Expertise
and insight
for the future

Amit Pandey

Design of a 5 A Constant Current Source

Metropolia University of Applied Sciences

Bachelor of Engineering

Electronics

Bachelor's Thesis

12 May 2021

Author Title Number of Pages Date	Amit Pandey Design of a 5 A Constant Current Source 34 pages + 6 appendices 12 May 2021
Degree	Bachelor of Engineering
Degree Programme	Electronics
Instructors	Matti Fischer, Principal Lecturer Jani Virtanen, Owner, MDS Finland

This work presents the development of three different current source circuits providing constant 5 A output. The purpose of this project is to provide enough thermal power to cut surgical sutures of lifeline thread.

This paper gives a theoretical framework of a power source and different topologies to develop the constant current source. Websites, books, journals, magazines, etc. were used as sources. PADS Logic and PADS layout were used to design the circuit. Circuit simulation was done in multisim. Tests were conducted mainly using a multimeter.

PWM was used to control the main circuit's output. The output of 5 A current was quite enough to cut the sutures in less than 4 seconds. At the end, the circuit was enclosed inside the PCB enclosure before delivering. Short circuit protection, development of both cable and tip, high current for a shorter time and development of circuit using switching regulator are the goal of the future development.

Keywords	Current Source, Linear Regulator, Switching Regulator, PWM

Appendices

List of Abbreviations

AC	Alternating Current
C	Capacitor
DC	Direct Current
I	Current
I_{AR}	Avalanche Current
L	Inductor
Op-Amp	Operational Amplifier
PWM	Pulse Width Modulation
PCB	Printed Circuit Board
R	Resistor
R_{DS}	Drain – Source on resistance
SMPS	Switch Mode Power Supply
θca	Thermal Resistance from the case to ambient
θjc	Thermal Resistance from the junction to case
V	Voltage
VBE	Base-to-Emitter Voltage
VZ	Zener Voltage

List of Figures/Tables

1 Introduction

Circuit designing is to solve the need of real-world using components like capacitors, inductors, resistors, diodes, op-amps, and wires. In the modern world, thousands of electronic devices consist of the circuit. Especially in the medical field electronics designs are being more vital to saving human lives as well as to live a healthy life. Equipment such as MRI, CAT, and other many tests all depend on electronics.

This work is also done for medical application purposes. This device work as a supplement device while inserting an implant inside the heart. For this high thermal power to the tip of the wire is needed for the definitive amount of time so that it can be used to severe medical suture lines which pass through the catheter to the valve. Therefore, the goal of the thesis is to create a reliable pulsing current source with a 5 A output for medical purposes. The final expectation is to create a power source capable of delivering high output current, generating the necessary thermal power to sever medical suture lines. In addition, the additional feature should be added to the system to make it easy to operate. For example, making it work with PWM, adding push-button, adding a potentiometer to change current output time and as well as preventing short circuits while using in real life. A further aim is not to have one, but rather to try many circuit design topologies with different methods and test them.

This thesis provides some brief theoretical background on the subject but focuses mainly on the design of the circuit. Multisim is used for simulation purposes. PADS Logic and PADS Layout are used for the design purpose. Afterward, circuit was printed using a milling machine. In the end, the circuit was mounted and wired inside the electronics enclosure as well as switches, connectors, led, and buttons are mounted outside of the box. Tests were measured by using multimeters. A bench power supply was used to test the circuit with different inputs.

2 Power Sources

The design of the circuits begins with some specification, which states the functionality of the finished design must provide. In general, a designer should be focused on what to achieve rather than by which means to achieve. Designing the circuits always should start by designing on paper (simulation or schematic software) then followed by the cost of the materials, verification, and testing, prototyping, results, and documentation. [1.] Before designing any circuit, it is vital to go through the sources. In electronics designs, types of circuits are taken into consideration: Current Sources and Voltage Sources.

2.1 Ideal Voltage Sources

The ideal voltage source is a two-terminal black box that maintains a fixed voltage drop across its terminals, regardless of the load resistance. It states, it must supply the current according to this formula: $I = V \div R$. But it is always good to remember that the real voltage source can only provide a finite maximum current. A battery can be taken as an example of a voltage source. A voltage source “likes” an open circuit load and “hates” the short circuit load. [2.] Figure 1 describes the symbols used to indicate the voltage source.

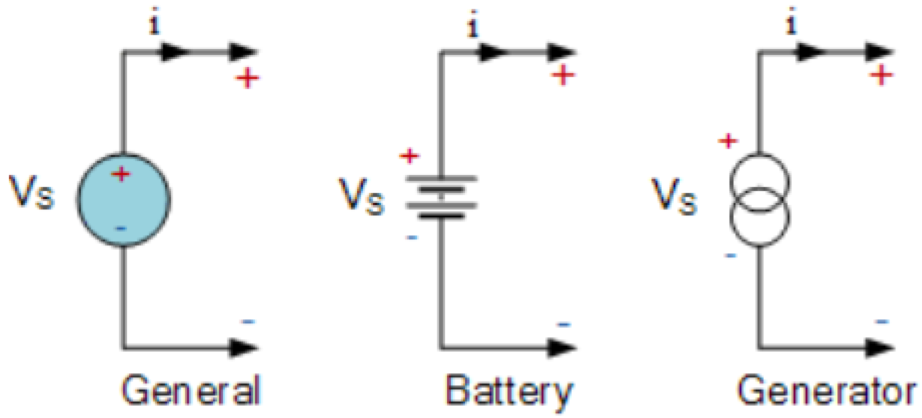


Figure 1. Voltage sources symbols [3].

2.2 Current Sources

An ideal current source is a two-terminal black box that maintains a constant current through the external circuit, regardless of load resistance or applied voltage. So, it should be able to provide any voltage across the load terminal according to the variable resistance value. Designing even simple current sources includes theories of electronics. A current source “likes” a short-circuit load whereas “hates” an open-circuit load. [2.] The symbols used to indicate a current source are shown in Figure 2.

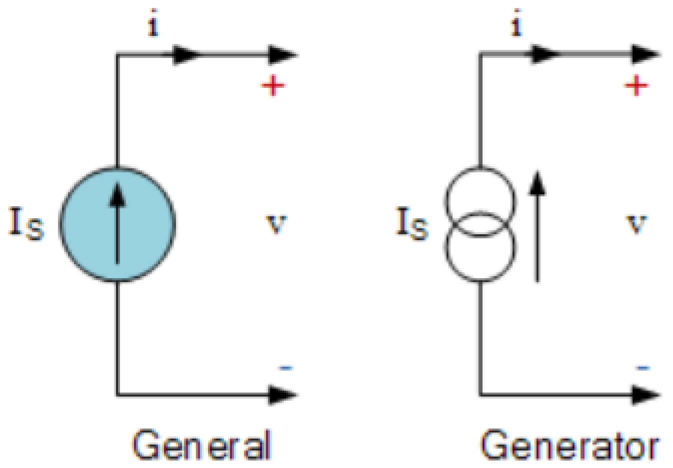


Figure 2. Current sources symbols [4].

2.3 Voltage Regulator

Voltage regulator works as a system providing constant free of noise and disturbance, smooth output voltage regardless of ups and downs of the input voltage. The output from the voltage regulator is independent of the load current, temperature, and AC variation. To sum up, the voltage regulator eliminates/minimizes the variation in voltage to protect the device. There are varieties of topologies/circuits to regulate the voltage some of them are mentioned below. [5.]

2.3.1 Linear Voltage Regulator

It is a system that maintains constant output voltage independent of the input voltage or load. Linear voltage regulator only works for voltage step down. So, the output voltage is always lower than the input voltage. The resistance of the system keeps the output voltage constant since it varies following input voltage and load. It is meant to act as a variable resistor, continuously maintaining the voltage divider network [6]. Switching regulators are more popular due to their efficiency but on other hand, linear regulators are simpler to use. Unlike switching regulators, linear regulators do not produce switching noise. Linear regulators can further be classified in shunt type and series type which are explained below.

2.3.1.1 Series Linear Voltage Regulator

The series linear voltage regulator can be built using a transistor and the Zener diode as shown in figure 3. Zener diode provides voltage reference therefore it can maintain the almost constant voltage across a load resistor. By adding an amplifier in series with the load, regulation and efficiency of the regulator can be improved. This circuit is also known as a linear series voltage regulator since the transistor is in series with the load. By using the following equation, output voltage can be calculated where V_Z is the Zener diode voltage and V_{BE} is the base-to-emitter voltage of the transistor. [7, p. 661.]

$$V_O = V_Z + V_{BE} \qquad (1)$$

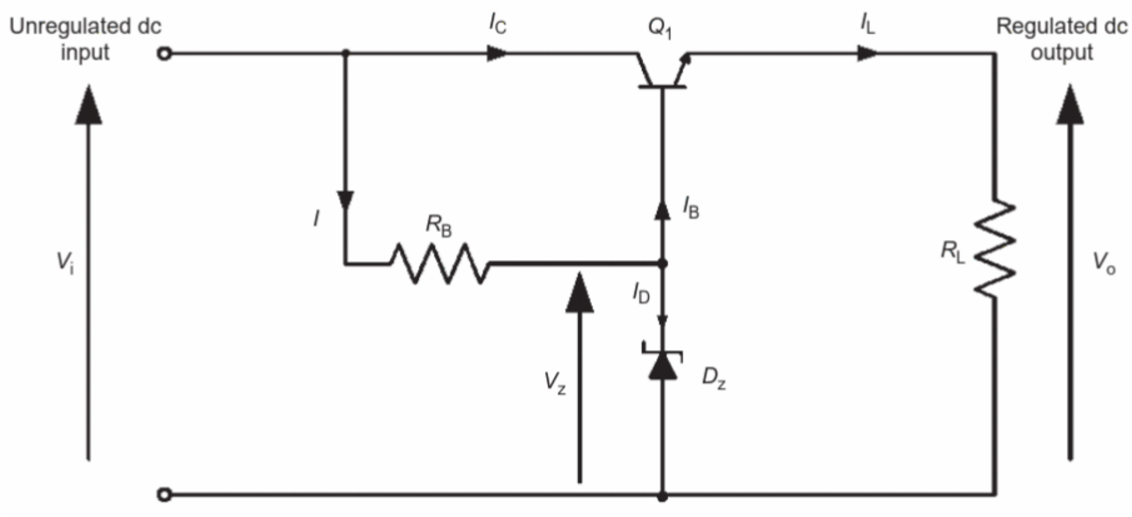


Figure 3. Basic circuit of a linear series regulator [7, p. 661].

2.3.1.2 Shunt Linear Voltage Regulator

In this type of regulator, pass transistor Q_1 is connected in parallel with the load as shown in figure 4. The operation of the circuit is similar to the linear series regulator, but regulation is achieved by controlling the current through the Q1. In this circuit, when the output voltage tries to increase due to the change in the load resistance, the voltage in the non-inverting terminal of the op-amp also increases. This voltage is compared to the reference voltage, and the resulting voltage causes Q_1 conduction to increase. This leads I_L to decrease. Hence, the constant output voltage is achieved. On the other hand, if output voltage tries to decrease due to a change in the load resistance opposite action occurs.

