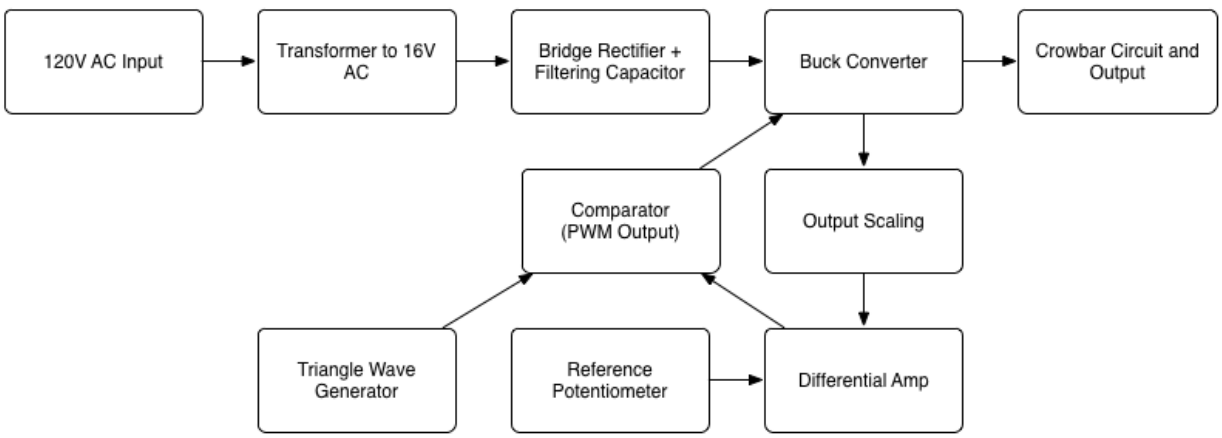
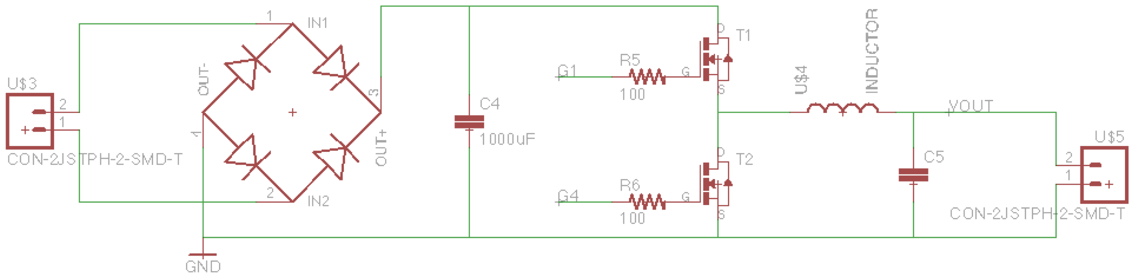


Figure 2.1: A block diagram of the variable power supply

Power Stage

After the rectifier and filtering capacitor there is a buck converter, which consists of a MOSFET half bridge and LC low pass filter. I chose a 30 kHz switching frequency, 100uH inductor, and 100uF capacitor. The resonant frequency of the LC tank is 1/(2*pi*sqrt(L*C)) = 1.5kHz, which is safely away from the switching frequency. I did not have a voltage ripple specification to meet, so I decided to build the circuit and confirm that it is reasonable experimentally. With varying loads, I found that the voltage ripple never exceeded 10% of the voltage output. For example, at 10V output, the voltage ripple was on the order of 100mV without load, and under a load that drew just under 1A, the voltage ripple was under 1V.



<u>Figure 2.2:</u> Power Stage

Triangle Generator

The triangle wave generator uses a 555 timer in an astable configuration. By keeping R9 small compared to R10, the output duty cycle is approximately 50%. For a switching frequency of 30 kHz I chose R9 = 1k, R10 = 22k, C2 = 1 nF.

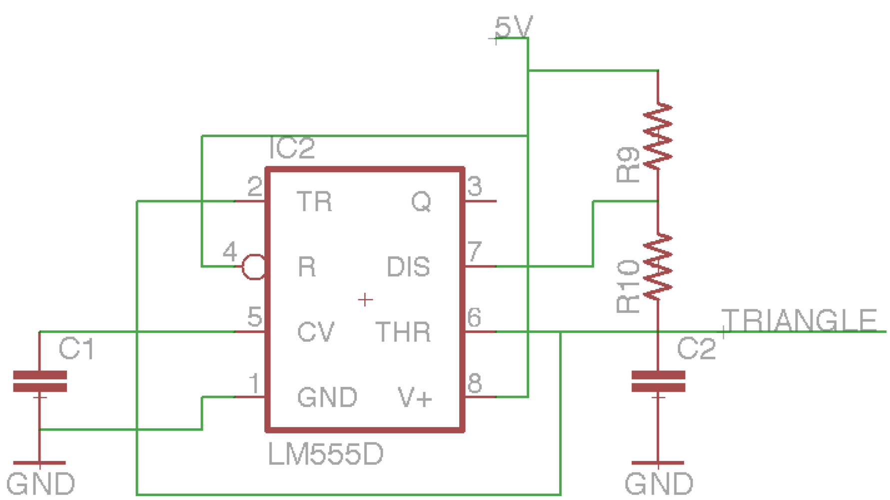


Figure 2.3: Triangle Wave Generator

PWM Generator

The triangle wave and differential amplifier output are fed to a comparator, producing a PWM wave output. This PWM is then fed to the delay circuitry.

Delay Circuitry

In order to prevent shoot-through in the half bridge, a delay circuit takes the PWM and enforces that neither MOSFET is turned on before the other is first turned off. The circuit accomplishes this using Schmitt trigger inverters with RC circuits at their inputs. For each circuit, I chose R =

1k, C = 1uF for a delay time of 1 microsecond, well above the turn-on and turn-off time of the MOSFETs used.

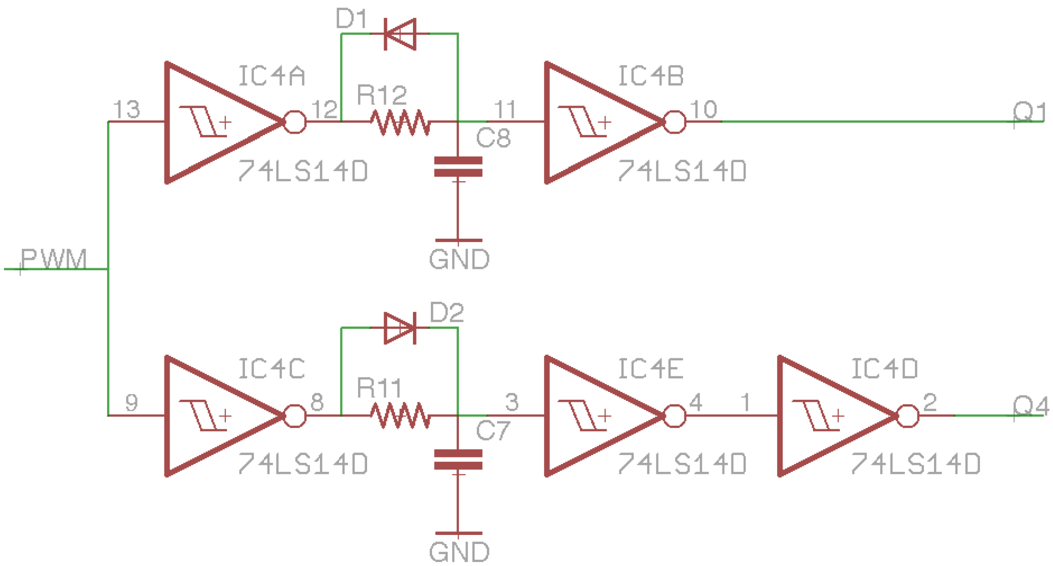


Figure 2.4: Delay Circuit

Differential Amplifier
The output, Vout, is then fed to a feedback path. After passing through a voltage divider to scale the 0-16V output down to 0-5V, a differential amplifier computes the difference between the output and a reference, and multiplies it by a gain. I chose R2=10k, R1=1k for a gain of 10. R8 and R7 are therefore 10k and 1k respectively.

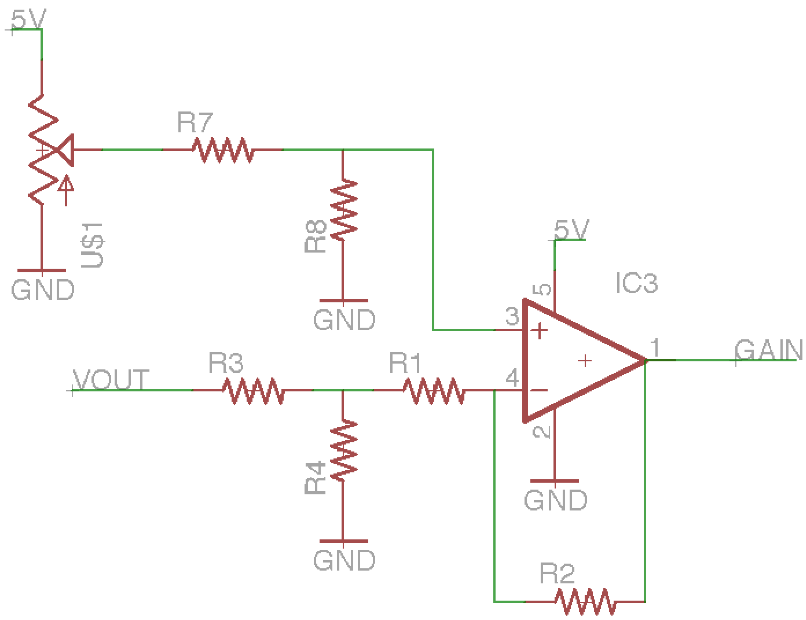


Figure 2.5: Differential Amplifier

Performance
The feedback control worked very well. The rectified AC input had some degree of ripple on the filtering capacitor. I tested the circuit both with and without feedback, and found that the feedback path mostly eliminated the low frequency oscillation. One issue I found was that the delay circuit requires a minimum pulse width of 2 microseconds to cause the output to rise at all (because it delays by 1 microsecond on the rise and fall). This means that the power supply cannot output near-zero voltages. In testing the minimum voltage output was around 500 mV.

