Group No. 18

Department of Electronic and Telecommunication Engineering
University of Moratuwa
EN2090 – Laboratory Practice 2

LAB Project – Semester 3

Name	*Index No.*	*Contribution*
Priyashan B.W.S.	190476V	Enclosure Design, Printing the Enclosure, Soldering Components
Punsara K.K.S.	190481G	Designing the Circuit, Choose the electronic components, Troubleshoot the dot board implementation
Ranasinghe K.K.H.	190494A	Testing the circuit, Creating the data sheet of the product, designing the circuit
Rajapaksha R.M.P.A.P.	190484T	PCB designing, Printing the PCB, Managing the Project Report

This is submitted as a partial fulfillment for the module
EN2090: Laboratory Practice 2
Department of Electronic and Telecommunication
University of Moratuwa

May 7, 2022

Contents

Abstract

We were tasked to design a **$10V$ linear power supply with a maximum current rating of $10A$**. A linear power supply is used to drive a load under constant voltage conditions. A step-down transformer was provided to reduce the $230V$ input line voltage to a $15V$ (rms) AC voltage. After the transformer output, our design includes a rectification stage implemented using a *KBCP3510* bridge rectifier and a smoothing stage consists of three $12mF$ electrolytic capacitors. Then, the voltage regulation and current limiting for $10A$ maximum current is achieved by a *MJ11016G* Darlington transistor, $6.3V$ Zener diode, *2N5551* transistors and high-power resistors. A potentiometer was added to fine-tune the output voltage to the required $10V$. Our design also includes short circuit protection. The prototype design showed great results with the required voltage regulation and current limiting when used with low resistive loads. Heat sinks were used with the power transistor and the bridge rectifier to allow proper heat dissipation. Our project was completed with a single layer PCB with high power paths and a 3D printed enclosure to house the PCB. Furthermore, a $12V$ DC fan was integrated to the enclosure with adequate vents to cool down the heating components of the circuit.

Section 1 – Introduction – Our Design

The main goal of this project was to design and implement a $10V$ linear power supply with a maximum current rating of $10A$ under constant voltage conditions. We were expected to design a voltage regulator from scratch that can drive a high-power load ($100W$) from $230V$ input voltage. The final design of the linear power supply had to be designed only using Analog electronics components.

The design of a linear power supply includes stepping down an input AC line voltage using a transformer and a series of rectifier circuitry and a filtering process to produce a clean regulated DC voltage.

Several factors such as power efficiency, load and line regulation, circuit protection mechanisms have to be taken into consideration when designing a linear power supply.

Linear power supplies are used in applications that require excellent regulation, low ripple, low electromagnetic emissions and excellent transient response. Examples of applications that may require a linear power supply are communication equipment, medical equipment, low noise amplifiers, laboratory test equipment, etc.

A complete schematic of our design is shown in the following diagram.