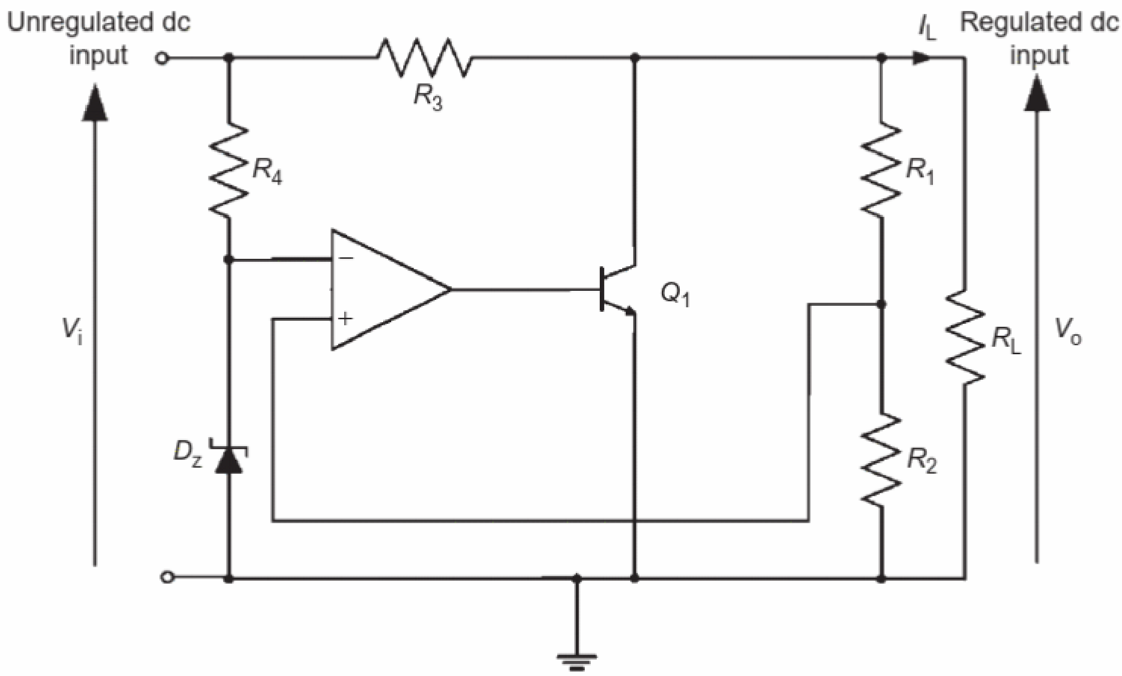


Figure 4. Basic Circuit of a linear shunt regulator [7, p. 665].

2.3.2 Switching Voltage Regulator Circuit

This type of voltage regulator uses a switching element to transform the incoming power supply into pulsed voltage, which is then smoothed using capacitors, inductors, and other elements. Power is supplied from the input to the output by turning on the switch until desired voltage is reached. A high-speed switch is needed to perform this operation. Generally, MOSFET is used as a switch. The benefit of using the high-speed switch is to generate less heat. [8.] Switching voltage regulator topologies are explained below.

2.3.2.1 Boost Type

Boost type converter is a type of converter which steps up the input voltage while stepping down current to the output (load). So, the output voltage is always equal to or greater than the input voltage. Figure 5 illustrates the topology of the boost converter circuit. The working principle of the boost converter is dependent upon four components: inductor, diode, switch and capacitor. When V_{IN} is supplied and MOSFET is on, L1 stores the energy. Then, when MOSFET goes off inductor changes the polarity and V_{IN} + V_L pushes the current through the diode. In this process, the capacitor is charged and when the MOSFET is ON capacitor is discharged. So, the current is passed through the load continuously. The output voltage is dependent upon the duty cycle of the square wave. Formula 2 presents the output voltage in the boost converter.

$$VOUT = \frac{VIN}{(1 - DUTY\ CYLE(D))} \tag{2}$$

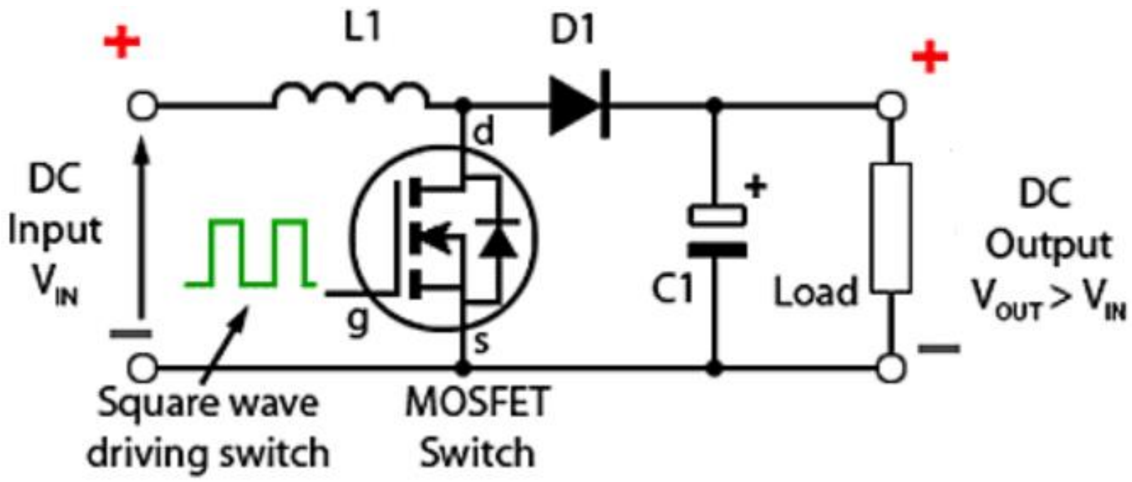


Figure 5. Basic boost converter circuit [9].

2.3.2.2 Buck Type

A buck-type converter is needed when the output voltage should be higher than the input voltage. Because of the high efficiency of the switch-mode regulator, the output current becomes bigger than the input. Figure 6 illustrates the topology of the buck converter circuit. The working principle is dependent upon a high switching transistor (MOSFET) and flywheel circuit. When a switch is on, the current flows from the inductor to load as a diode (D1) blocks current to go through the diode path and in this period, energy gets stored in the inductor. Then, when the switch is off the energy stored in the inductor gets collapsed and polarity gets changed and current flows through load and diode. The theoretical formula is as follows to determine the output voltage in the buck converter. [10.]

$$VOUT = \frac{VIN\ x\ (On\ time\ of\ switching\ waveform\ (tON)}{periodic\ time\ of\ switching\ waveform\ (T))} \qquad (3)$$

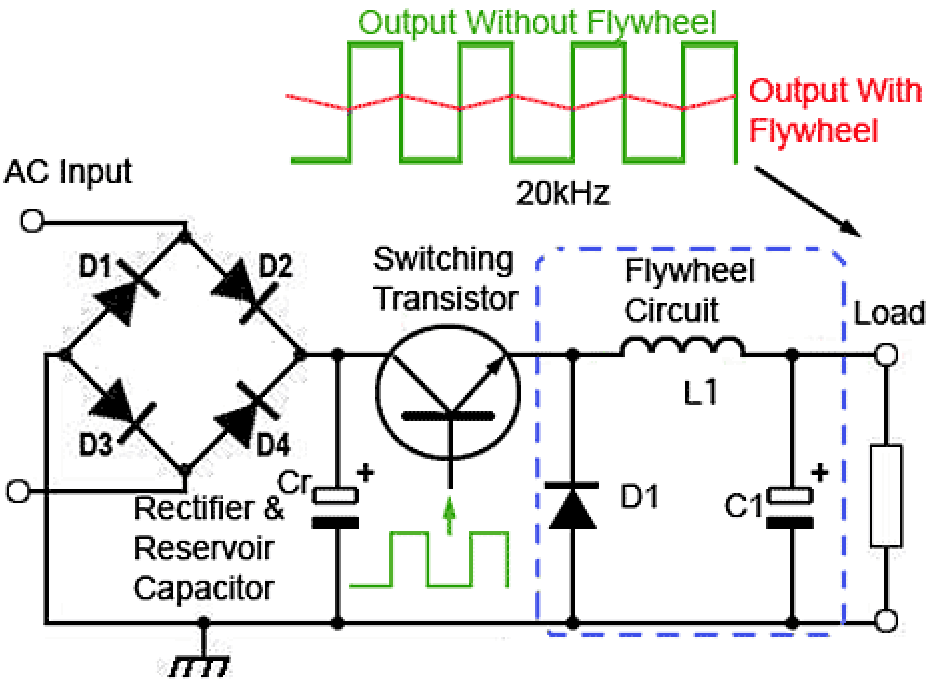


Figure 6. Basic Buck Converter Circuit [10].

2.3.2.3 Buck-Boost Type

Buck-Boost converter is a type of Switch Mode Power Supply (SMPS) that combines the principles of the buck converter and boost converters in a single circuit. Applications that need constant voltage but where input voltage might get changed over time i.e. battery-powered systems, in this kind of systems/applications buck-boost converter becomes ideal. Figure 7 demonstrates the working topology of the buck-boost converter. Equation 4 presents the formula for the output voltage of buck-boost converter:

$$V_O = \frac{-Duty\ Cycle}{1 - Duty\ Cycle} * V_S \qquad (4)$$

Duty cycles vary from 0 to 1. If Duty Cycle > 0.5, the output voltage is larger than the input voltage. And, if Duty Cycle < 0.5, the output voltage is smaller than the input voltage. [11.]

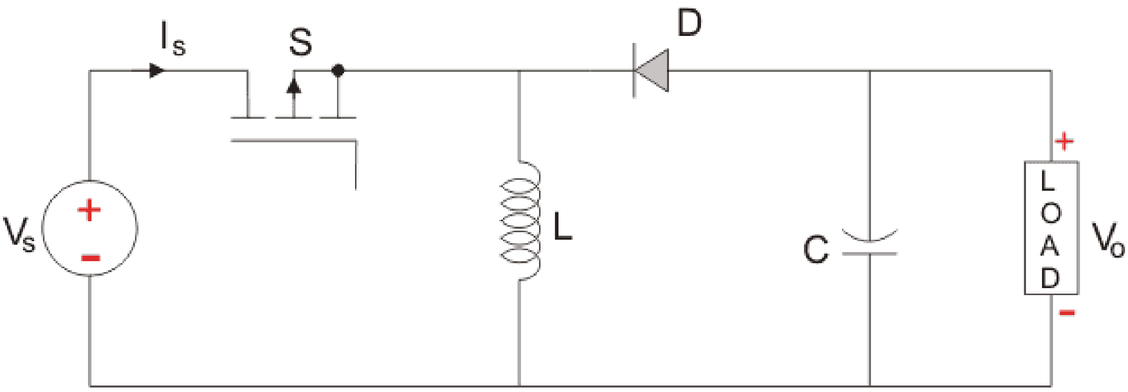


Figure 7. Basic Buck-Boost Converter Circuit [11].

3 Current Source

Based upon the above-explained concept, the following system was designed to provide a constant current source for load with about 1 Ohm resistance. The control/block diagram is shown (Figure 8).

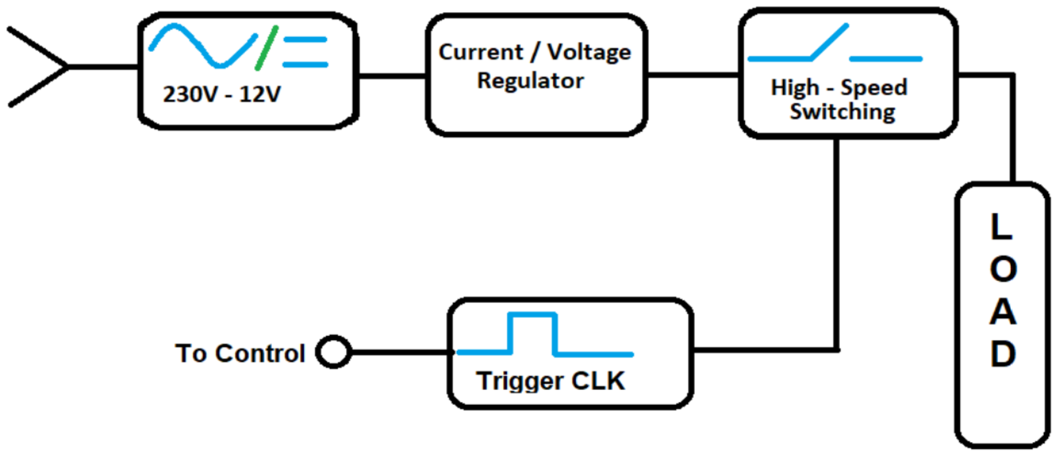


Figure 8. Block diagram explaining the system of the circuit.