3. Wireless Battery Charger (Joseph Driscoll)

The wireless battery charger was designed and constructed by Joseph Driscoll. Figure 3.1 shows a block diagram of the converter. The wireless battery charger is composed of four modules: the wireless power transmitter, the wireless power receiver, the variable battery charger, and the battery charge indicator. Each was individually designed and tested before being added to the final product. Figure 3.2 shows a schematic of the wireless battery charger.

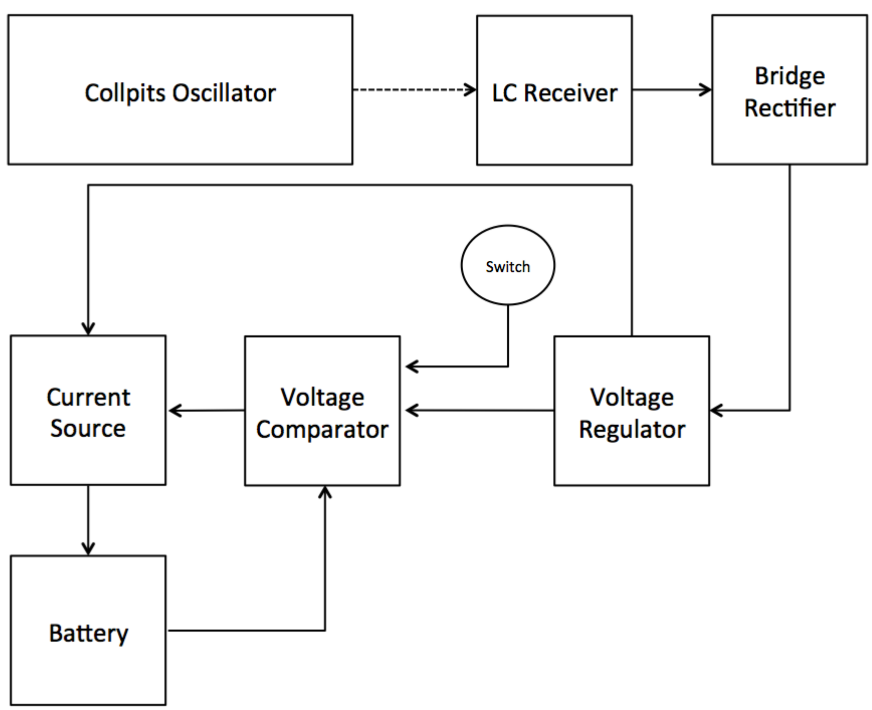


Figure 3.1: A block diagram of the wireless battery charger

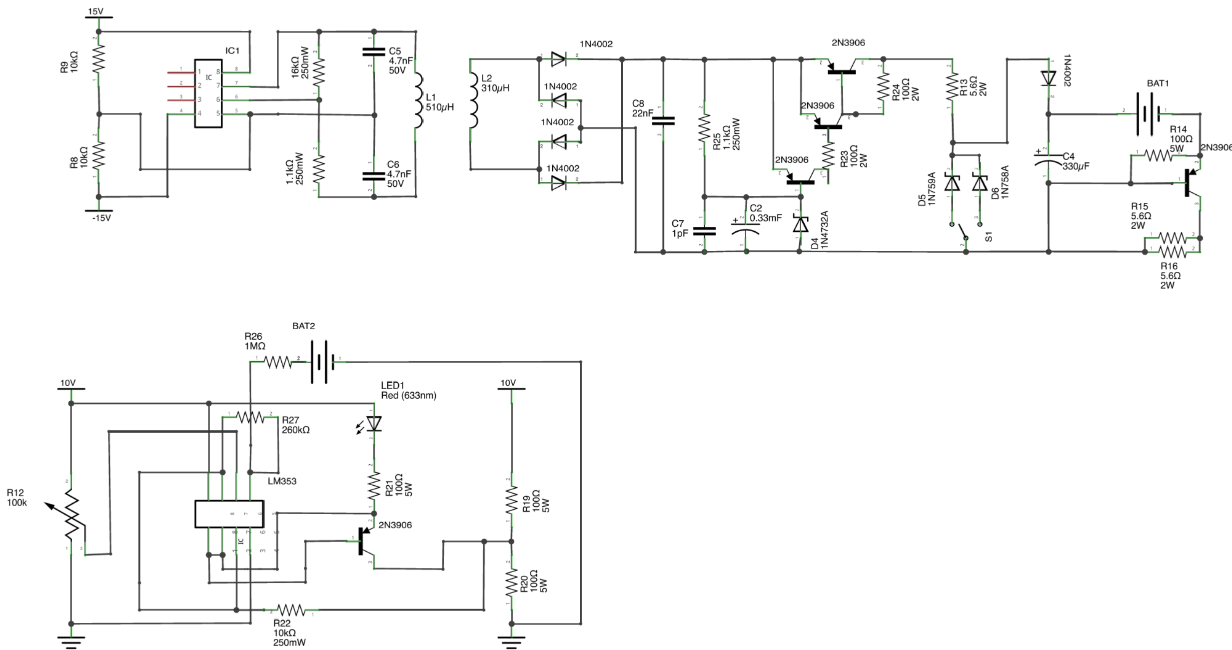


Figure 3.2: A schematic of the wireless battery charger

Wireless Power Transmitter

The wireless power transmitter was designed to convert direct current into a magnetic field, allowing it to transfer power to the receiver.

Initial Design:

The initial design of the transmitter received 30 volts from the bench power supply. The wireless power transmitter would use an opamp Colpitts Oscillator with a resonant frequency of 700 kHz. The inductor would be turned by hand out of 22-gauge wire so that its inductance and radius could be easily reconfigured.

Initial Concerns:

The initial design presented two significant challenges. The first was transferring power with limited input current, as the bench power supply is limited in power by the transformer it uses from the wall. The second challenge was maintaining high efficiency energy transfer. To address these challenges, the plan was to experimentally find the best resonant frequency to transfer power and to create an LC circuit at the power receiver with the same resonant frequency. If that was not sufficient, the backup plan was to match impedance between the transmitter and receiver. During the initial design, there was also a concern that due to the chosen transformer, the bench power supply would not be able to supply the power transmitter with 30 volts. The backup plan was to use the lab kit power supply but to continue to limit the amount of current the power transmitter pulled from this supply in order to simulate a low output current bench power supply.

Final Design:
The final design of the wireless battery transmitter was similar to the initial design. Figure 3.3 shows a schematic of the final design. The transmitter consisted of a colpitts oscillator with two 4.7 nF capacitors, a hand wound .51 mH inductor, and an lf353 opamp. The wireless transmitter accepted +15 volts and -15 volts from the lab power supply to create a 30 volt DC signal that it then oscillated to a 15 volt peak to peak sine wave with a frequency of 150 kHz. It was constructed to pull less than 1 A from the power supply.