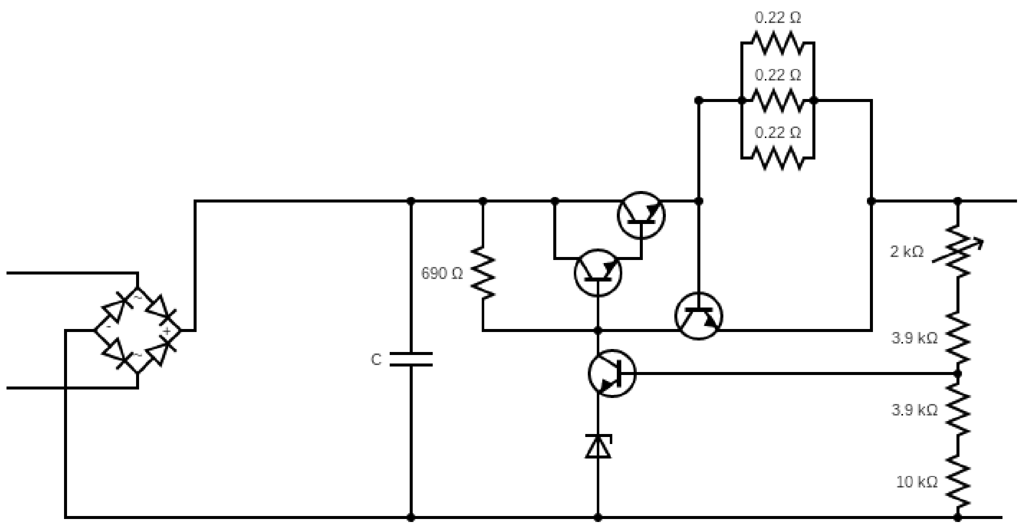


Figure 1 - Complete Circuit

Section 2 – Methodology

Section 2.1 – Rectification

The first stage of a DC voltage power supply is Rectification, which is responsible for converting the input sinusoidal AC voltage to an output voltage with a single polarity. There are many different techniques to implement the rectification stage in a circuit, however, the most common and simplest way is using a Wheatstone bridge rectifier.

A Wheatstone bridge is a collection of four diodes configured in a way that the output current will always flow in a single direction irrespective of the polarity of the input voltage applied. The below Figure 2 shows a diagram of a bridge rectifier. By analyzing the circuit, we can obtain few equations for the rectifier regarding its output voltage (V_{out}) and the voltage across a reversed-biased diode (V_{rev}), in terms of the input voltage (V_{in}) and the nominal voltage drop across a forward-biased diode (V_d). Notice that the following equations only hold true when there is an output forward current, thus when $|V_{in}| \geq 2V_d$.

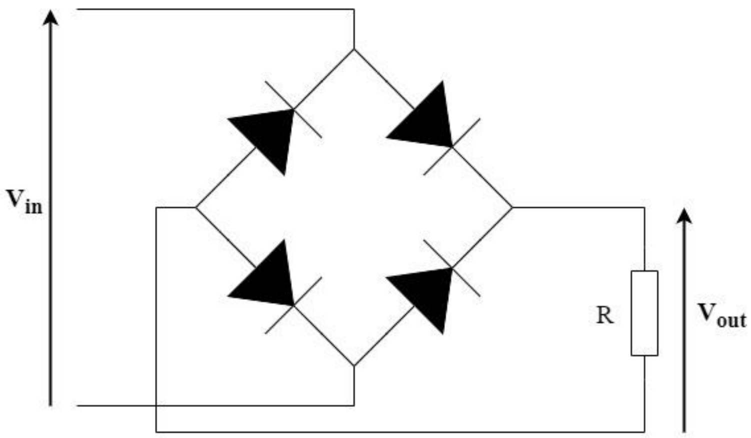


Figure 2 - Wheatstone Bridge Rectifier

$$V_{out} = |V_{in}| - 2\,V_d \qquad \textit{Eq. 1}$$

$$V_{rev} = |V_{in}| - V_d \qquad \textit{Eq. 2}$$

The transformer provided for this project is a step down transformer with a voltage ratio of $230:15$. Thus, when connected to the household power supply, the transformer will provide a $15V$ rms input voltage to the rectifier which has a peak voltage value of $\sqrt{2} \times 15 = 21.21V$. If we assume a value of $0.7V$ for V_d, then the peak voltage of the rectified output would be $21.21 - 2 \times 0.7 = 19.81V$ as given by the *Eq. 1*. The waveforms of the AC input and the rectified DC output of the rectifier bridge is shown in the following Figure 3.