As shown in figure 8, the system is divided into five parts. They are power conversion from AC to DC, Voltage/Current regulator, High-speed switching, Trigger CLK and load. Interchanging between the above-mentioned blocks is possible mainly with a switch as different solutions are expected. A short description of each block is given below.

3.1 DC Power Supply

The requirement of the project was providing constant DC. So, some topology that converts 230 V coming from the wall socket to 12 V was needed. Therefore, a power supply of at least 10 A current rating is needed. Although, building the power supply from the scratch was possible but concerning the safety and the available resources, it was decided to rather buy it. The low-cost supply was bought at the beginning for testing. Later, a power supply with good quality and protection was bought in the final stage of the prototyping.

3.2 Current / Voltage Regulator

In electronics, some applications need to have a supply of constant output current which gets regulated by using the current regulator circuit. The current regulator circuit is mainly dependent upon variation in voltage and current to supply constant output current. As Ohm's law states $I = V \div R,$ to gain the constant output current voltage and resistance in the output should be constant or the value of one gets adjusted as another value gets changed. Design of the current regulator circuit is done standardly by using IC-based voltage regulators, transistors, op-amps and Zener diodes. This can be done using these individually or using in combinations. [12.]

Although each topology described in chapter 2.3 was worthwhile for this project, the linear series voltage regulator and buck converter which is interpreted in chapter 2.3.1.1 and chapter 2.3.2.2 respectively were found to be suitable. Therefore, the different circuits using these topologies were built which is presented in chapter 4.

3.3 Electronic Switch

In general, a switch is designed to interrupt the flow of the current. Switches are used in every part of the circuit which is needed on and off state. So, switches are used to control the operation of the circuit. Types of switches that are more used in electronics circuit design are electromechanical switch and electrical switch. The electromechanical switch needs to activate physically whereas the electronic switch gets activated by semiconductor action. Since the project concerned working with 5 A current, it is best to use an Electronic switch. Electronic switches are also called solid-state switches as they work in the absence of physical touch. There are different types of electronic switches of which most common are bipolar transistors, power diodes, MOSFET, IGBT, SCR, TRIAC, DIAC, etc. [13.] In this project, N-channel power MOSFET with very low R_{DS} and very high I_{AR} values should be considered.

3.4 Control

The project requirement was to engage some user-friendly push button that triggers the system to on state for a certain time and returns to off state after a certain time. With the experience of the past project, the idea was appeared of using 555 timers. 555 timers are great to use in this sort of project since these can be used to control the timing from microsecond to hours. The 555 timer works in three different modes: Astable, Bistable, and Monostable.

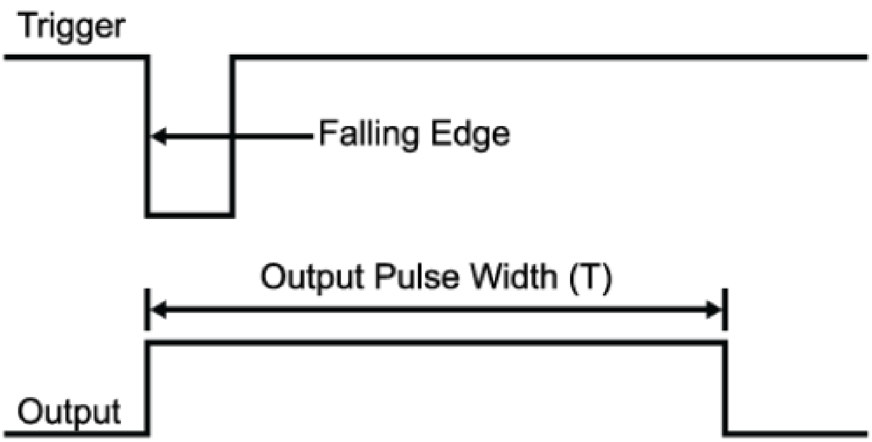


Figure 9. 555 timer in monostable mode [14].

Output Pulse Width (T) which has been shown in figure 9 can be controlled by choosing the right value of the resistor(R) and capacitor(C) as shown in the following pin configuration of the 555 timer.

The formula for the Output Pulse Width (T) is given as [14]:

$$T = 1.1 * R * C \qquad (5)$$

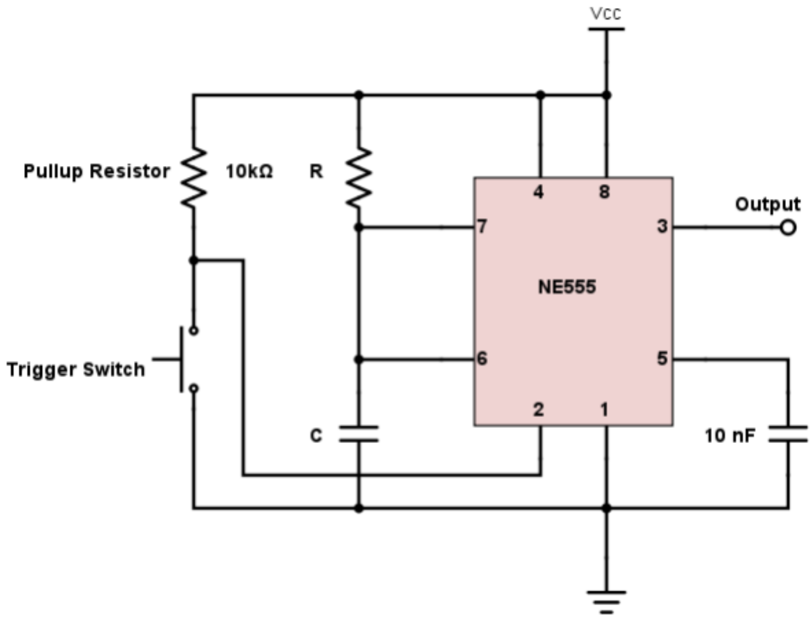


Figure 10. Monostable pin configuration of 555 timer [14].

As shown in figure 10, pin 2 acts as the trigger point. In our case, the trigger switch is the push button. So, push-button in the ON state triggers the output. This means that the output voltage becomes high as the trigger pin detects the falling edge. Then, the output time can be adjusted by the selection of the R (pin 7) and C (pin 6) according to the requirements. For the final prototype, a potentiometer of 22 kΩ and capacitor of 220 µF is used which provides about 5.4 seconds of Output Pulse Width (T). Whereas, for other prototypes, fixed values resistors and capacitors are used. It is also good to remember that the difference between the supply voltage and output voltage should be at least 1.5 V. For this project, the V_{CC} value is used 12 V which is enough to drive N - channel MOSFET fully.

4 Circuits, Measurements, and Results

The rule of the building electronics porotype is we should try different topologies so that we will be able to choose one of the structures in the final stage of the product. In this project, many different circuits were tested. In this section, circuits, measurements of those circuits, and results obtained are presented.

4.1 A Circuit Using the Linear Voltage Regulator

In the initial phase of the project, one application found in the datasheet of MC7805 was considered applicable. In the beginning, it was understood that the resistor of the tip of the wire is 1 Ω. There wasn't supposed to be a significant change in resistance of the tip from what was given at beginning of a project. It was supposed to be almost static. So, according to ohm's law 7805 linear regulator was thought to be a good choice.

4.1.1 Circuit

The circuit was built according to the datasheet. Apart from the voltage regulator the PNP transistor namely, MJ2955 was used for boosting the current as 7805 is not rated for

5 A current. In this case, MJ2955 with the TO-3 package is mounted to the heatsink to dissipate excess heat. In this way, the circuit is good to go for a constant 5 A current. The circuit as found in the datasheet [15] is shown in figure 11.

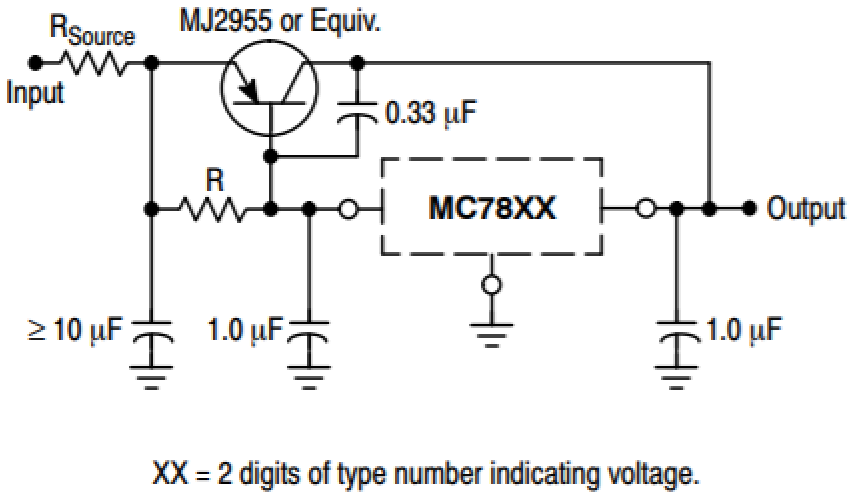


Figure 11. Current Boost Regulator [15].

The circuit was transferred then to the breadboard which is demonstrated in below figure 12.

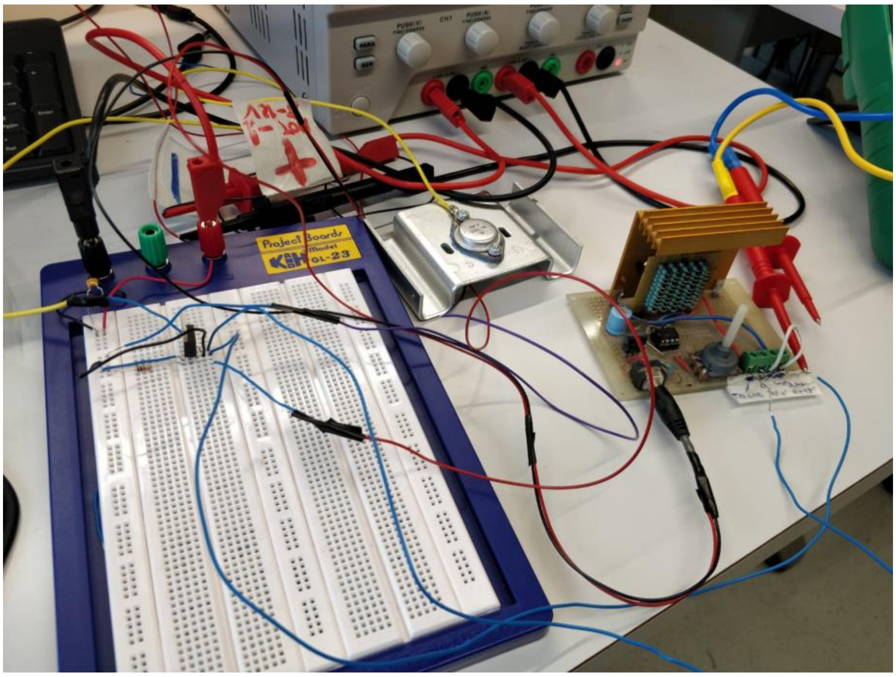


Figure 12. First prototype circuit in a breadboard.

LM7805 is used as the linear voltage regulator which provides the constant output voltage of 5 V. The MJ2955 transistor is mounted in the heatsink. Also, in front of the transistor, there is a bulk resistor that works like a potentiometer.

4.1.2 Measurements

Firstly, the circuit was simulated in multisim to analyse component's behaviour and their functions. In multisim, the circuit was functioning properly and efficiently. In figure 13, the snippet of the design while simulating is shown. In the following condition, the circuit was providing about 5 A current to 1 Ω load. This simulation was helpful to build the prototype.