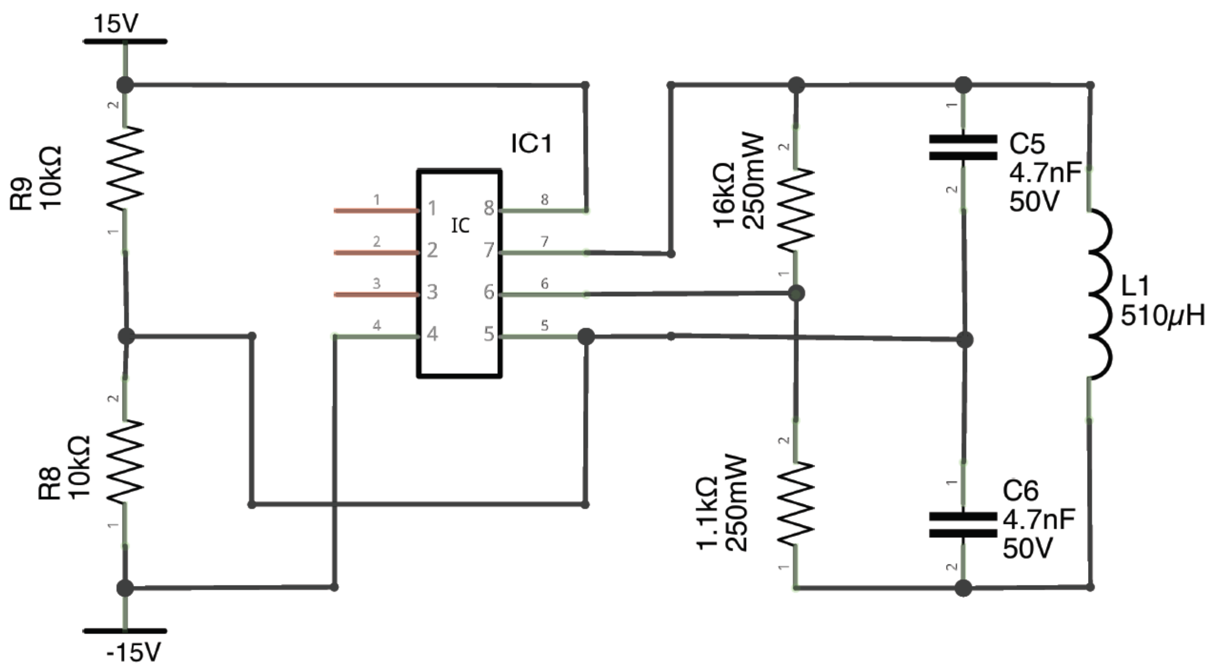


Figure 3.3: A schematic of the wireless power transmitter

Implementation:
Because creating an inductor of specific inductance is much more difficult than purchasing a capacitor of appropriate capacitance, the wireless transmitter was designed around the hand-wound inductor. The transmitter's and receiver's inductors were designed to have the same area, so one could easily be stacked on top of the other. However, winding two inductors with similar diameters proved challenging, and three methods of winding were experimented with until the final method was established. The final inductor was wound using a power drill with a plastic cylinder instead of a drill bit attached to its head. The transmitter's inductor consisted of 51 turns with a diameter of 1.5 inches and a height of .25 inches. The resonant frequency of the oscillator was determined experimentally. While the initial design was a resonant frequency of 700 kHz, the slew rate of the lf353 required it to be lower. The largest sin wave was determined to occur at 150 kHz. The module was tested using the lab power supply and the oscilloscope.

Future Design:

In a future design, it would be interesting to remove the low input current restriction to the transmitter. The original reason for the current restriction was to allow the bench power supply to power the wireless battery charger and while it was an interesting challenge to to overcome, higher input current would provide a much faster charging rate for the battery. To pull more current from the power source, the wireless charger would additionally have a current amplifier output stage that pulled current directly from the power supply.

Another interesting future design would be to use better wire for the inductor. The current design used standard 22-gauge wire because it was easily available in lab. Using more ideal wire for the inductor would allow for a greater energy transfer between the transmitter and receiver.

Wireless Power Receiver

The wireless power receiver was designed to draw power from the transmitter using magnetic inductance and convert that power into a stable DC output.

Initial Design:

The receiver was initially designed to collect power using a parallel LC circuit with a resonant frequency of 700 kHz. A bridge rectifier with a ripple capacitor would then convert the alternating current into direct current. A voltage regulator, T0220-3, would then be used to create a constant 10 volts output. A backup regulator circuit was designed in the event that the T0220-3 failed to work at such low power. The backup voltage regulator consisted of a large capacitor to store energy, a Darlington pair to pull current out of the circuit, and a 10 volt zener diode to set the output voltage at 10 volts.

Initial Concerns:

The biggest challenge presented by the initial design was maintaining high efficiency power transfer. Just like the transmitter, the plan to achieve high efficiency was to tune the transmitter and receiver to the same resonant frequency and, if necessary, match their respective impedances. Another concern at the beginning of the design processes was the ability of the module to regulate the voltage to 10 volts while using very little power.

Final Design:

The final design of the power receiver was implemented using the backup voltage regulator. Figure 3.4 shows a schematic for the final design of the receiver. Wireless power is captured for the module by the hand wound 0.31 mH inductor. An iron ferrite rod is inserted in the center of the transmitter inductor and receiver inductor in order to increase the power transmitted. The AC signal is then rectified by a bridge rectifier consisting of four 1N4002 diodes. A 22 nF capacitor was inserted in between the new ground and DC voltage in order to limit rippling. This signal is then fed into the voltage regulator. The voltage regulator begins with two capacitors, one 1pF and one 330 uF, to capture ripple and store energy from the DC signal. Three pnp bjts, 2N3906, are then used to pull current out of the circuit. They are configured like a Darlington pair, where the emitter of one feeds into the base of another. This allows for a much greater amplification of current than could be achieved with a single bjt. Finally, a 12 volt zener diode, 1N759A, was placed at the emitter of the first bjt. This held the output at 11.2 volts.

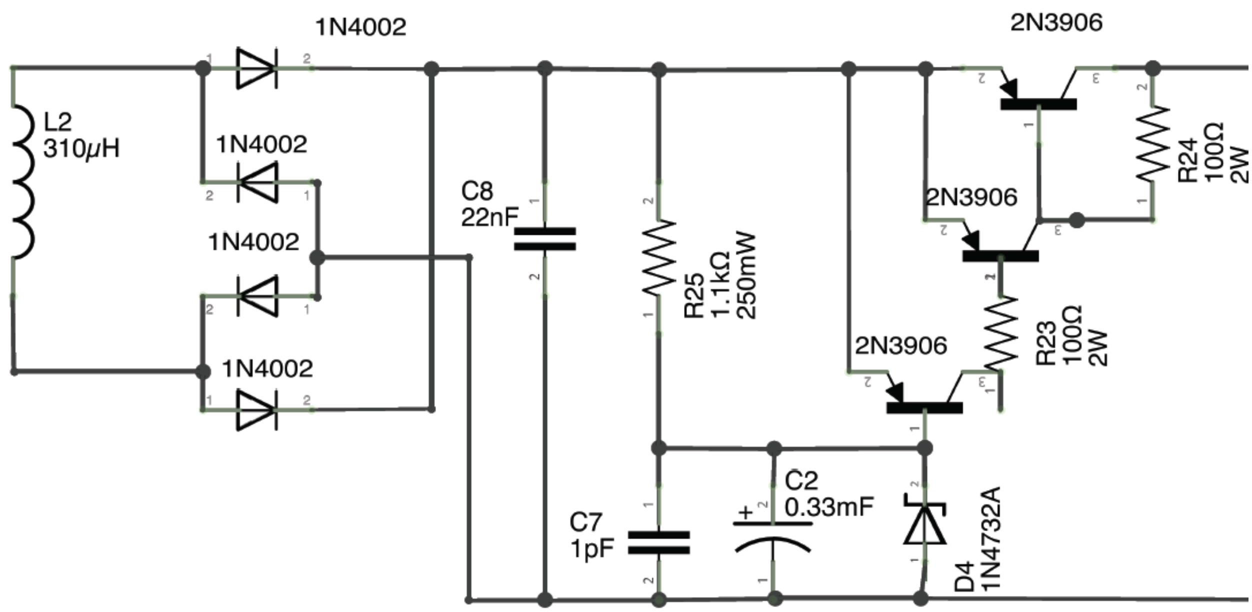


Figure 3.4: A schematic of the wireless power receiver

Implementation:

The power receiver was also designed around the hand wound inductor. As described for the power transmitter, a power drill was used to create an inductor with a radius of 1.5 inches, a height of .2 inches, and an inductance of .31 mH. To create a resonant frequency of 150 kHz, a ripple capacitor of 22 nF was used. Initially, the T0220-3 voltage regulator was added to the circuit. Unfortunately, the T0220-3 could not operate correctly with such low power, so the backup plan: creating a linear voltage regulator with a zener diode and bjts, was implemented instead. While the original backup plan called for only a single Darlington pair, additional current was needed for the battery charger to operate correctly. To create this current, a third bjt was added to the Darlington pair configuration, effectively doubling the voltage regulator's current output. The original voltage regulator circuit also depicted a 10 volt zener diode to hold the output at 10 volts. However, this failed to take into account the voltage drop from the bjt transistors. To account for this voltage drop, a 12 volt zener diode was used, changing the output voltage of the power receiver to 11.2 volts. The voltage conversion and regulation of this circuit was tested using a function generator and the power receiving capability was tested using the wireless power transmitter.

Future Design:

In a future implementation, it would be interesting to charge batteries up to 15 volts. This would require at least a 16 volt output voltage from the voltage regulator. To generate this output, an 18 volt zener diode would be used instead of a 12 volt zener diode. If I was to increase the power that is transferred between the transmitter and the receiver I would also have to replace the current resistors in the receiver for ones that are rated for higher power.

Variable Battery Charger

The variable battery charger was designed to intelligently charge rechargeable store-bought batteries that vary in voltage. The term intelligent refers to the circuit's ability to stop charging the battery once the battery has been fully charged, even if the battery is still connected to the charger. The circuit is able to charge batteries ranging in voltage from 1.5 to 9 volts, as long as the appropriate zener diode is selected.