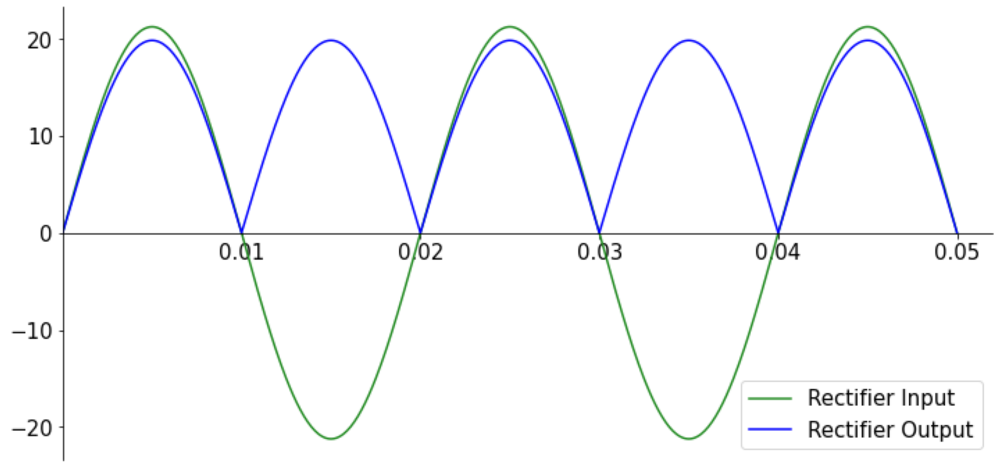


Figure 3 - Rectifier Input and Output

In context of a rectifier bridge, the *Peak Inverse Voltage* (*PIV* rating) is defined as the maximum voltage applied across its reverse-biased diodes. Therefore, by applying the *Eq. 2* at the peak of rectifier input, the *PIV* value can be calculated as $PIV = 21.21 - 0.7 = 20.51V$.

In our design, we have used a commercially available bridge rectifier chip *KBPC3510* to implement the rectification stage. This chip supports up to a maximum average output current of $35A$ which is sufficient for a $10A$ DC power supply. And it has a maximum peak reverse voltage of $1000V$ which is much larger than what our calculations suggested for the PIV.

Section 2.2 – Smoothing

As the name suggests, the smoothing process smoothens out the large voltage variations (ripple) of the rectifier output and converts it into a closer representation of a DC voltage. The implementation of the smoothing process can be simple as adding a capacitor parallel to the output (see Figure 4). The working principle of a smoothing capacitor is simply followed by the usual charging-discharging behavior of a capacitor.

When a capacitor discharges over a resistive load, the resulting voltage variation is characterized by the following Eq.3. Here, V_p is the peak voltage of the rectifier output, and the other terms have their usual meaning.

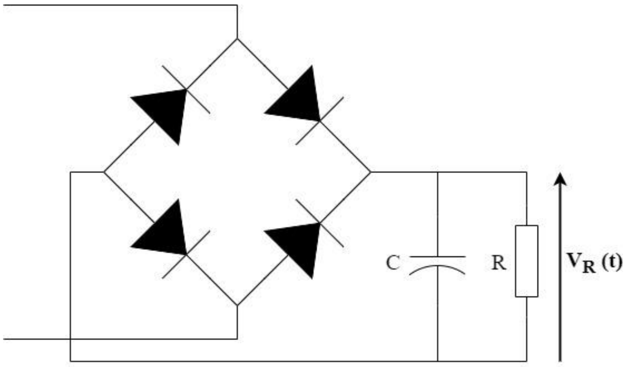


Figure 4 - Rectifier with the Smoothing Capacitor

$$V_R(t) = V_p e^{-\frac{t}{RC}} \Longrightarrow \frac{dV_R}{dt} = -\frac{V_p}{RC} e^{-\frac{t}{RC}} \quad Eq.3$$

In order to have a better smoothing effect, the capacitor has to discharge slowly as possible, thus dV_R/dt gradient value has to become smaller. Although this can be achieved by increasing the value of the RC time constant, since the resistance of the load R is something outside of the power supply, the only controllable factor in the design is the capacitance of the smoothing capacitor C.

However, as C increases, the transient current drawn to charge the capacitor in the first voltage cycle becomes larger and may damage the diodes in the rectifier bridge. Therefore, the smoothing capacitor has to be selected carefully so that the surge current is within a tolerable limit and the level of smoothing is also sufficient.

In our design, we have used three $12mF, 50V$ capacitors in shunt for the smoothing stage, which has a resultant capacitance of $36mF$. Since we are designing a power supple of $10V$ for a maximum current rating of $10A$, the smallest load in interest is a 1Ω resistance. We wrote a python program to analyze the behavior of our smoothing capacitor for the targeted minimum load, and the resulted smoothened voltage is illustrated in the following figure.