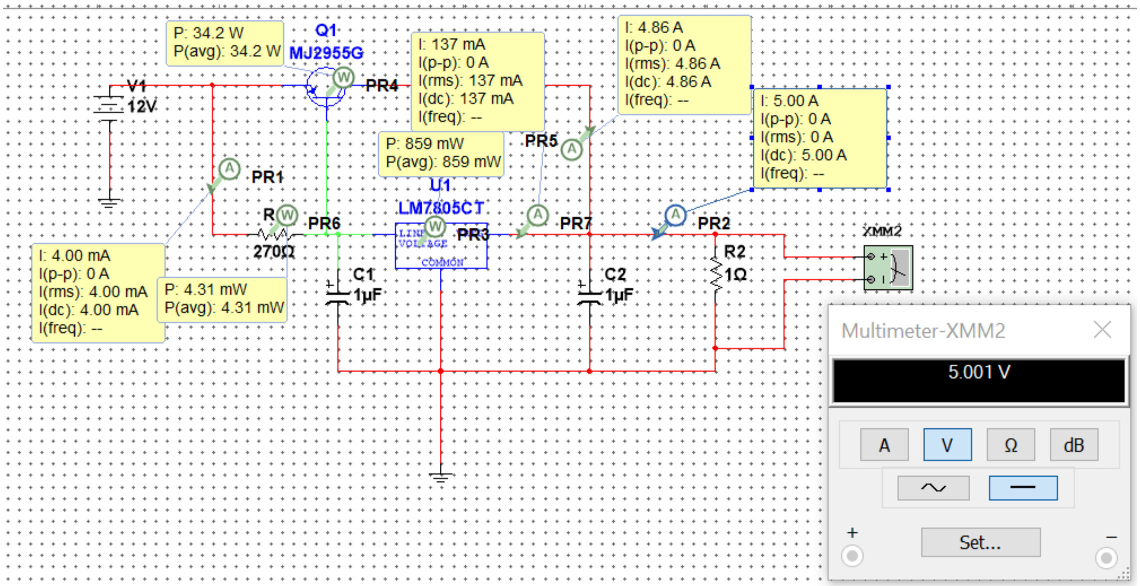


Figure 13. Simulation of a circuit in Multisim.

Most of the power was dissipated in a transistor. From this simulation, it was clear that this circuit can be performed only by using a good heat sink for the transistor and if the low load has been used it should be with a high power-rating resistor. After performing on the breadboard, the following results were obtained.

Table 1. Testing circuit with different load.

Input Voltage	Load Current	Load Resistance	Output Voltage
12	0.001	5040	5.04
12	0.08	62.875	5.03
12	0.161	31.24223602	5.03
12	0.241	20.87136929	5.03
12	0.319	15.73667712	5.02
12	0.399	12.55639098	5.01
12	0.48	10.4375	5.01
12	0.558	8.960573477	5
12	0.64	7.8125	5
12	0.72	6.930555556	4.99
12	0.8	6.2375	4.99
12	0.88	5.659090909	4.98
12	0.96	5.1875	4.98
12	1.02	4.87254902	4.97
12	1.549	3.189154293	4.94

As shown in table 1, there has been steady output voltage across the load regardless of the load resistance. Similarly, there has been a steady increment of current in output as the resistance decreases in the load. Initially, the test was not performed with 1 Ω resistance as there was no resistor available with a good power rating at that time. But, following Ohm's law the current through the load should be around 4.9 A at a load resistance of 1 Ω which is also explained in the following figure 14.

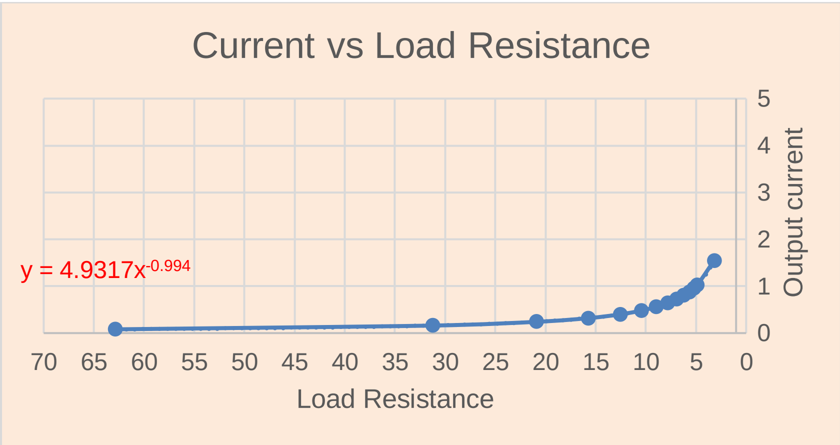


Figure 14. The behaviour of the output current as load resistance changes.

In the given chart, the y- axis represents the current and the x-axis represents the load resistance.

4.1.3 Results

The formula shown in figure 14 illustrates that the output current will be in the range of 5 A when a 1 Ω load is applied. So, the circuit was tested by connecting a 1 Ω resistance tip (initial load) at output resulting in a 4.89 A output current which was supposed to be enough to cut the suture. The tip was red hot and the power supply was switched off manually since the timer was not included in the very first prototype.

4.2 Circuit after Adding 555 Timer and Switch

Switch and the timer were added to the previous circuit, then circuit was printed using a milling machine, and components got soldered according to the schematic. In this chapter, circuit board design, multisim simulation, measurements, and results are explained.

4.2.1 Circuit Board Design

The circuit was performing as expected on a breadboard. Therefore, it was transferred to the schematic in PCB Logic. The schematic is represented in Appendix 1.

The Schematic in Appendix 1 represents two different parts of the circuit with two different sources. The circuit on the left is a timer circuit that controls the circuit on the right. The circuit on the right provides 5 A current for 1 Ω load. Each circuit has a 12 V source with a separate ground. The timer circuit has an LED that is turned on when the output is high and turned off when output is low. As it is described in chapter 3.4, the timer controls with the value of Resistor and Capacitor. In this case, the value of the capacitor is 470 µF and the value of the resistor is 10 kΩ. Therefore, the output time width(T) is 5.17 s. Meaning that it provides about 10.5 V output voltage for 5.17 s and then comes down to 0 V.

U2 in Appendix 1 represents the N-Channel MOSFET which works as a switch. To saturate MOSFET fully, it is connected before the voltage regulator. J3-1 is the emitter of the transistor MJ2955, J3-2 is the collector and J3-3 is the base of the transistor.

This was transferred to the PADS layout where the design of the circuit to make it print-ready was done. The layout design is shown in Appendix 2.

Appendix 2 shows the top view of the design layout. Components are mounted on the top side of the board and traces are made on the bottom side of the board. Traces on the left side seems to be bigger, the reason behind that is high current flows through those big bulky traces. Later, the circuit board was printed, and components were soldered as shown in figure 15.

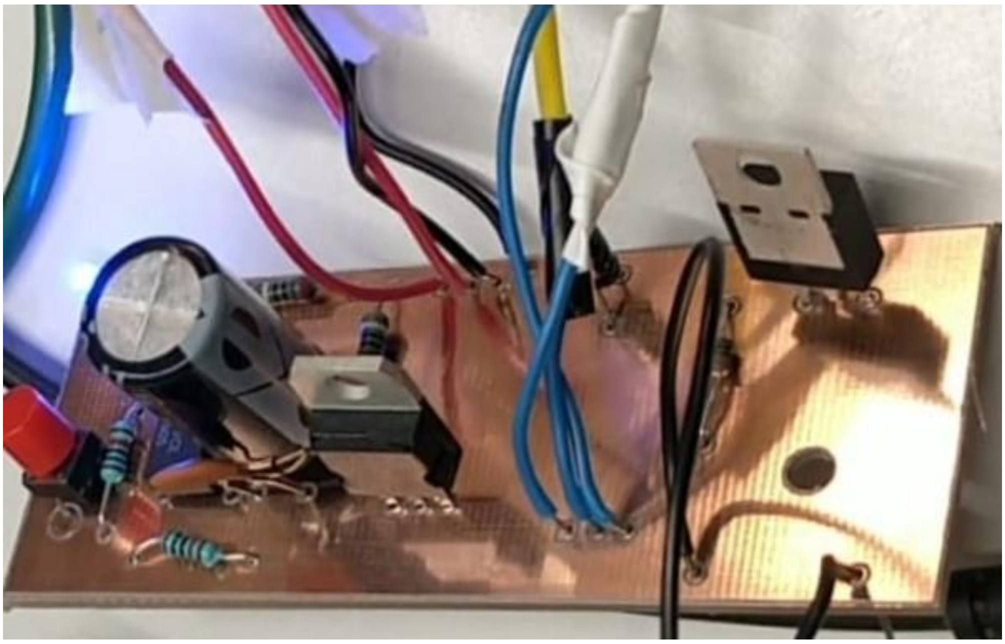


Figure 15. The working circuit on the copper board.

4.2.2 Multisim Simulation

Simulation of the circuit was done after adding the timer circuit. But simulation led to some error. It can also be seen from figure 16, the simulation is not performing well, and it is short-circuited. Since voltage for MOSFET's gate and source cannot be provided from the same source, two different sources are in use. And, the circuit is supposed to have isolated grounds. However, multisim ties both grounds together which leads to a short – circuit. Therefore, the circuit could not simulate in multisim.

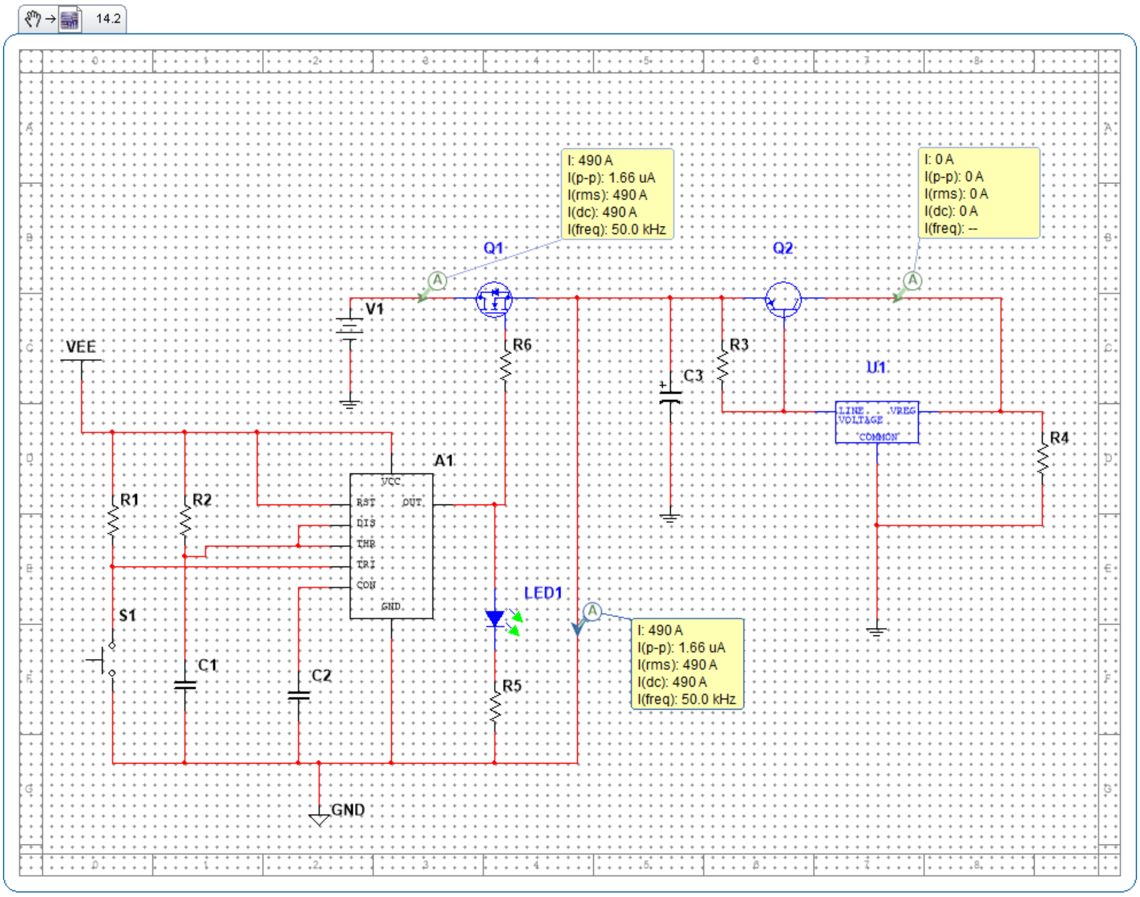


Figure 16. Circuit simulation after adding timer circuit.