Initial Design:

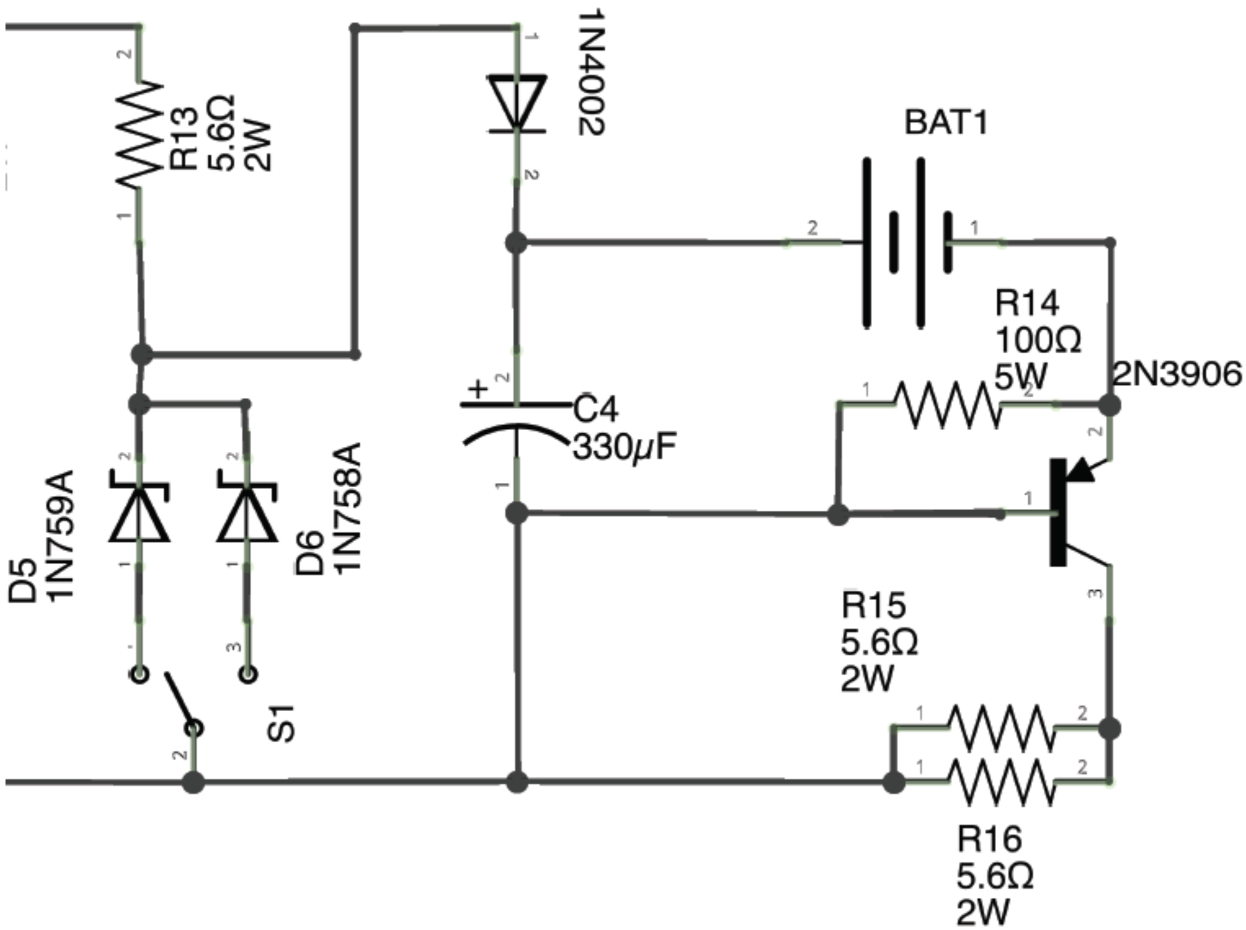
The initial design was quite different from the final design of the battery charger. The design received an input voltage of 10 volts. It included a potentiometer that allows the user to select what voltage (out of ten) they wanted to charge their battery to. This potentiometer provided an offset voltage for a Schmitt trigger. The Schmidt trigger was designed to output +10 volts if the input was .2 volts + the offset voltage and 0 volts if the input was -.2 volts + the offset voltage. It received its input signal from the battery. The output of the Schmitt trigger was the input to an opamp and bjt current source. This current source worked by making the collector of the bjt a specific voltage (either 10 or 0 volts). The collector was connected to 10 volts through a 100 ohm resistor. Thus the current through the bjt was either 0 A or 100 mA. The battery was originally between the emitter of the bjt and virtual ground. An LED is used to display to the user when the battery is charging.

Initial Concerns:

The initial concern with this design was the ability of the circuit to intelligently charge the battery. However, during construction, two major problems were discovered with the module, and it was abandoned in favor of the final design. The first problem was that the batteries voltage prevented the current source from working correctly. The second problem was that the opamp of the Schmitt trigger required too much power to operate correctly. This circuit was instead used to construct an optional module: the battery level indicator.

Final Design:

Figure 3.5 depicts the final design of the smart battery charger. In the final design of the smart battery charger, the user sets the voltage that they wish to charge the battery by flipping a switch. This switch connects a specific zener diode to the circuit whose voltage is .1 volts above the battery's charged voltage level. For example, to charge a 9 volt battery, the user flips the switch to the 9.1 volt zener diode. After the input voltage is set by this zener diode, it flows through another diode, a 1N4002, that ensures current flows in the correct direction, A 330 uF capacitor connects the diode to ground and stores energy when the battery is not connected to the circuit. The battery's positive side is connected in between this capacitor and the 1N4002 diode. It's negative side is connected to a parallel 100 ohm resistor and the collector of a pnp bjt (2N3906). A 100 ohm, 5 watt resistor then feeds into both the base of the bjt and ground. Finally, the emitter of the bjt feeds into two parallel 5.6 ohm, 2.5 watt resistors that then connect to ground. When the battery is connected and if the input voltage is greater than the battery's voltage, the bjt is turned on and pulls current from the capacitor through the battery. However, if the battery has a higher voltage than the input voltage, the bjt turns off and current flows in the opposite direction, discharging the battery into the capacitor. When the bjt is turned on, an average of 4 mA flow into the battery. Experimentally, this resulted in a charging rate of 1 volt per hour using a 9 volt Energizer rechargeable battery (NH22NBP).



<u>Figure 3.5:</u> A schematic for the smart battery charger

<u>Implementation:</u>

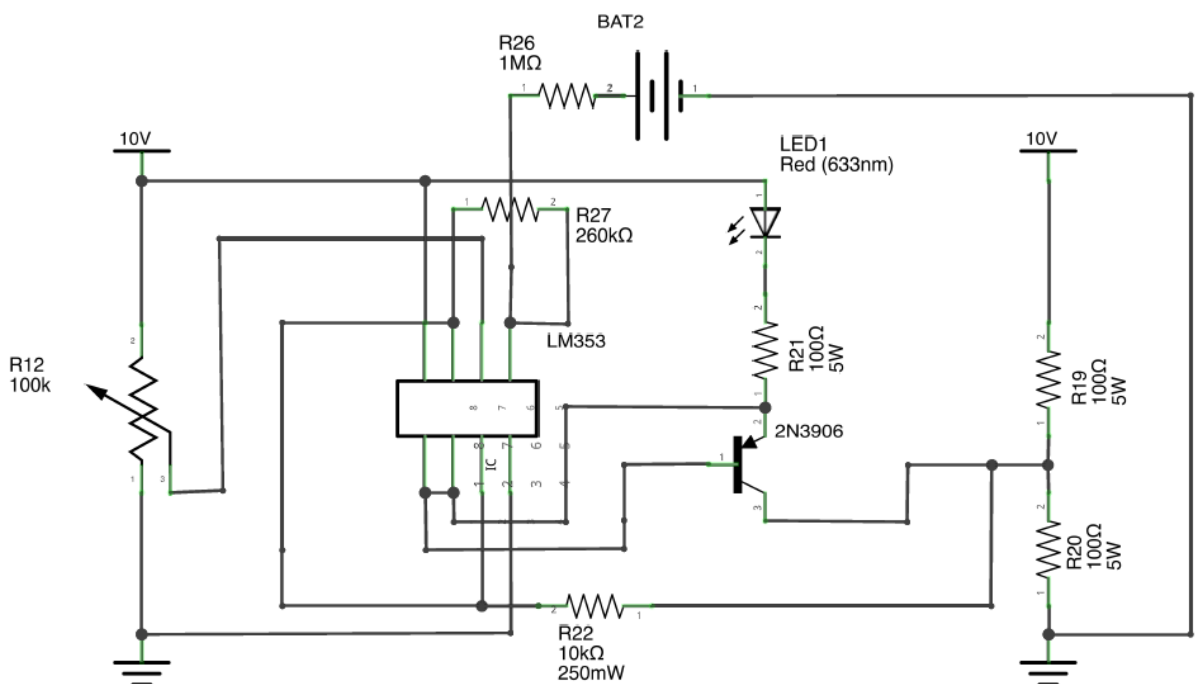
This module was very difficult to test because with a high power supply like the lab bench, most of the components are unprotected and can easily burn out. Fortunately, this was the last module constructed so it could be tested using the low power wireless power receiver as a supply. While the original plan was to demonstrate safe charging with both a 9 volt and 1.5 volt battery, the zener diode required for the AA 1.5 volt battery, a 1.9 volt zener, was determined to be too expensive($60). Therefore during the project showcase, the demonstration consisted of only safely charging a 9 volt battery.

<u>Future Design:</u>

For a future design, many it would be interesting to include more zener diodes into the module so that more than two types of batteries (9 volt and 11.9 volt) could be charged. If I were to make increase the power transmitted wirelessly to the power receiver, I would have to either use higher rated components or a different battery charging design. Additionally, with slightly higher power it would have been interesting to include an LED after the battery to indicate whether or not the battery was charging.

Battery Level Indicator

Because the smart battery charger did not contain an LED light to indicate whether the battery is charging or not, the initial design for the smart battery charger was repurposed into a battery voltage indicator. Figure 3.6 depicts this new circuit. The user determines what voltage the battery should be at full charge and turns the potentiometer to reflect that voltage. The LED then lights up when the battery is .2 volts below the specified voltage, indicating that the battery needs to be charged.



<u>Figure 3.6:</u> A schematic for the battery level indicator

Acknowledgements

I could not have completed this module without help from Gim, Devon, electronics-tutorials.ws, brettcave.net, ecircuitcenter.com, Paul, and David.

4. Battery-powered Smartphone / USB Charger (Paul Hemberger)

The converter was designed to be a high-efficiency design that can charge a USB device from any battery in the range of 1.5V to 14V. Its target output was a fixed 5V and with up to 500mA of current, the standard for USB charging.

