ELEC3106 Design Project Report

By Riley Dean (z5308666)

Circuit Design

The circuit design for the project can be separated into three smaller subsystems: Power supply, switch latching, and constant current supply.

Figure 1: Complete Circuit Diagram

Power Supply

The purpose of the power supply in this design is to create a “smooth” 5 Volt supply from the given battery supply, which is done in two stages: A boost converter, followed by a 5V Linear Regulator.

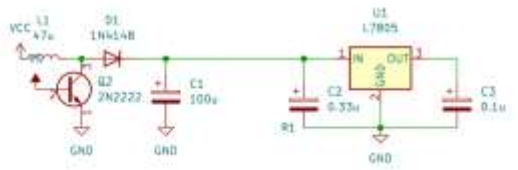

Figure 2: Power Supply Circuit

Firstly, the boost converter was designed so that it would produce an output voltage of 7V, hence being above the dropout voltage of the linear regulator. In this design, a 2N2222 NPN transistor was used for switching because it was determined that a MOSFET, while useful for switching, would either not operate at the low input voltage supplied (1.2-1.5V), or have a high value for $R_{DS}(on)$, causing greater power dissipation, making the system more inefficient. The 2N2222 was chosen due to its availability and its maximum Collector-Emitter voltage being 40V, which is well above the expected output of the converter. What wasn’t expected was the magnitude of the collector current was greater than the rating of the device, which dramatically shortened the life of the device, and prevented the boost converter from working as effectively as it could be.

For the choice of inductor, the following equation was used:

$$L = \frac{V_{in} \times (V_{out} + V_D - V_{in})}{\Delta I_{ripple} \times f_{sw} \times (V_{out} + V_D)}$$

With the value for ΔI_{ripple} being estimated as 20% of the inductor current, which can be calculated by the equation

$$I_L = \frac{V_{out} \times I_{out(max)}}{V_{in} \times \eta}$$

Which, with estimations of power efficiency and load current, can be used to estimate the value of the inductance required. As can be seen in the equations, there is an inverse proportionality between load resistance and inductance, suggesting that the smaller the inductor value, the better the converter will be in responding to load resistance changes, Due to this, an inductor value of 47µH was chosen. The capacitor value was chosen to be large enough so that it discharged slow enough that while the switch was off, the output voltage would not drop greatly.

The duty cycle of the signal generator was chosen according to the equation $D = 1 - \frac{V_{in}}{V_{out}}$, giving a value of 83%.

The output of the boost converter can be seen in figure 3, where it is worth noting that the noise caused by switching creates large peaks in voltage.

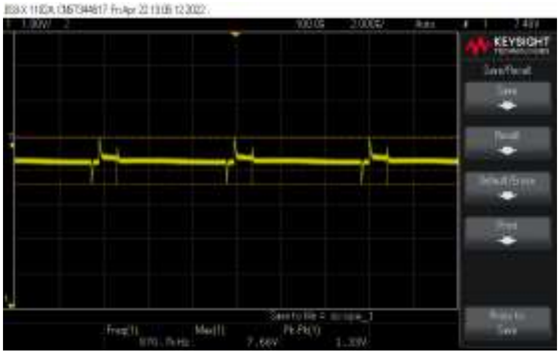

Figure 3: Boost Converter Ripple

In terms of the linear regulator, the L7805 was used in order to give a steady output voltage for the rest of the circuit to operate under. In the datasheet for this component, the circuit to be used, including component values, is supplied, with the capacitor formation used to filter out noise generated by the regulator.

Overall, the output of the power supply can be seen in figure 4, where it is worth noting that the output voltage swings caused by the boost converter were not handled as well as expected by the linear regulator and its filtering capacitors. As the constant current output required a constant 5V supply, this ripple caused problems throughout the rest of the circuit. In future iterations, the ripple from the output voltage could be removed through the use of an RC Low Pass Filter.

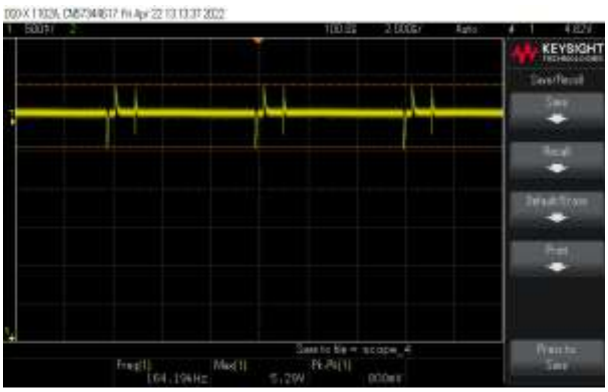

Figure 4: Linear Regulator Ripple

Pushbutton Circuit

To create a pushbutton switch circuit that acted as a toggle, two options were considered, a T flip-flop, and a latch.