4.2.3 Measurements and Results

Following, the circuit was tested using a lab power supply with a rating of 0-30 voltage and 5.1 A current rating. The output current through the load when the ammeter was in series appeared to be 4.808 A. This time, load resistance was 1 Ω with a 20-watt power rating. Later, the circuit was tested with the tip which was completely different from what was given at the beginning of the project. Assuming, tip resistance is still 1 Ω, the voltage in the power supply dropped as the voltage cap was present. Surprisingly, it was discovered that the output current was higher than 5.1 A. As electronics lab power supplies were able to supply only 5.1 A maximum current, the circuit tested in a power lab where power supply with a current rating of 10 A was available. While testing, in about 3 seconds, the wire burnt which meant the current through the load was more than 5 A.

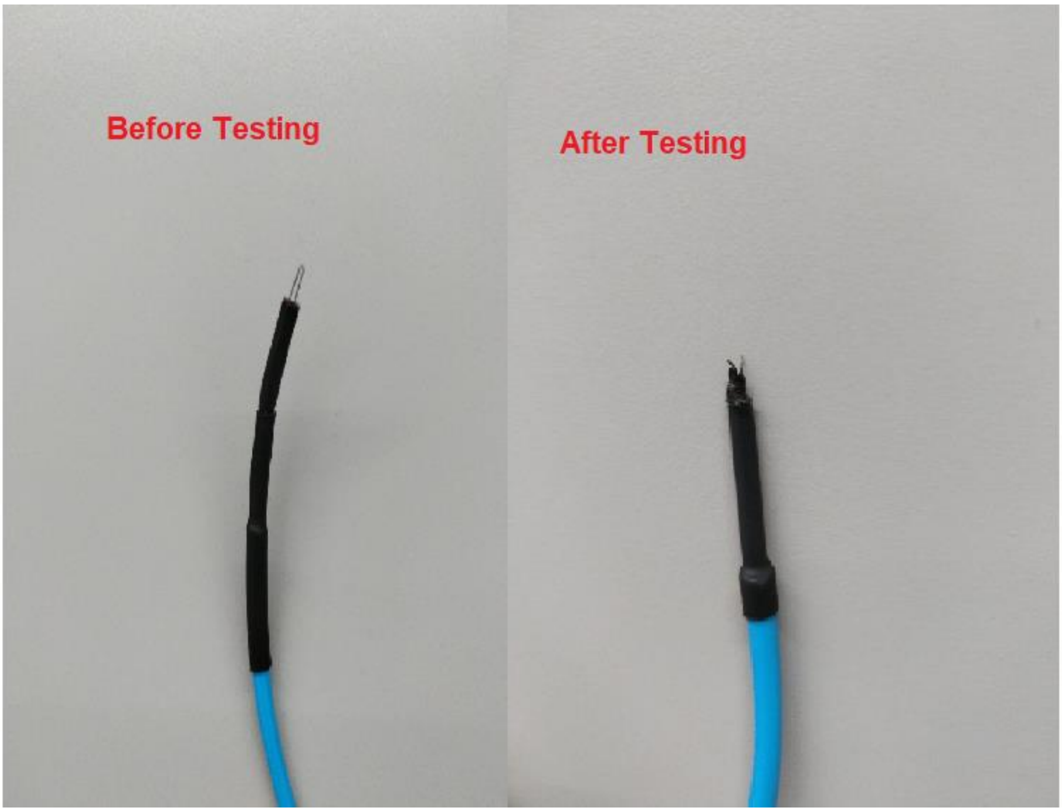


Figure 17. Burnt wire after testing with 10A rated power supply.

As the voltage across the load was regulated to 5 V, it was crucial to measure the resistance of the wire to know tip resistance. By using the 4-wire resistance measurement method, the resistance of the wire was found to be 0.670 Ω with dc connector and 0.459 Ω without dc connector. This means the current was firing more than 5 A and the load (tip) could not handle thermal power generated by the current. The solution to this problem would be adding a suitable high-power rating resistor in series to load. But it was not the best solution. Therefore, more research was needed to design a new circuit.

4.3 A Circuit Using an Adjustable Voltage Regulator

After doing some research, it was found that LM317, 3- terminal adjustable regulator which already comes with several applications appeared to be suitable. This regulator always keeps a reference voltage of 1.25 V of a voltage difference between output and the adjustable pin. The PNP transistor (MJ2955) would be the best to use with this regulator to get the required result as the transistor comes with a good current gain. A similar technique was used in the datasheet of LM317 [16], but a different approach would also be better. An article explanation [17] was found to be suitable for our project. The approach in the article was drawn in figure 18.

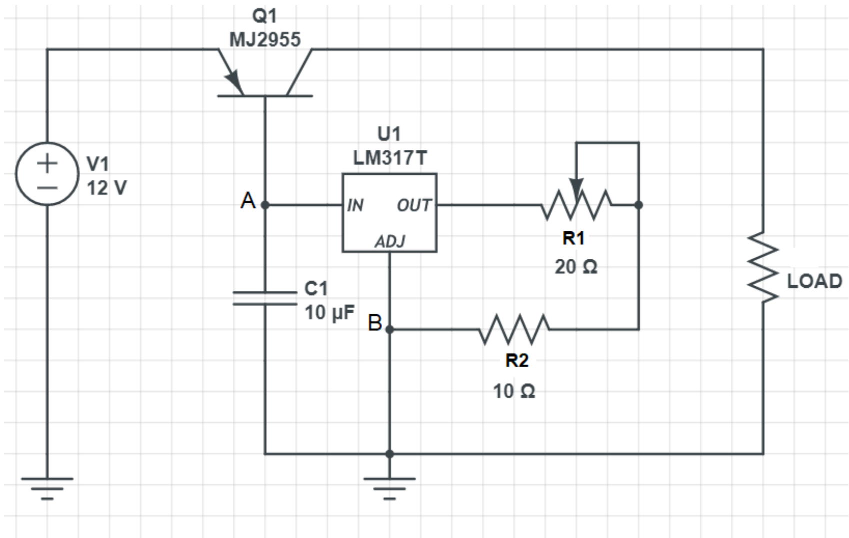


Figure 18. High power constant current source circuit [17].

The working principle of the circuit is mainly based upon amplifying the transistor's base current using a transistor with high current gain. In figure 18, the current in point A (base current) and current in point B (regulator output current) are the same as the LM317T is a linear regulator. The idea was to use LM317T as a current source and to amplify the current using a PNP transistor. The power rating of the resistor (R2) should be considered as variable current passes through it while tuning the circuit. The output current at point B can be calculated using equation 6.

$$Regulator\ output\ current\ = \frac{V_{ref}(1.25)}{(R1\ +\ R2)} \qquad (6)$$

Furthermore, constant output current (as required) can be adjusted using a potentiometer (R1).

4.3.1 Circuit Board Design

The circuit was first tested on the breadboard and printed on the copper circuit board. The whole circuit schematic is presented in appendix 3. The layout of the circuit is given in appendix 4.

Design demonstrated in appendix 3 displays that the N-channel MOSFET switch is connected in series with load in high side configuration. The circuit gets on when the Output Pulse of the 555 timer is high which provides about 10.5 V and 5 A current flows through MOSFET to the load. Since MOSFET with very low R_{DS} was chosen, there was not any concern with MOSFET getting heated.

After the circuit was milled using the milling machine, all components were soldered to the copper board. In addition, two holes were drilled in PCB to attach the circuit to the prototype box using screws. Finally, the whole circuit was attached to the PCB enclosure which is shown below in Figure 19.

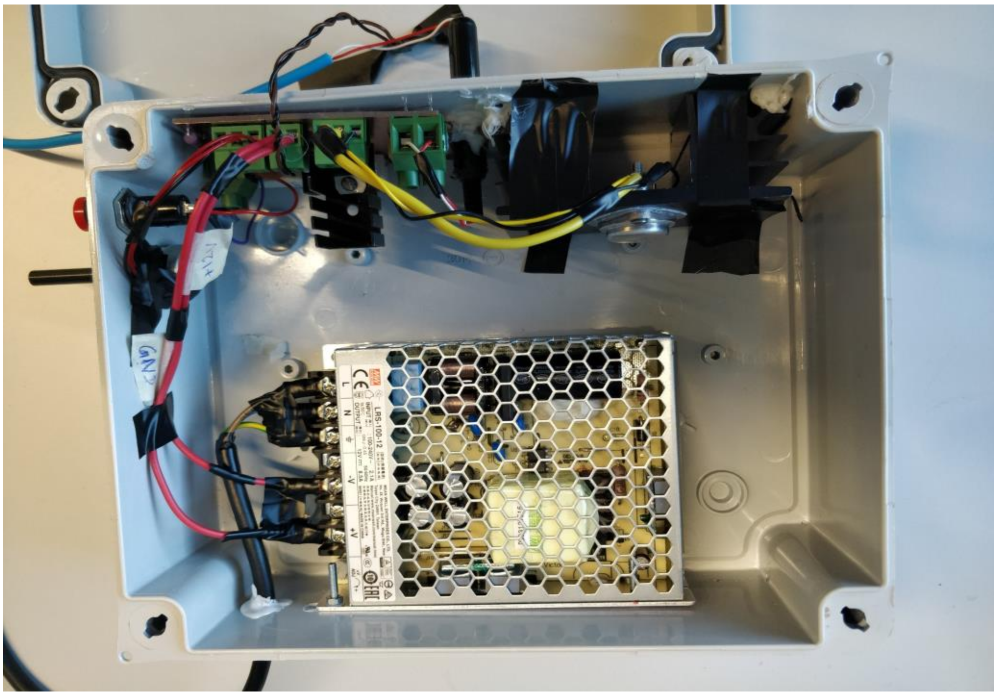


Figure 19. The working circuit is connected to the power supply inside the PCB enclosure.

As shown in figure 19, a 100 W single-output power supply is used which is also a good quality power supply. It comes with very good features such as short circuit/ over voltage/overload protection. Additionally, it is also in compliance with IEC/EN 60335- 1(PD3) and IEC/EN 61558-1,2-16 for household appliances. [18.] For that reason, it has added an extra layer of safety to the device. Since AC voltage is in used, the circuit is isolated from the power supply taping the wire as well as exposed metal parts. Wires connected to the power supply are very well crimped and tied together so that even one wire loses its place there will not be any danger to be connected to the AC line. Concerning safety, DC parts and AC parts should have some distance which is well maintained inside the enclosure.   In addition, the push-button, potentiometer, and output DC connectors are well isolated from the AC line. There is a huge heatsink mounted on the MJ2955 transistor, which is supposed to be enough for the operating circuit.

The heatsink suitability was chosen using the following calculation.

In this case,

I_{IN} = 5 A, V_{IN} = 12 V,

Input power = 60 W

Then,

I_{OUT} = 5 A, R_{LOAD} = 0.670 Ω,

Output power = 16.75 W

Therefore,

Power dissipated by MJ2955 = 60 W - 16.75 W = 43.25 W.

$R_{\theta jc}$ = 1.52 °C/W [15]

The heatsink thermal resistance maximum value (θca) = 2.53 K/W [19].

The thermal resistance of the selected heatsink was 2.3K/W which was found to be sufficient for this project. In addition, the circuit was on for only 5 seconds so cooling requirements were easier to meet.

4.3.2 Multisim Simulation

Figure 20 represents the multisim simulation of the circuit explained in chapter 4.3.1. 555 timer in the left works as a controller circuit which job is to turn on and offload the circuit.