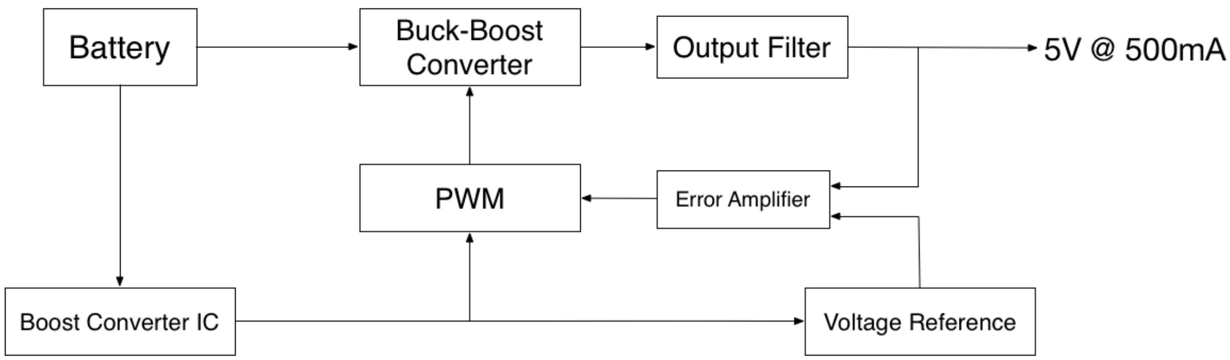


Figure 4.1: A block diagram of the USB converter

4.1 Buck-boost Converter

The central piece of the design is a buck-boost converter. The buck-boost is a switching converter topology that allows the output voltage to be less than and greater than the input voltage depending on the duty cycle. This lends itself well to our wide input range. The buck-boost is also a logical choice because it is an efficient design, as switching converters typically are, so it is well suited for battery-powered operation.

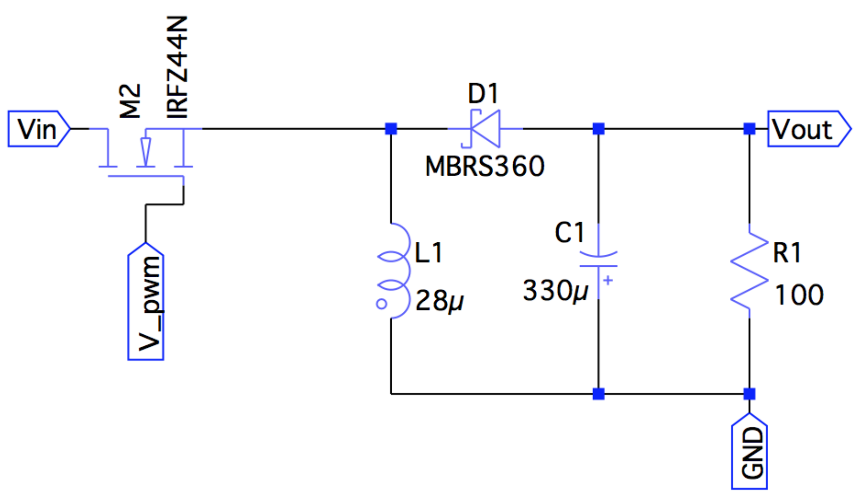


Figure 4.2: Schematic for buck-boost design

My initial values for the buck-boost's inductor and capacitor came from working through a TI design note[1] given a switching frequency of 80kHz. However, after simulation efforts and consultations with the TAs, they were revised to the values shown. Unfortunately for readers looking for step-by-step details, the final values were not derived in any systematic way, but a series of, "Oh let's try this!" experiments.

4.2 Error Amplifier

The error amplifier is conceptually simple: compare the buck-boost's output voltage against a reference, and adjust the duty cycle depending on how close it is. As I found out, conceptually simple circuits are not necessarily easy to design, and indeed can be far from it. I spent the majority of my time working out kinks in this part of the circuit, and could not have succeeded without the incredible help and insights from Devon and Gim.

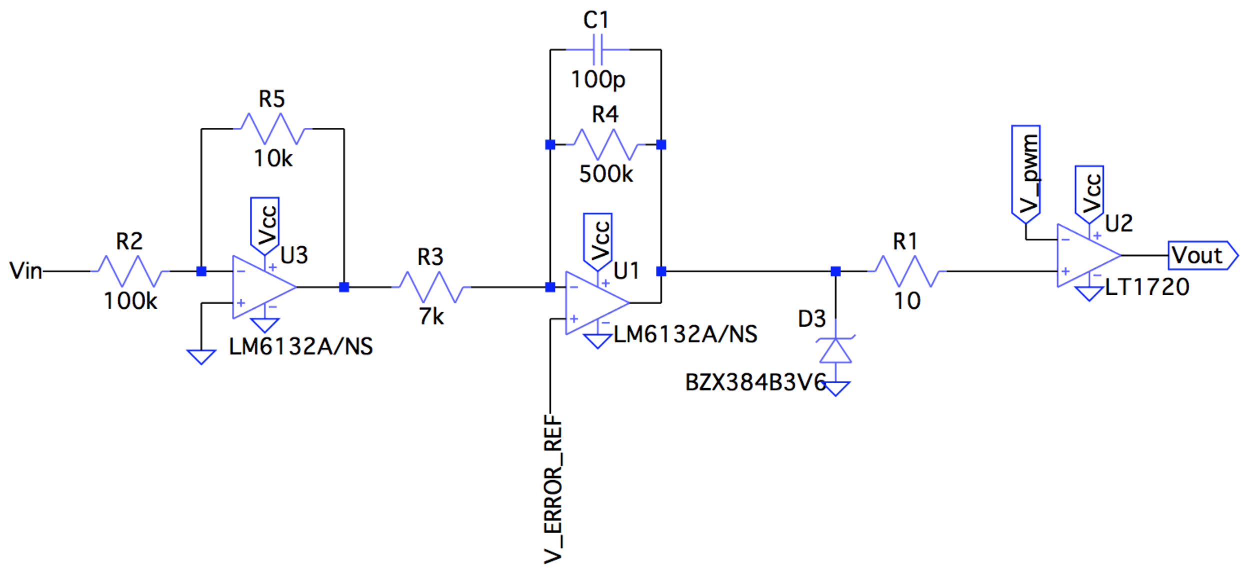


Figure 4.3: Schematic of error amplifier

[1] http://www.ti.com/lit/an/slva059a/slva059a.pdf

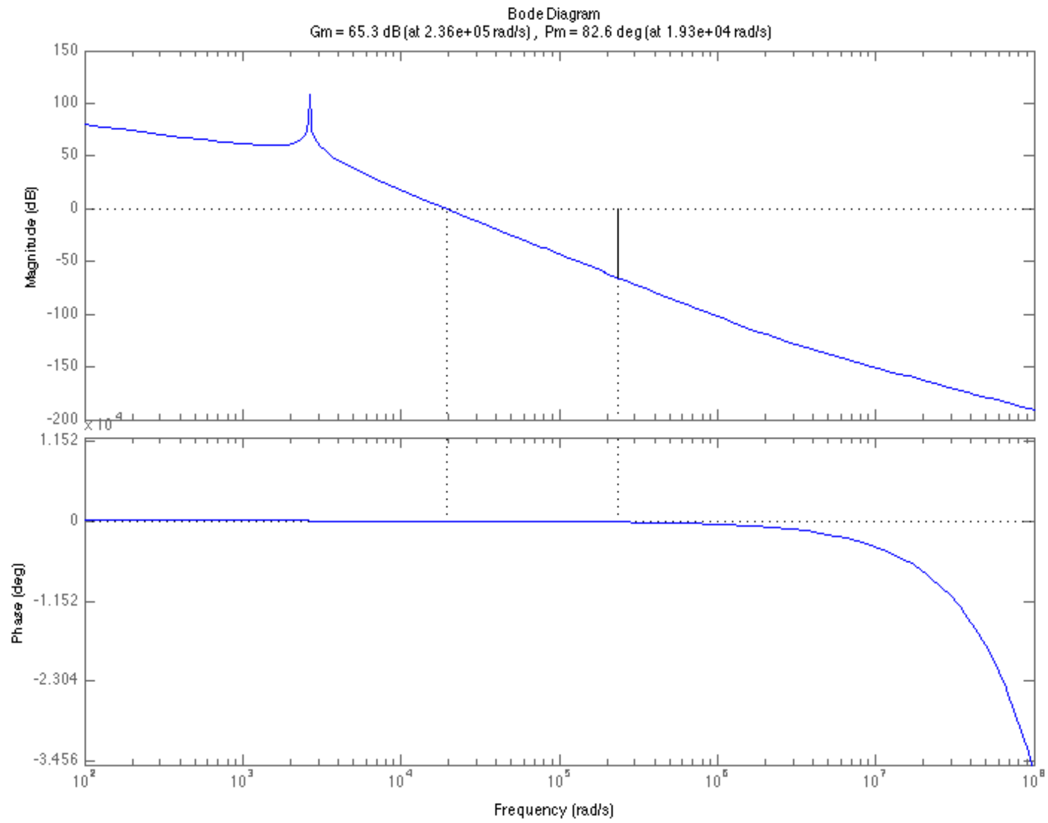


Figure 4.4: MATLAB simulation of control circuit stability

The first piece of the error amplifier is an inverting amplifier with a gain of 0.1x. Buck-boost converters invert their output voltage, so the inverting amplifier switches it back to a positive voltage for comparison. This attenuated signal gets compared against a 0.5V reference in the second op-amp stage. When the buck-boost is at the target 5V output, the voltage at the terminals of the second op-amp should be just about 0.5V. The RC negative feedback provides compensation for our error signal and helps stabilize our output.