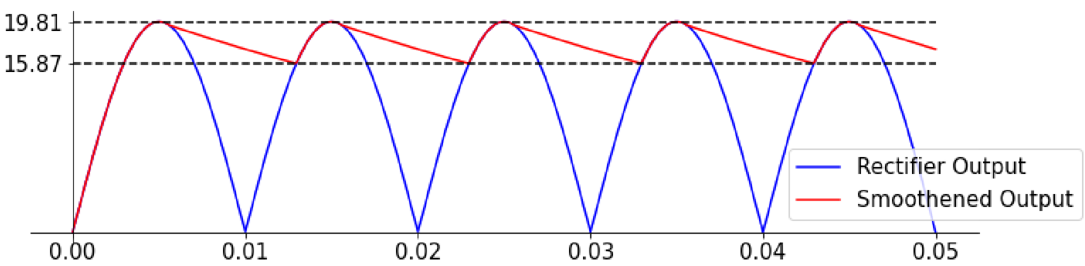


Figure 5 - Rectifier Output and Smoothened Voltage

From the above graph, the maximum ripple voltage (which corresponds to the minimum output load) can be calculated as $19.81 - 15.87 = 3.94V$. Although this voltage ripple may seem much larger, since the resistance introduced by the latter stages (such as the series voltage regulation), the effective load resistance is much larger than 1Ω. Thus, the amount of ripple in the smoothened output becomes much smaller and can be filtered out in the regulating stages.

Section 2.3 – Voltage Regulation

Voltage regulation plays a huge role in a linear power supply. It removes the voltage ripple in the smoothened voltage and provides a constant DC voltage as the output. The following Figure 6 shows a simple circuit for voltage regulation.

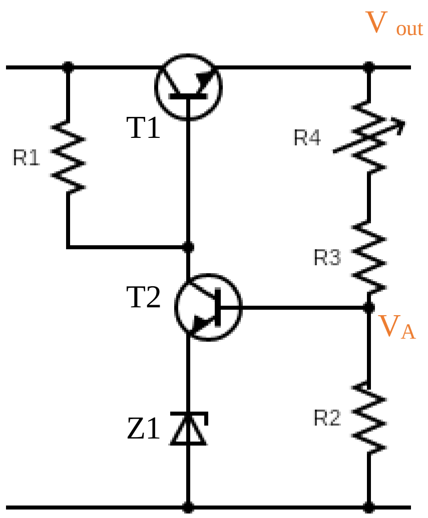


Figure 6 - Regulating Circuit

Here, $Z1$ is a Zener diode with a Zener breakdown voltage (V_z) of $6.2V$. Thus, when the $T2$ transistor is in active region, the voltage across the $R2$ resistor becomes $V_{R2} = V_z + V_{BE,T2} = 6.2 + 0.7 = 6.9V$. Since the V_z and V_{BE} voltages are mostly independent of their respective currents, V_{R2} is nearly a fixed voltage.

Since the base current of $T2$ can be considered as negligible, the path containing $R2$ and $R3$ resistors acts as a voltage divider. Thus, we can select resistors $R2$ and $R3$ such that the output voltage of the regulator (V_{out}) becomes 10V. Therefore, we have chosen $14k\Omega$ and $6k\Omega$ as $R2$ and $R3$ respectively. See the following calculation.

$$\frac{V_{R3}}{V_{R2}} = \frac{R3}{R2} = \frac{6k\Omega}{14k\Omega} \simeq 0.43$$

$$V_{out} = V_{R2} + V_{R3} \simeq V_{R2} + 0.43 \times V_{R2} = 1.43 \times 6.9V = 9.867V \simeq 10V$$

Since the resistors values are not much accurate, to adjust the output voltage, we have added an $R4$ potentiometer to the design.

As we saw earlier, the output voltage given by the smoothing stage has a voltage variation of roughly $15V$ to $20V$. Thus, the excessive 5 to $10V$ input voltage drops across the series $T1$ transistor as its collector-emitter voltage. Therefore, $T1$ transistor has to operate in the active region, and to bias the transistor, the base current is supplied through the $R1$ resistor.

Notice that, since the base current of a transistor and the bias current of a Zener diode are usually smaller, the total current through $R1$ is lower in value. Our power supply can supply an output current up to $10A$ just by keeping the inner currents of the regulation circuit less than 600 mA. Therefore, the power dissipations over $R1$, $Z1$, and $T2$ devices are much smaller, thus power efficiency of the regulation design is maximized.