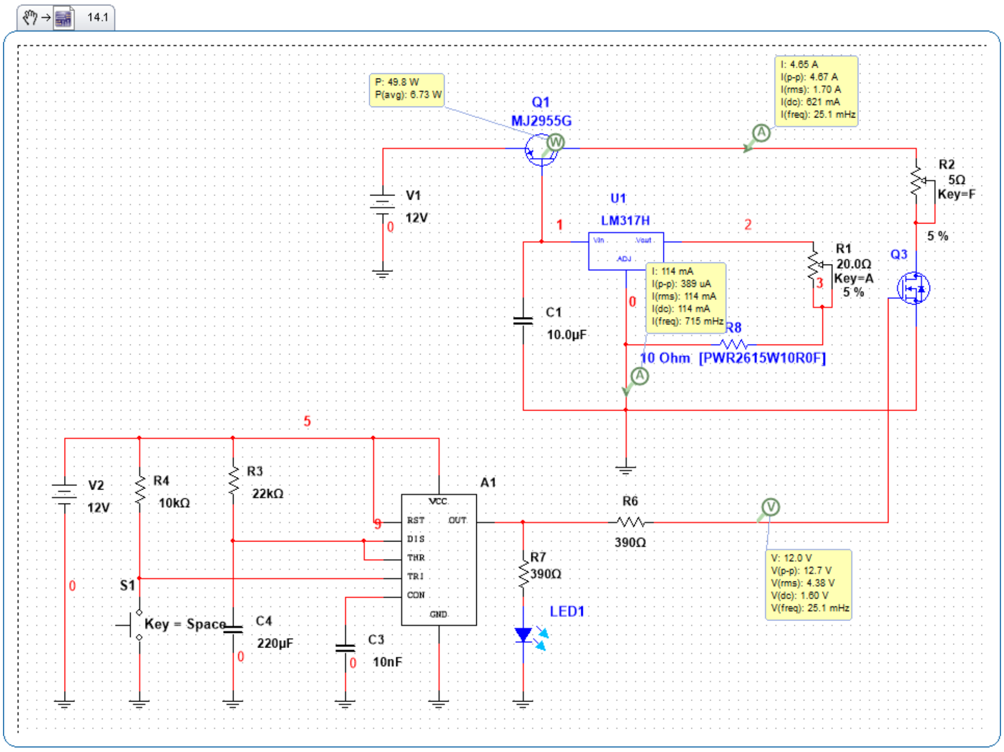


Figure 20. Multisim simulation of a circuit using an adjustable voltage regulator.

The circuit was analysed in multisim by adjusting the load from 0.25 Ω to 5 Ω. A current probe was connected in series to load. The readings are shown in table 2.

Table 2. Circuit simulation in Multisim with the different load resistance.

Resistance(Ω)	Current(A)	Voltage(V)
0.25	4.65	1.1625
0.5	4.56	2.28
1	4.39	4.39
1.5	4.23	6.345
2	4.08	8.16
2.5	3.94	9.85
3	3.78	11.34
3.5	3.27	11.445
4	2.88	11.52
4.5	2.57	11.565
5	2.32	11.6

As interpreted in table 2, the current remains near to constant under 1.5 Ω. The current through load starts to drop significantly as load resistance increases. This is due to the high collector-emitter saturation voltage of the transistor [20]. It concludes that the transistor (MJ2955) should operate inactive region to use as a current amplifier.

4.3.3 Measurements and Results

The measurement setup was done by adding an ammeter in series to the load. The circuit was tested several times before it was sent to Paris for testing with their application. The output current through the load was constant at 5.04 A. There was a quiescent current of 58 mA present in the circuit. This current is mainly consumed by the LM317T when the load side was off. And, this current was not present on the load side of the circuit therefore there should not be an issue. 5.04 A current had provided enough thermal power to cut surgical sutures of lifeline thread. In figure 21, the red-hot tip of the wire can be seen. Besides, the test was done by keeping surgical sutures and the tip of the wire in a bowl filled with cow's blood. Although the blood was quite cold, the surgical suture was cut up just in 4 seconds.

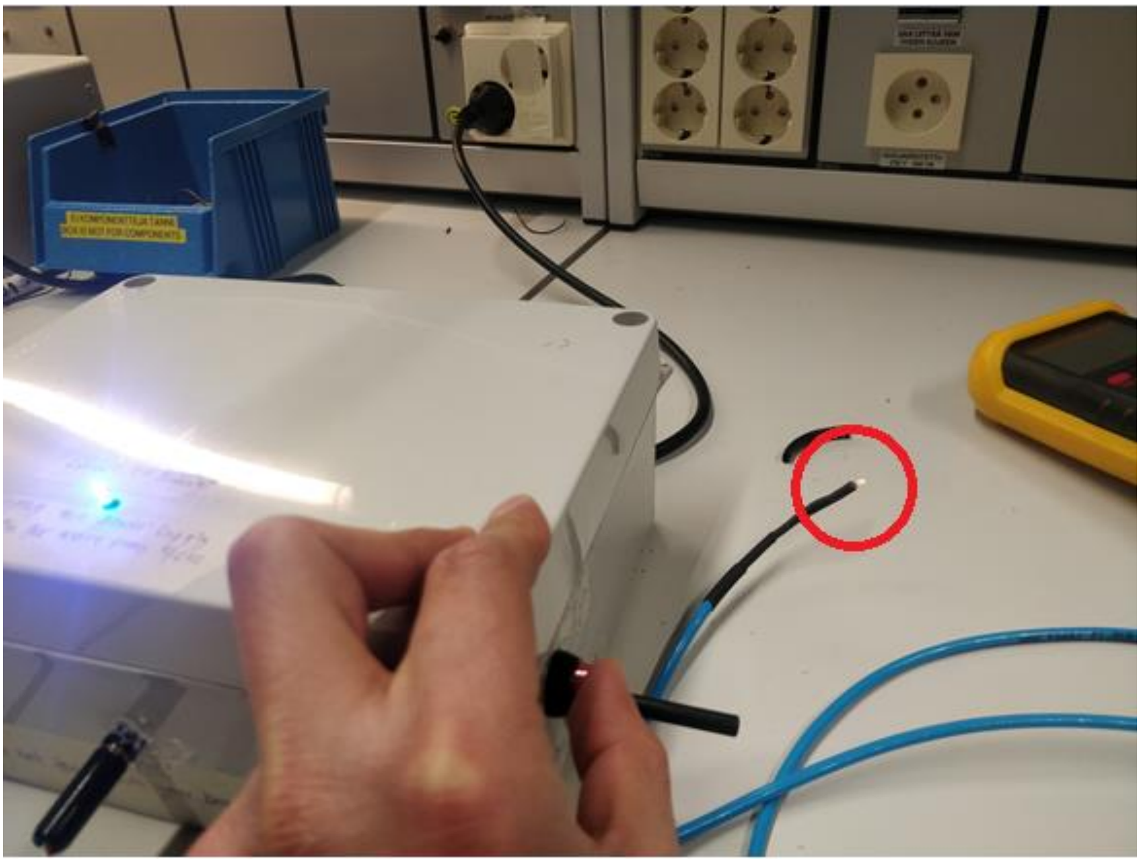


Figure 21. Wire's tip getting red hot when 5 A current passing through it.

The real-time output current was higher than the output current from the simulation. The factor was due to the gain of the transistor. When transistors are tested in the circuit, the gain of the transistors is not exactly in the same range as explains in the datasheet. In this case, the gain was higher than provided by the datasheet. Also, this constant current circuit is meant for only a small load which is less than 2 Ω because of high collector-emitter saturation voltage (2V-3 V when current is more than 5 A [20]).

4.4 Circuit Design Using Switching Regulator

In chapter 2.3.2, there was a short explanation about switching regulators. As this project's main goal was to make different prototypes using different topologies, a switching regulator was also chosen for prototyping. Since the application needed 5 A, 10 A switching regulator would be enough for this project. L4970A stepdown power switching regulator delivering 10 A current with voltage varying from 5.1 V to 40V would be a good choice. Datasheet of the L4970A [21] includes the following circuit as shown in figure 22 which resembles our requirement.

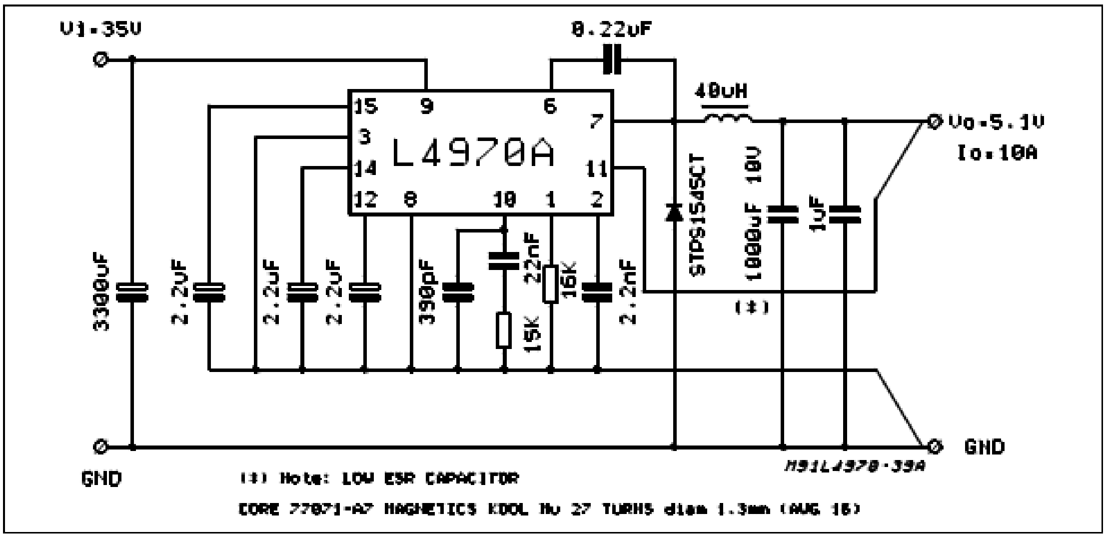


Figure 22. 5.1V /10A Low-cost application [21].

It was believed that after tweaking the circuit a little bit, it would be possible to build a constant current source. Circuit design, measurement, and the results are explained here.

4.4.1 Circuit Design

As it can be seen from figure 23, the pin 11 function is providing feedback to the system taken from output. Pin 11 is connected to the comparator's inverting side which has 5.1V in its non-inverting side. By this, it maintains the constant output current. The idea behind making the above circuit work as the current source was by adjusting the circuit. This can be done by using differential amplifier topology to sense voltage drop directly across the sense resistor and with certain gain amplifying the sense voltage such that the output of the amplifier becomes 5.1 V. [22.] By this, there will not be any change in the voltage feedback pin (pin 11). As a result, a very reliable constant current source can be made. The complete schematic of the circuit which works as a constant current source is presented below in figure 23.