With the use of a flip-flop, this was a more complex option, as it required the use of 6 NAND gates, as well as requiring an input clock signal, which, in this project would have been the same as the one used for the boost converter. The magnitude of this clock signal is below the minimum voltage of the 74LS series of ICs, which is 2V, and hence, this was solution was considered to be not ideal.

A latch solution involves using two Schmitt triggers, with feedback between the output and input. Due to the RC charging circuit with the 220k resistor and 47n capacitor, this feedback allowed for the circuit to retain the current state of the circuit for as long as the capacitor was charging or discharging based on the equation $\tau = RC$. This RC circuit also acts as a debouncing circuit for switch, hence its connection after the first Schmitt trigger, allowing for it to filter out the high frequency spikes in the signal. It is also worth noting that the 5V supply from the linear regulator was used to supply the 74LS14 Schmitt Trigger IC, as this was within the 2-7V input voltage range of the circuit.

The output of the switch was connected to the base of a transistor which would turn on or off the current mirror. Due to this, when the circuit was off, there was a slight decrease in the load resistance of the circuit by around 100 Ohms, increasing the boost converter output voltage, however, due to the linear regulator, this change was not observed to affect any other part of the circuit.

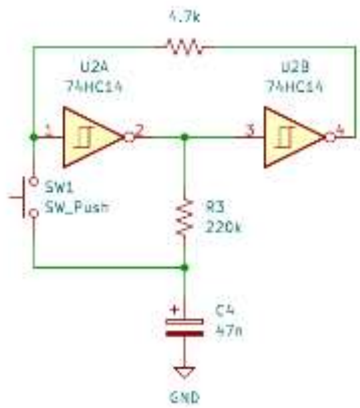

Figure 5: Latching Circuit

Current Mirror

A current mirror was chosen for the constant current supply, as this is a device that is able to “mirror” the current passing through one BJT into another BJT. This is used because it allows for the LED to be changed to one of a different colour, and hence, forward voltage without changing the current passing through the LED. A current mirror is dependent don the current amplification factor β however, but the process of finding two transistors that exactly match is very difficult, meaning that the resistor values of each emitter vary slightly, so that the current passing through each matches. To get a constant current of 20mA, the voltage in the loop from the 5V input, through the switching transistor, and then through the input branch of the mirror, which can be seen to have a value of slightly over 100 Ohms. As 100 is a standard value, this was chosen. From there, a range of value were tested with a multimeter to determine what value of resistor was correct for the right-hand side of the mirror, with 82 Ohms being the closest standard value found.

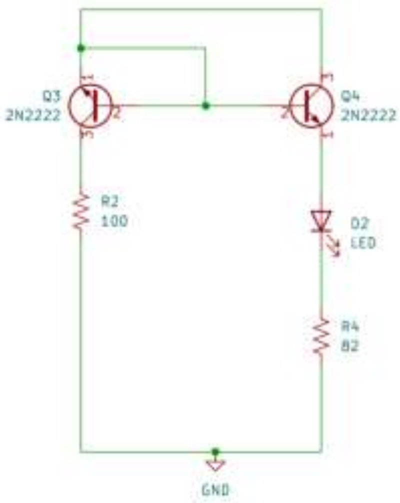

Figure 6: Current Mirror Circuit

As alluded to before, the ripple in the power supply means that the output current measured through the LED was not constant, as seen in figure 8. It is worth noting that the ripple is noticeably smaller, however it is still large enough that it drastically changes the value for current seen.

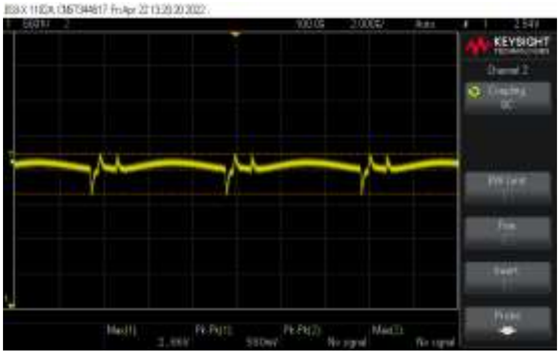

Figure 7: Output Current Ripple

Simulation

To verify the functionality of the circuit, an LTSpice schematic was used, allowing for the verification of the design’s base functionalities. As seen in the figure below, the circuit was constructed analogous to the physical circuit, however the switch was replaced with a voltage source, as this made deciphering results easier. At this stage it is worth noting that the BJT used is not a 2N2222. This is because the collector current was too great for the 2N2222 as talked about before, however availability of components is not a challenge with LTSpice, so this could be replaced.