The sizing of the compensator values was admittedly a bit of guesswork, and the chosen values are actually remnants of a previous attempt at an error amplifier circuit. As it turns out, after working through the math with Devon, these values are quite stable with the new design. *Figure 4.4* shows a phase margin of 83 degrees when running a MATLAB simulation of the feedback control, which is a comfortable margin. Sometimes you just get lucky.

One thing to note in the design of the error amplifier is the importance for its output to remain within the voltage bounds of the ramp signal. If the error signal exceeds the ramp, then the buck-boost will be permanently switched on. This causes the inductor to charge up but never release energy into the capacitor. Similarly, if the error signal falls below the ramp, the switch will be off and no current flows. Only when there is switching will the buck-boost do anything useful.

To prevent overshooting, D3 is placed as Zener diode chosen to limit the error signal from exceeding the Zener breakdown voltage. It was chosen to be roughly 3.6V, since when imposed on a 4Vpp ramp signal enforces a maximum duty cycle of roughly 90%.

Undershooting is addressed through the level shifter (explained in 4.3), where the ramp signal's lowest voltage can be adjusted down to zero, so that the error signal is always greater than or equal to the ramp.

The last piece of the error amplifier is a simple low-power comparator that compares the error signal to the ramp. In building the design I used a TLC3702 rather than the LT1720 in Figure 4.3 (the LT1720 actually wouldn't work in real life because it cannot take a 12V supply voltage), although the standard LM311 would also work here, if the higher current draw isn't a concern.

4.3 Ramp Generator & Level Shifter

To generate the PWM's ramp, a low-powered 555 timer was configured for a 50% duty cycle at 80kHz. The ramp output of a 555 is taken from its capacitor, which means that it varies between ⅓Vcc and ⅔Vcc. The supply voltage was chosen to be 12V, giving the ramp a voltage swing from 4V to 8V. Interestingly, I found that between the low-power and standard 555s I tried, the low-power ones behaved much closer to the ideal 555 model. The standard 555 had an output between 3V to 8.5V, whereas the low-powered chip was nearly identical to ideal simulation.

Preventing the error signal from undershooting the ramp is easy if the ramp's lower bound is 0V, so we level shift the ramp down to have a swing from 0V to 4V. Level shifting simply involves a subtracting op-amp and a 4V reference.

Once built, the level shifting circuit turned out noisier than I'd hoped, and I'd consider replacing it with an op-amp based oscillator going forward, to avoid having to level shift in the first place. The noise was not a limiting factor of the design, but it's not ideal. The 555 and level shifting circuits are straightforward however, which is important when you're on the clock for producing a working final project!

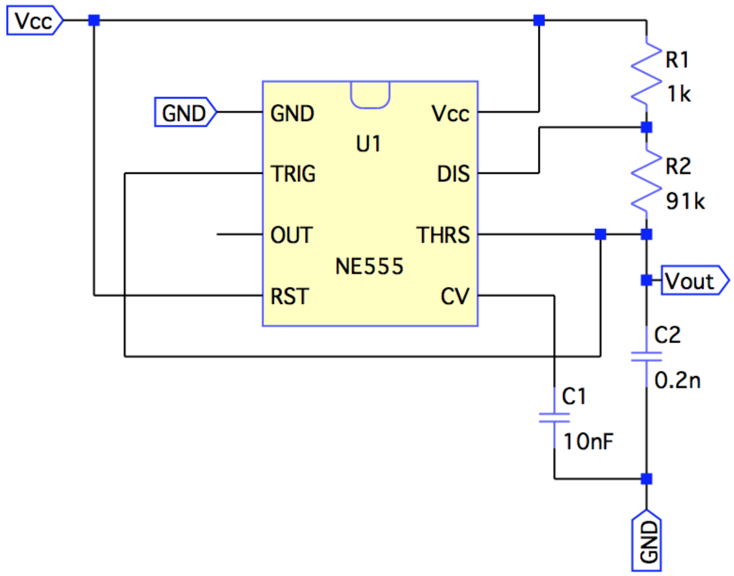


Figure 4.5: 555 timer triangle wave generator. Outputs with 80kHz, ~50% duty cycle

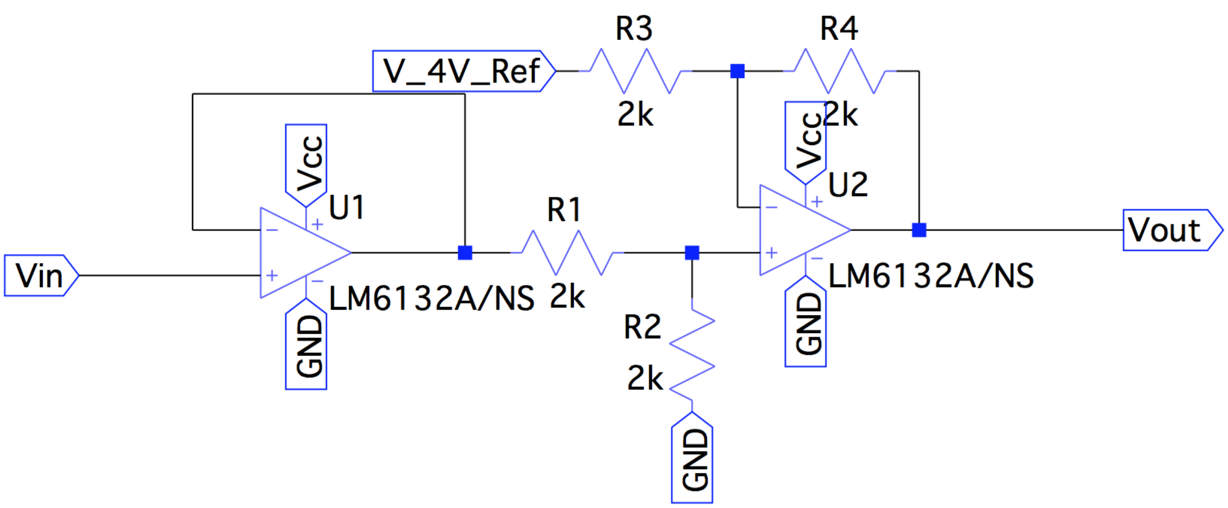


Figure 4.6: Level shifter as a buffer and subtracting op-amp

4.4 Gate Driving

The buck-boost requires a high-side gate driver because of the MOSFET's position in the circuit. I had attempted to use the LTC4440-5, which worked well in simulation, but the only available package was MSOP which required soldering onto DIP adapters. This was fairly tedious, and not recommended for time-pressed 6.101 students. It's a lot easier to work with DIP packages while breadboarding and when you're not 100% certain the part will work. As such, I melted all of my LTC4440-5s almost immediately after I soldered them to the adapters because of a series of sloppy assembly mistakes.

As a replacement, I used the IR2125 gate driver. Operation is fairly straightforward: the output from the comparator is fed into the IR2125's input. The IC uses a 1uF bootstrap capacitor and diode to push the output to Vcc + Vgs, so that a high-side MOSFET can be switched on.

Had I more time, I would try replacing the IR2125 with a transistor-based gate driver as an attempt for lower power consumption. The IR2125 is also out of production, and a design with common transistors would be more sustainable.

4.5 Voltage References

I used an LT1073-12 boost converter IC to create the supply voltage within my circuit. The circuit for operating it was straight from one of their reference designs.

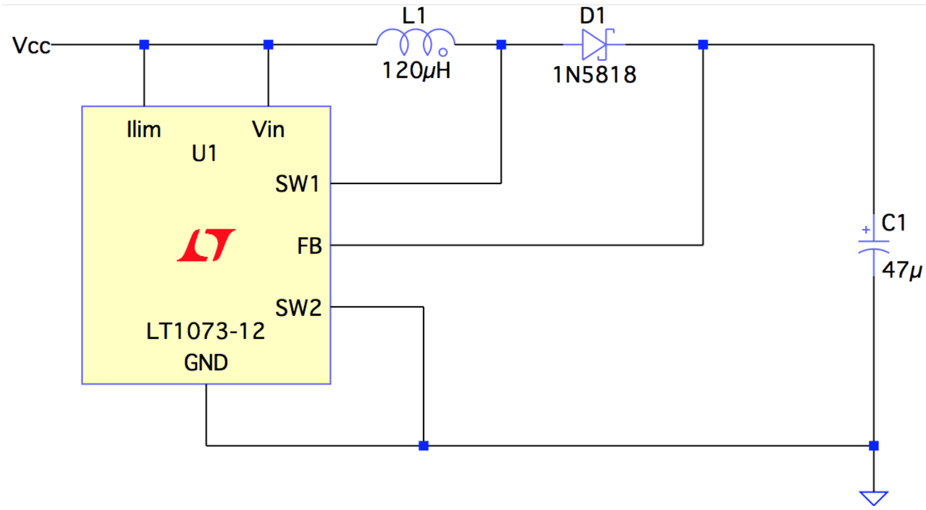


Figure 4.7: LT boost converter reference design

I also had to create a ~4V reference for the level shifter, and ~0.5V reference for the error amplifier. I used a 5V shunt regulator and two buffers with resistor dividers to create these values. I added trimpots on each divider so I could tune my output to exactly 5V.