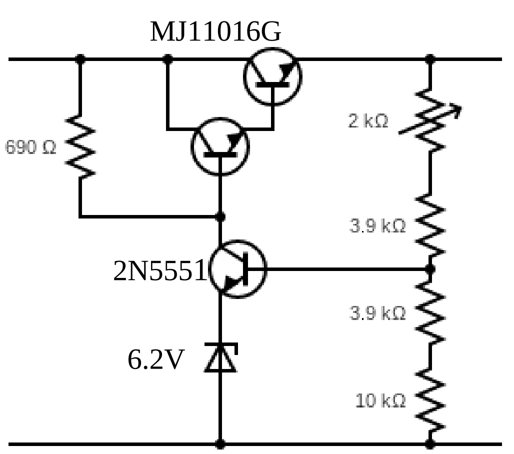


Figure 7 - Voltage Regulating Circuit used in the Design

Instead of using a single transistor for $T1$, we can use a *Darlington Pair* as shown in the Figure 7, which has a much higher current gain than an individual transistor. Therefore, when the output current or the collector current of $T1$ increases from lower levels to $10A$, the resulting base current variation of $T1$ tends to be smaller for a Darlington pair than for a transistor. This scenario keeps the Zener current steady, thus making the output voltage stable at $10V$.

For $T1$, we have selected the *MJ11016G* Darlington pair which can support up to a maximum collector current of $30A$. For the $T2$ transistor, we selected the *2N551* transistor which has $600mA$ maximum rating for collector current and $0.7V$ forward base-emitter voltage drop. In the Sri Lankan market, we were unable to find $14k\Omega$ resistors for $R2$, so we replaced it with two series $10k\Omega$ and $3.9k\Omega$ resistors. Also, we used a $3.9k\Omega$ resistor for $R3$ and a $2k\Omega$ trimmer potentiometer for $R4$. For $R1$, we used a 690Ω resistor which is sufficient to provide the base current for $T1$ and the collector current for T2 (or the Zener current for $Z1$). Our final voltage regulating circuit is shown in the Figure 7.

Section 2.4 – Current Limiting

The maximum current rate for the power supply is $10A$. If the load draws more current than that, it may cause damages to the load as well as to the components of the circuit. The current limiting circuit is designed to avoid such situations.

To limit the current, we have added three paralel 0.22Ω resistors (available in the market) series with the $T1$ transistor, thus a total resistance of $= \frac{0.22}{3} \simeq 0.07\Omega$. Therefore, when the output current exceeds $10A$, the voltage across the resistors becomes larger than $0.7V$ forward-biasing the $T3$ transistor. Note that the 2N5551 transistor which is used for $T3$ has a base-emitter forward bias voltage of $0.7V$.

Then, this added path from the base of T3 to the output will pass a portion of the bae current of T3 reducing its collector current thus limiting the output current.

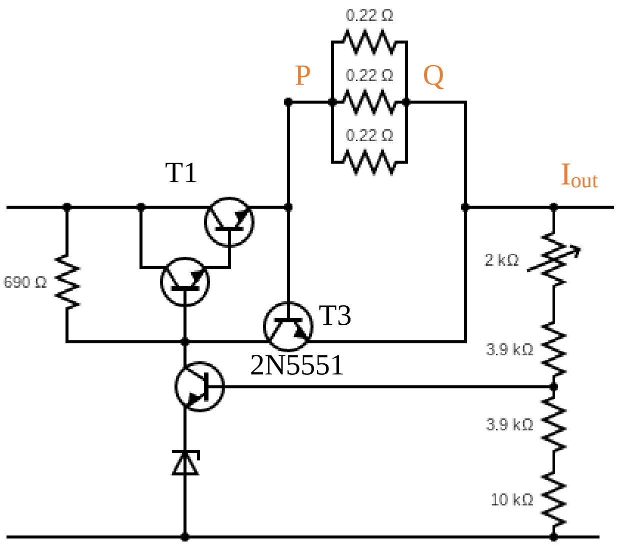


Figure 8 - Current Limiting Ciruitry

Section 2.5 – Circuit Protection

The Linear Power Supply we built can provide $10V$ DC current with a maximum current rating of $10A$. Therefore, this is a high-power power supply, hence we have to consider the necessary safety measurements for the protection of both the consumer and the power supply unit itself. We divided our safety mechanisms into three main categories, which are *Overvoltage Protection*, *Short Circuit Protection*, and *Thermal Protection*.