Figure 8: LTSpice Simulation Circuit

By simulating the circuit with the conditions above, the output of the boost converter and linear regulator can be seen.

Figure 9: Power Supply Voltages

Figure 10: Param Sweep of R Values

To determine the resistor values required for the current source, a step command can be used, allowing for a range of resistor values to be looked at efficiently. The command used for this sweep was *.step param X 50 150 10* where *X* was the assigned value for R2. In the graph below, each resistor value caused a decrease in the value for current, hence showing that when R2 = 140 ohms the current is closest to 20mA. The closest standard resistor value to this is 150 ohms, so this would be the most relevant to this simulation. It is worth noting however that when simulating transistors, they will have a consistent value for β, which means the specific resistor values found here are nothing but a guide for the implementation of the final circuit design.

Power Efficiency

The LTSpice simulation allows for a useful measurement of the power efficiency of the circuit, as the power input into the circuit by the battery can be estimated, and so can the output power used by the LED. By completing a simulation of the circuit with an LED with a forward voltage of 0.7V due to the available Spice models all having this property, it can be seen that $P_{in} = 418.72mW$ and $P_{out} = 12.018mW$, giving a value for $\eta = \frac{12.018}{418.72} =$ 2.87%. This is a very small value, with several factors influencing this.

The first factor to consider is that the LED forward voltage is quite small, being only 0.7V. LED forward voltages can range all the way up to 3V for a white LED, and the power efficiency of this circuit is proportional to the forward voltage of the LED, meaning the efficiency could go up to 12.3%, however this is still very inefficient.

The other considerations are related to the efficiency of the circuit. The combination of a boost converter and a linear regulator can be highly inefficient, due to the dropout voltage of the linear regulator caused by the internal design of these ICs. In the simulation, the power efficiency of the 7805 is $\eta = \frac{192.9}{357.52} = 53.9\%$, which is higher than most linear regulators, due to the fact that it is driven at just above 7V, which is the closest you can get to the dropout voltage while not affecting the functionality of the regulator, however it is still not efficient. From there, the circuit also has four BJTs, with the switching BJT for the boost converter having a particularly large current passing through due to its proximity to the inductor, which will also adversely affect efficiency. Regarding the BJTs, there will always be a voltage drop across them of at least 0.7V when in saturation, which in this circuit, they will all be, further contributing to inefficiency. The last aspect of note is that the current mirror effectively halves power efficiency, as it will draw double the current required through one branch to create the "mirroring effect," which was utilised in this circuit.

The figure below illustrates the power going through the linear regulator and the overall circuit.

Figure 11: Power of System

Improvements

There are two major improvements that could be made to the physical circuit to improve the overall effectiveness of this solution: Replacing the switching BJT, and creating a more efficient power supply.

As covered in earlier sections, the collector current permissible in the 2N2222 is too low, and a solution with greater than around 1.5A should be used for safety. A short search on any reputable electronics supplier shows a range of options, but a potential option is the TIP31C power transistor by STMElectronics, which has a range of properties that make it applicable to this circuit.

Table 1. Absolute maximim ratings

Symbol	Parameter	Value	Unit
V_{CBO}	Collector-base voltage ($I_E = 0$)	100	V
V_{CEO}	Collector-emitter voltage ($I_B = 0$)	100	V
V_{EBO}	Emitter-base voltage ($I_C = 0$)	5	V
I_C	Collector current	3	A
I_{CM}	Collector peak current	5	A
I_B	Base current	1	A
P_{TOT}	Total dissipation at $T_{case} = 25°C$ Total dissipation at $T_{amb} = 25°C$	40 2	W W
T_{stg}	Storage temperature	-65 to 150	°C
T_J	Max. operating junction temperature	150	°C

Figure 12: TIP31C Maximum Ratings

Lastly, in any implementation of this design, a boost converter would be required, however the linear regulator would not necessarily need to be. If it was to be kept, an RC filter would have to be used to filter out noise as covered previously. Otherwise, more comprehensive solutions could use a boost converter with feedback to allow for voltage control, however the theory required for this is out of the constraints of this course, or alternatively, using a Zener diode regulator to simplify the circuit could also be useful.

Overall, the design project circuit provided a reasonable solution to the given specification, however modifications to the power supply would be required to generate a comprehensive solution.