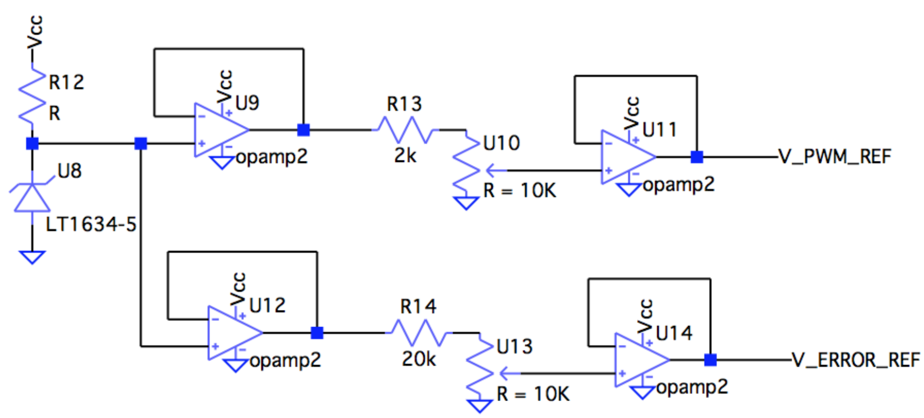


Figure 4.8: Voltage references from shunt regulator

4.6 Results & Reflections

Considerable effort was put into getting the converter to work in simulation before building. Although simulation results should be taken with a grain of salt, they give a decent approximation of real-world behavior, and it seemed foolish to build a circuit that was absolutely not working in simulation. In the end, I think this was the right approach for my project.

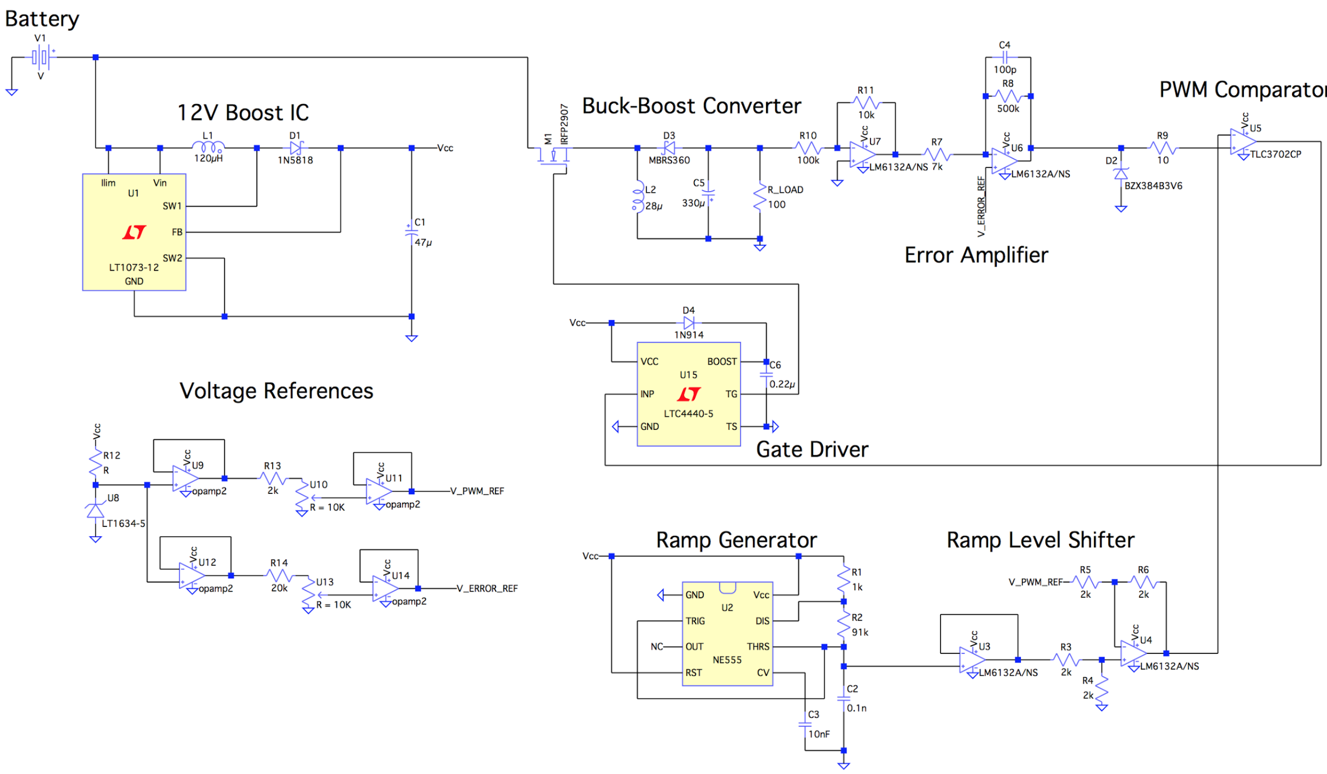


Figure 4.9: Full schematic of the USB power converter. The gate driver was eventually replaced with an IR2125

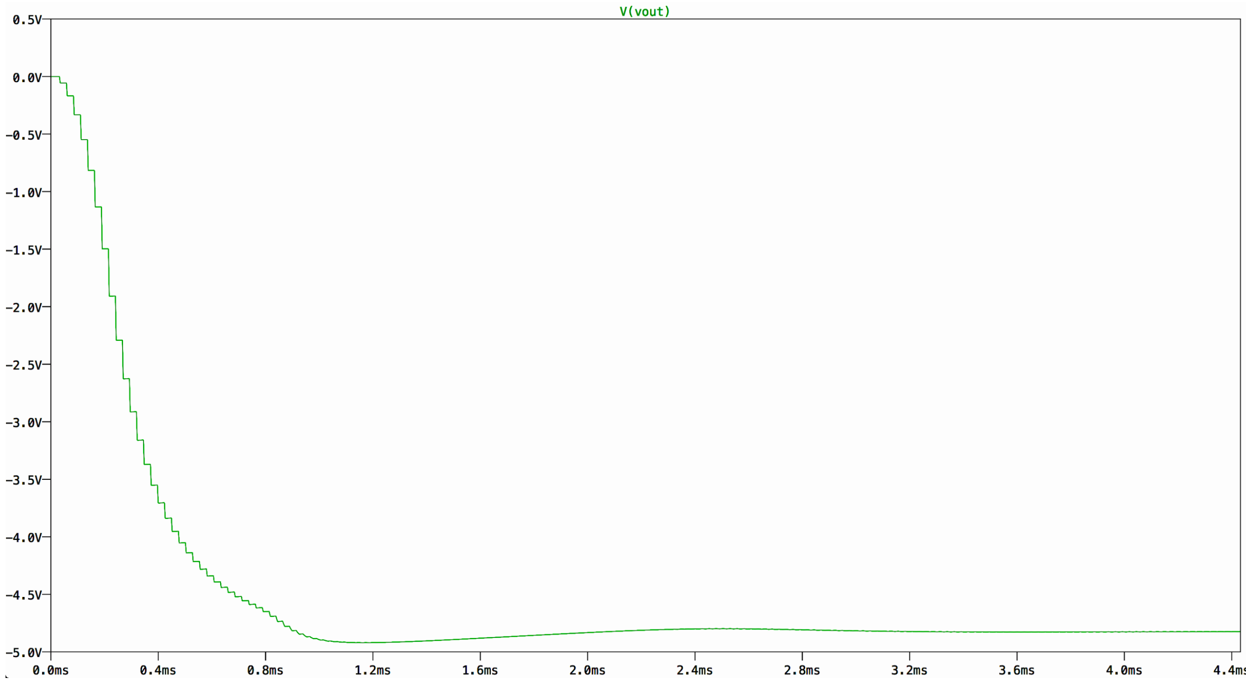


Figure 4.10: Simulation of the converter into a 10 Ohm load. Note that the output howevers slightly off from the ideal -5V. This could be fixed by adjusting the reference voltage into the error amplifier.

One part of the project had to be immediately scrapped: powering the circuit off of the LT1073-12 boost chip. With an input voltage of 1.5V, it cannot push more than 12-15mA. While I had specifically chosen low power components, the power draw from the active devices was enough so that the boost IC's voltage sagged to 8V. I didn't have the time to measure the actual current draw from each device, so I'm not sure which component was most responsible--from the spec sheets, I had anticipated the current draw to be around 8mA.

Rather than replacing the LT1073 with another chip or boost circuit, I decided to focus my efforts on reaching a functioning buck-boost circuit when the devices were powered from a 12V benchtop power supply. The more time I spent with the project, the more I realized that the feedback control was the critical component and most interesting part of the design, and that self-powering from the battery was more of a "nice to have" feature in the time given.

The actual outputs of my circuit were strongly dependent on the load. With a 100 Ohm load (50mA), the converter was able to get a steady 5V output from inputs in the range 3.5V-18V. With a 20 Ohm load, (250mA) it managed 5V output from 4V-18V, and with a 300 Ohm (17 mA) load it produced 5V for the full voltage range of the kit's power supply, 1.2V-18V. With heavier loads and low input voltage, the output was never able to reach the 5V target, as the converter wasn't able to produce enough current.

Had I more time, the first change I would make would be replacing the inductor. There was a strange effect I observed when the powered the buck-boost directly from a function generator: when I bumped the duty cycle up to 80%, the voltage would initially surge high as expected, but then it would quickly decay back down. This looked like it could potentially be the inductor saturating and therefore unable to supply enough current. The inductor I used was rated for 1.7A which should have been sufficient, but the behavior I saw made me question this rating. This would also help explain the converter's failure to provide the anticipated current.