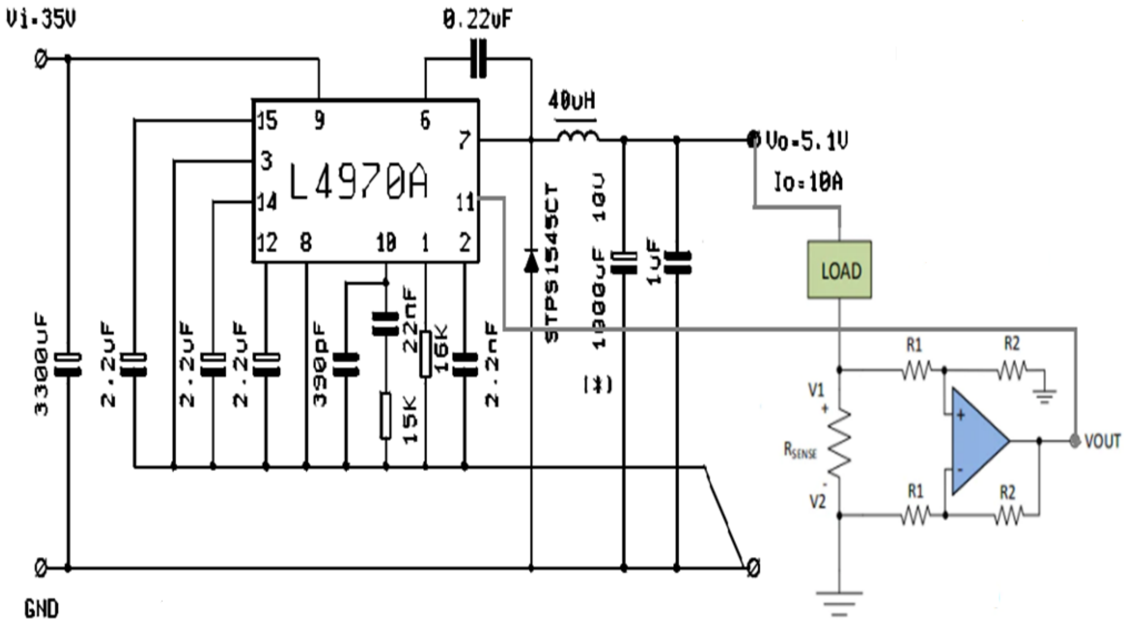


Figure 23. Constant current source using switching regulator.

The circuit was built on the breadboard on this basis which is shown in Appendix 5. In Appendix 5, OPA340 rail-to-rail operational amplifier from Texas Instruments, 0.01 Ω sense resistor are in used. By using the following equation [22] value of resistors (R1 and R2) were determined.

$$Output\ voltage\ of\ Op - Amp\ (VOUT) = \frac{R2}{R1} \times (V1 - V2), V1 - V2 = V_{RSENSE} \quad (5)$$

Current required across load = 5 A

Sense voltage (V_{RSENSE}) = 0.01 Ω × 5.1 A = 0.051 V.

This voltage needs to amplify with a certain gain to get 5.1 V in output.

R2 / R1 = Gain = 100. (From equation (5))

Therefore, the combination of R2 and R1 which gives a gain of 102 will work. In this case, R2 is 333 kΩ and R1 is 3.3 kΩ which provides a constant 5.1 A current.

4.4.2 Measurements and Results

The measurement setup is presented in appendix 6. Ammeter is connected in series to load. The input voltage for the regulator was set to 24 V and 5.80 V for the op-amp.

The green object in appendix 6 represents the load with the resistance of 2.2 Ω and power rating of 1000 W. While analysing the circuit and taking the measurement, the load resistance was changed but the power rating of the load was taken into concern. The measurement readings are shown in table 3. Here, UUT represents the wire (tip) with a resistance of 0.460 Ω.

Table 3. Current through the load for different load resistance.

Load	Current(Load)
1Ω	5.01A
2.2 Ω	5.06A
1Ω + UUT	5.06A
2 Ω + UUT	5.06A
UUT	5.07A
SHORT CIRCUIT	5.08A

With these results, it is verified that circuit works without any problem and it provides constant output current regardless of the load resistance. As this is the switching regulator, the circuit is more efficient and there was not any worry of heat dissipation.

As the circuit was running for a long time there was a decrease in output current by 200 mA. This was due to R_{SENSE} resistor heating up as the circuit was for a longer time resulting slight decrease in output current. But this problem can be solved by using high power rated R_{SENSE} resistor.

5 Conclusion

The goal of the project was to develop different prototypes that can provide a constant 5 A current source for a certain time (in our case up to 5 seconds). Three prototypes were developed of those two are working very well. The circuit built using an adjustable regulator (LM317T) was able to provide 5 A constant current for the time determined by the 555 timer. Although the circuit built with a switching regulator (L4970A) was also providing 5 A constant current, a timer for providing current to load for designated time needs to be developed. Undoubtedly, it can be done by using a 555 timer (similar approach as in LM317T). Since L4970A is already an advanced regulator, output current timing is possible to engineer using its pins. While comparing linear regulators and switching regulators, switching regulators are far better. Hence, building a full version using the switching regulator is important in future development. Since these are only the prototypes, vigorous testing and further development are still needed before it goes to market.

On the possible future work, some matters need to be addressed. Since switching regulators are the safest and efficient, the final product would be better by using a switching regulator. Furthermore, adding short circuit protection, high current for a shorter period to avoid a blood clot, and adding an emergency button to shut down the whole circuit if something goes wrong while operating should be taken into consideration. Additional features such as developing cable and tip, long connector cable on the output in the output so that it will be easier to work with while performing surgery can be developed.

6 References

[1] Wikepedia, “Circuit design,” 2020. [Online]. Available: https://en.wikipedia.org/wiki/Circuit_design. [Accessed 19 December 2020].

[2] W. H. Paul Horowiz, in *The Art of Electronics* , CAMBRIDGE UNIVERSITY PRESS, 1989, p. 9.

[3] Electronics Tutorials , “Voltage sources,” 2020. [Online]. Available: https://www.electronics-tutorials.ws/dccircuits/voltage-source.html. [Accessed 29 December 2020].

[4] Electronics Tutorials, “Current Sources,” 2020. [Online]. Available: https://www.electronics-tutorials.ws/dccircuits/current-source.html. [Accessed 29 12 2020].

[5] P. Khatri, “Electronic Circuits,” CircuitDigest, 27 July 2018. [Online]. Available: https://circuitdigest.com/electronic-circuits/voltage-regulators. [Accessed 29 December 2020].

[6] Wikepedia, “Linear regulator,” 2020. [Online]. Available: https://en.wikipedia.org/wiki/Linear_regulator. [Accessed 18 December 2020].

[7] Y. M. Lai, “Power Supplies,” in *Power Electronics Handbook*, M. H. Rashid, Ed., 2017, pp. 659-84.

[8] ROHM Semiconductor, “ROHM,” [Online]. Available: https://www.rohm.com/electronics-basics/dc-dc-converters/linear-vs-switching-regulators. [Accessed 26 02 2021].

[9] E. Coates, “Learnabout electronics,” 29 December 2020. [Online]. Available: https://learnabout-electronics.org/PSU/psu32.php. [Accessed 5 January 2021].

[10] E. Coates, “Learnabout electronics,” 2020. [Online]. Available: https://learnabout-electronics.org/PSU/psu31.php. [Accessed 5 January 2021].

[11] Electrical4U, “Electrical4U,” 2021. [Online]. Available: https://www.electrical4u.com/buck-boost-converter/. [Accessed 2 January 2021].

[12] E. Odunlade, “Current Regulators: Construction, Working and Design Types,” CircuitDigest, 05 June 2019. [Online]. Available:

https://circuitdigest.com/tutorial/current-regulators-construction-working-and-design-types#:~:text=Current%20Regulator%20Working&text=When%20the%20circuit%20is%20powered,load%20resistance%20or%20voltage%20supply.&text=These%20variations%20also%20affect%2. [Accessed 29 December 2020].

[13] ELECTRONICS HUB, “switches,” ELECTRONICS HUB, 16 September 2015. [Online]. Available: https://www.electronicshub.org/switches/. [Accessed 1 January 2021].

[14] ALL ABOUT CIRCUITS, “555-timer-monostable-circuit,” 2017. [Online]. Available: https://www.allaboutcircuits.com/tools/555-timer-monostable-circuit/. [Accessed 1 January 2021].

[15] On Semiconductor, “onsemi,” 2019. [Online]. Available: https://www.onsemi.com/pub/Collateral/MC7800-D.PDF. [Accessed 6 August 2020].

[16] Texas Instruments, “LM317 3-Terminal Adjustable Regulator,” Texas Instruments, September 1997. [Online]. Available: https://www.ti.com/lit/ds/symlink/lm317.pdf?ts=1620019123007&ref_url=https%253A%252F%252Fwww.google.com%252F. [Accessed April 2021].

[17] L. Loflin, “bristolwatch,” 2018. [Online]. Available: https://www.bristolwatch.com/ccs/LM317b.htm. [Accessed 14 January 2021].

[18] Yleiselektroniikka Oyj, “yeint,” 3 OCTOBER 2017. [Online]. Available: https://www.yeint.fi/amfile/file/download/file_id/227433/product_id/223008/. [Accessed 25 JAN 2021].

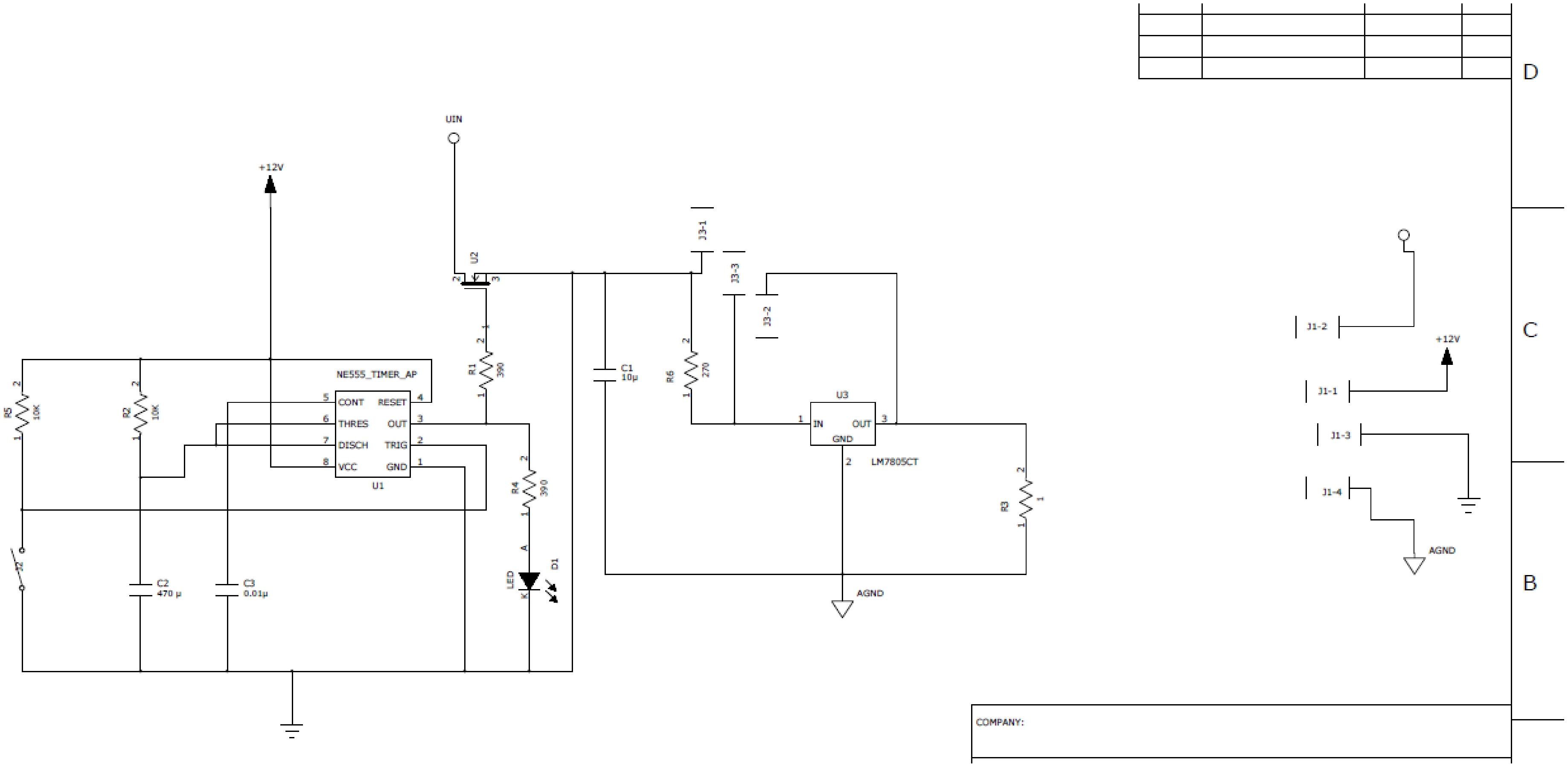
[19] H. W. Silver, “Choosing a Heatsink,” *NUTS & VOLTS,* pp. 67-69, July 2006.