The second change I would make would be revising my layout. Switching converters are very noisy, and my layout wasn't optimized to keep leads and connections as absolutely short as possible. Better performance would be made with more thoughtful layout.

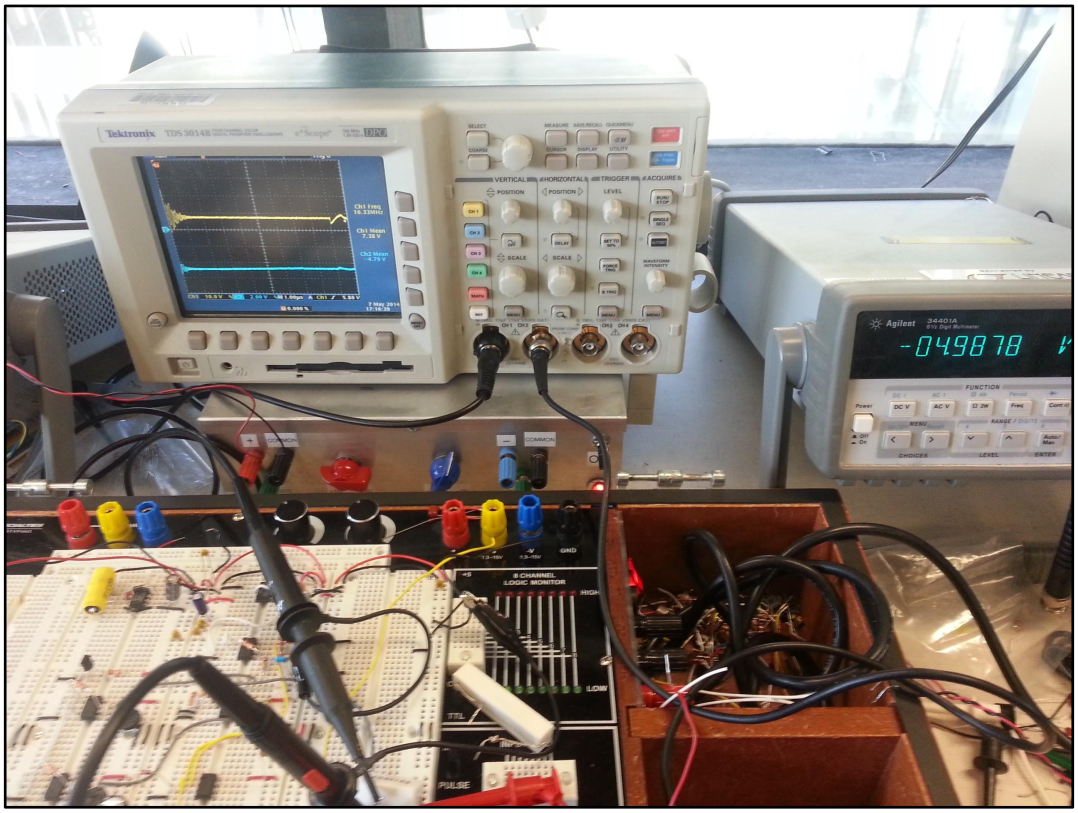


Figure 4.11: Proof that my circuit, could in fact, output 5 volts.

Beyond these immediate changes, more thought would also have to be given to powering the circuit itself. It would be interesting to explore a start-up circuit that provided just enough power for the converter to get going right when a battery is connected. Once it's powered up, the start-up circuit could shut off, and the power converter's output could be used to power the active devices.

I would also consider dropping the supply voltage down to 5V. The ICs should all be low power, so requiring them to have a supply voltage range up to 12V is a counterproductive goal and limits parts selection. This change becomes even more obvious when the motivation behind a 12 volt supply came from using the LT1073-12, which wound up not being able to supply enough current anyway.

The control theory around switching converters proved to be monumentally more complex than I had anticipated. Most block diagrams of schematics of switching designs I had seen typically had a simple op-amp as the error amplifier, which mislead me into thinking that the design would be straightforward. I had no idea what I was in for in terms of selecting the right configurations and compensator values to make the error amplifier actually behave as it should. This became particularly true when in an earlier design, my error amplifier always increased and decreased its output in the right direction, but the converter's output was never quite right. The problem was a lack of gain, and admittedly the concept of gain in a switching circuit is still a bit mystifying.

Gathering the right parts proved tricky and occasionally stressful. Because I was trying to target very low power consumption, I ordered several ICs from TI and LT. The shipping on these was typically slow, being at least 4-5 business days. My Bill of Materials was continually evolving as I refined my simulations, so I wound up receiving many parts that I never used, and using parts that arrived in the last week of the project. An additional week wouldn't have hurt!

This project was challenging and enlightening. I learned a considerable amount about power conversion, parts selection, simulation, layout, and generally how to piece a circuit together. Although the converter's current was less than I envisioned, I did feel immense satisfaction when I could adjust the kit's supply voltage from 1.2V to 18V and see a constant 5V on the output (for a 300 Ohm load). I would like to thank Gim and Devon for the countless hours they spent helping me troubleshoot my design and walking me through the theory I didn't know I didn't know, and Joe and David for being great teammates with their design insights and moral support.

5. Timeline (Joseph Driscoll)

The project was completed over a period of 5 weeks starting April 7th. Figure 5.1 consists of a Gantt chart displaying the stages of our project on a scale of weeks. We designated 1 week to designing our project, 2.5 weeks to build the circuits, 1.5 weeks to test them, 1 week to integrate our modules, and finally 1.5 weeks to write our report. The goal was complete our circuits by April 27th. We were thankful we set this early deadline, because some of the requested parts arrived late.

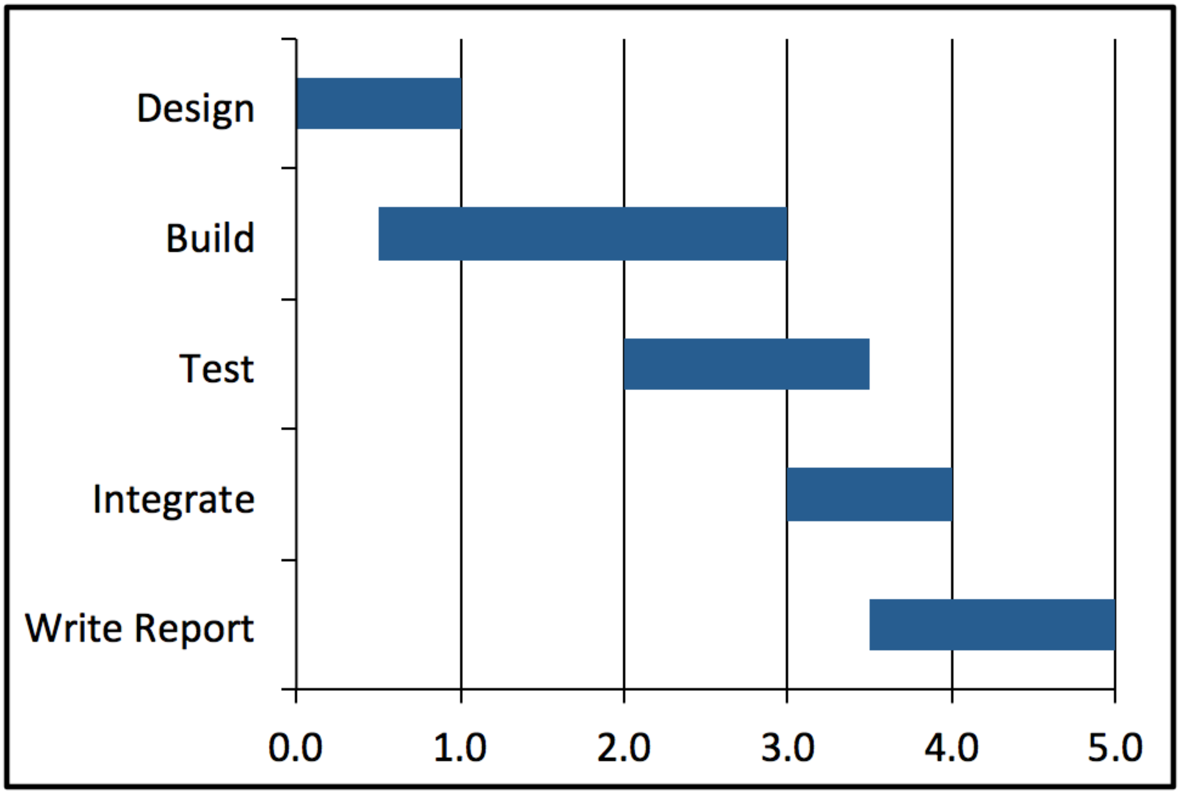


Figure 5.1: A Gantt chart of project stages in weeks.

6. Conclusion (Paul Hemberger)

In the end, we succeeded in building three power converters to provide power in a variety of applications. They are modular, in that each piece is distinct and functional on its own, and can be connected so that each converter can provide the power for the next stage. Almost all of our design goals were reached, and regardless of actual performance, all of us learned a substantial amount about designing and troubleshooting.

We hope that our report is informative of the insights we gained and mistakes we made along the way, and helps future 6.101 students choose, refine and finish their own final projects.

7. Works Cited

"Basic Electronics Tutorials and Revision." *Basic Electronics Tutorials*. N.p., n.d. Web. 1 Apr. 2014.

"Brett's Down Home Page." *Brett's Down Home Page*. N.p., n.d. Web. 12 Apr. 2014.

"Circuit Collection." *Circuit Collection*. N.p., n.d. Web. 2 Apr. 2014.