[20] ON Semiconductor, “Onsemi,” January 2000. [Online]. Available: https://www.onsemi.com/pdf/datasheet/2n3055-d.pdf. [Accessed 11 October 2020].

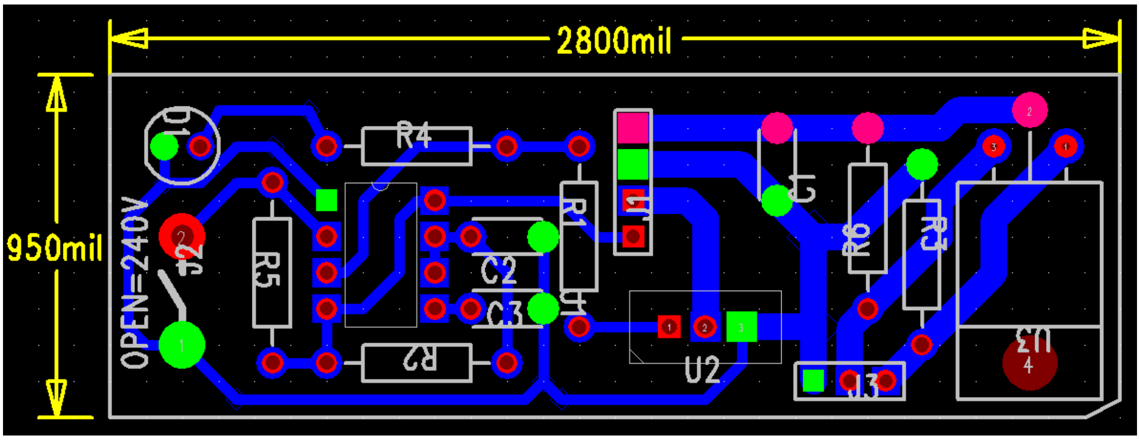
[21] STMicroelectronics, “st,” 2000. [Online]. Available: https://www.st.com/resource/en/datasheet/l4970a.pdf. [Accessed 20 August 2020].

[22] D. Hudgins, “Texas Instruments,” March 2017. [Online]. Available: https://www.ti.com/lit/an/sboa190/sboa190.pdf?ts=1615559085418&ref_url=https%253A%252F%252Fwww.google.com%252F. [Accessed 29 August 2020].

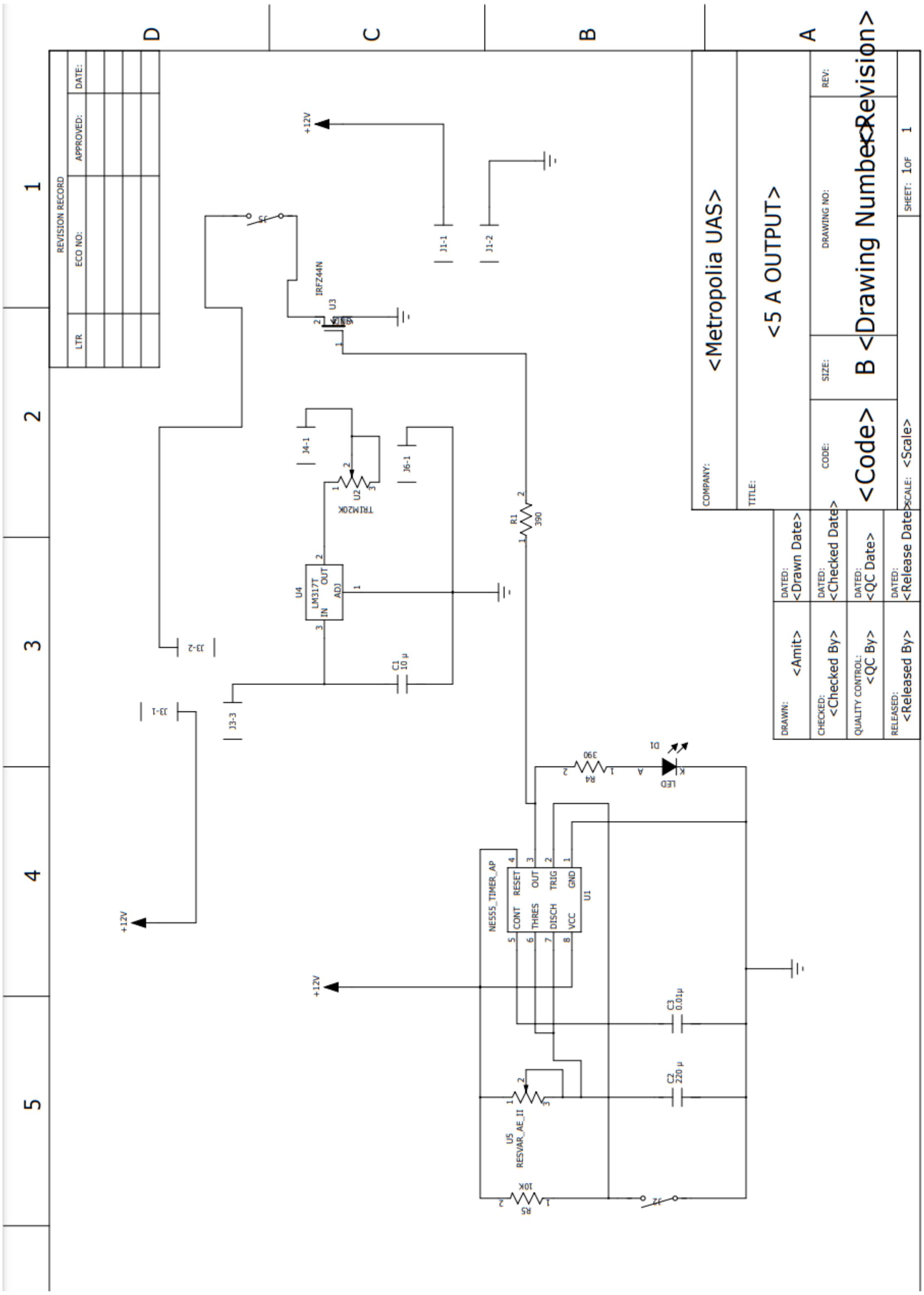
Appendix 1. Schematic of a circuit with linear regulator (LM7805), 555 timer, and switch.



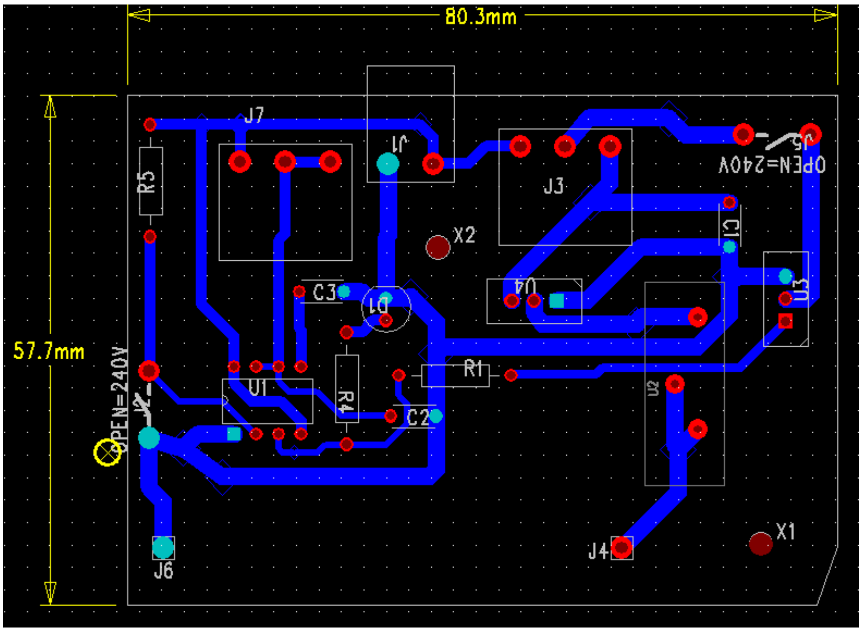
Appendix 2. Circuit's layout with linear regulator (LM7805), 555 timer, and switch.



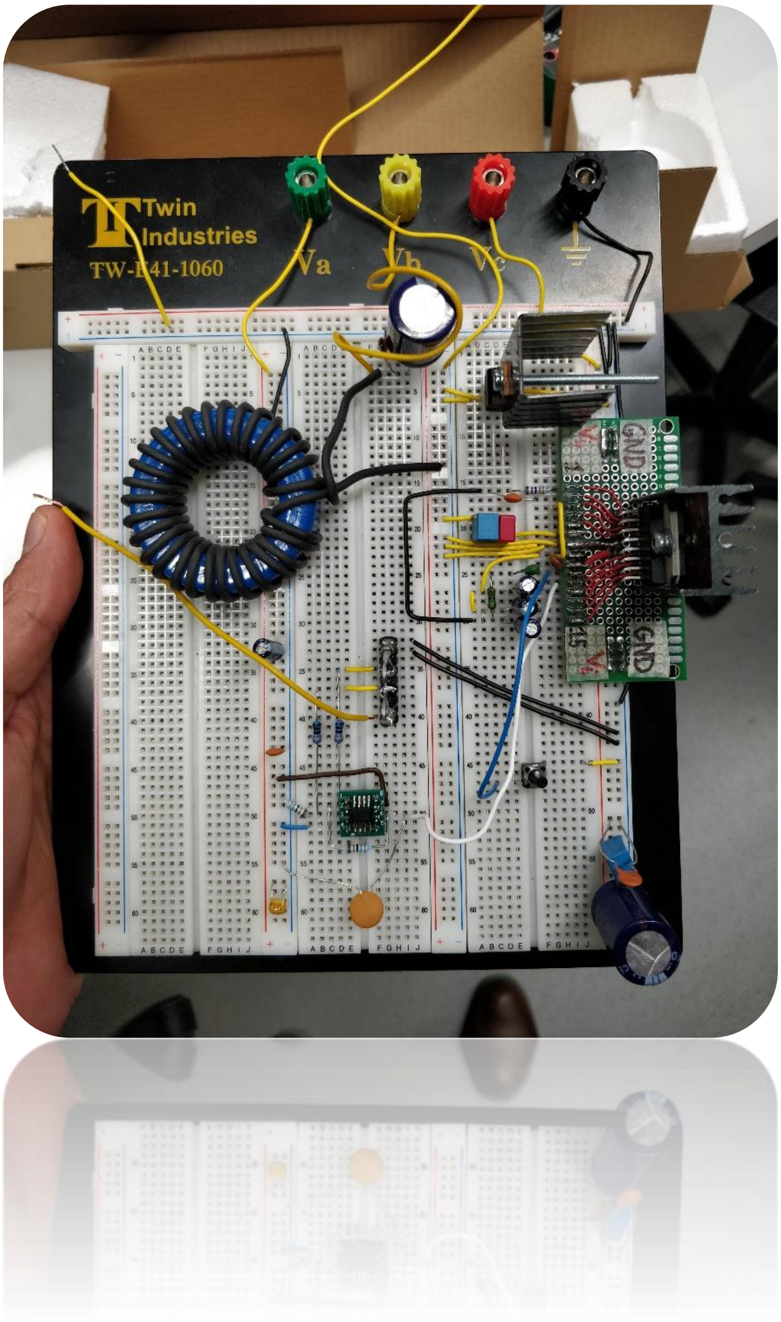
Appendix 3. Schematic design of a final prototype using regulator LM317T.



Appendix 4. Designing of a circuit using adjustable voltage regulator in PADS Layout.



Appendix 5. A breadboard is used to make a constant 5.1 A current source circuit using switching regulator L4970A.



Appendix 6. Measurement setup to analyse circuit while varying load resistance.