Section 2.5.1 – Overload Protection

The current limiting circuit discussed in the Section 2.4 – Current Limiting will limit the maximum output current of the circuit to $10A$ providing the overload protection.

Section 2.5.2 – Short Circuit Protection

We use a $12A$ $250V$ *Fast Blow Glass Fuse* (AC fuse) for short circuit protection. The maximum current that can pass through this fuse is 12A. We intend to connect this fuse to the $15V$ transformer output. Therefore, if the input current goes beyond $12A$ (which will occur in a short circuit situation), then the fuse will burn and prevent further conduction eliminating any damage to the circuit. However, there is a possibility that the circuit will experience a higher current for a short amount of time until the fuse wire melts. This current will go through the KBPC3510 bridge rectifier, MJ11016G transistor, and the set of capacitors. We choose those components to tolerate such high currents. Therefore, by using a fuse and proper components, short circuit protection can be achieved.

Section 2.5.3 – Thermal Protection

For the thermal protection, we use *Heatsinks***,** *Fans and Ventilation*, and *proper PCB Design***.**

In the power supply, we need heatsinks for MJ11016G transistor and for KBPC3510 bridge rectifier because a higher current of $10A$ will pass through both of these equipment in extreme conditions.

Therefore, we use heatsinks to promote effective thermal conductance. Also, we used thermal paste and thermal conducting current insulated mica sheet to make the contact between the heatsinks and the components. Furthermore, we mounted those heatsinks using spacer screws to avoid any contact with the PCB, and for an efficient heat transfer, they were placed directly under a cooling fan.

Then there is the $12V$ DC fan to ventilate the power supply unit. There is a separate PCB to provide a $12V$ DC supply for this fan. Also, we added air vents in the enclosure design for the heated air to flow outside the device.

In the PCB, the copper traces that carry the load current are designed to be wider and shorter, thus they have a less path resistance. Therefore, in extreme conditions, the PCB will not get overheated. Also, we used proper wire connectors with necessary current ratings (for the devices connected to the external heat sinks) to avoid any overheating and other failures.

Section 3 – Results

After the designing stage and the simulation stage, first we built our circuit on a dot board and check the functionality of the power supply unit. Dot board implementation was a success, so we printed the PCB and built our circuit. Then we checked the circuit for its functionality and got the results we needed.

Checking the Circuit

Here, we checked for any error in the circuit, including the component soldering, electrical continuity of the paths, wire connections, and component pin configurations.

Verifying the Functionality

Then, the circuit was checked for its functionality. First, we connected the transformer and measured the output voltage for the no load condition and then adjusted it to be $10.2V$ using the potentiometer. Then, we checked the voltages at several places in the circuit and compared the values with the simulation. Afterwards, we got the measurements for the data sheet by measuring the output voltage and input current for several output loads.

Load vs Output Voltage.

The output voltage vs. resistive load curve is given in the Figure 9 below. There is no observable ripple in the output voltage for any load. However, there was an $800mV$ noise throughout the checking process, most probably due to the internal noise of the oscilloscope.

If we look at the results, we can observe that if we reduce the load from no load condition to 100 ohms, we can observe only slight 0.1V change in the output voltage. (We set the no load condition voltage to become 10.2V) Now if we further reduce the load up until 1.2 ohms there is a 0.45V voltage drop (10.1V to 9.65V) So, we can observe a total 0.55V voltage drop in output terminals.

The reason for this voltage drop is when the load draws a higher current, there will be a higher current through the Zennor diode. (Which acts as voltage reference) This will reduce the output voltage because in the output voltage calculations we neglect this current assuming it is negligible. Since even at 1 ohms load we can get a voltage around 9.6V we think our power supply unit is up to its required specifications.

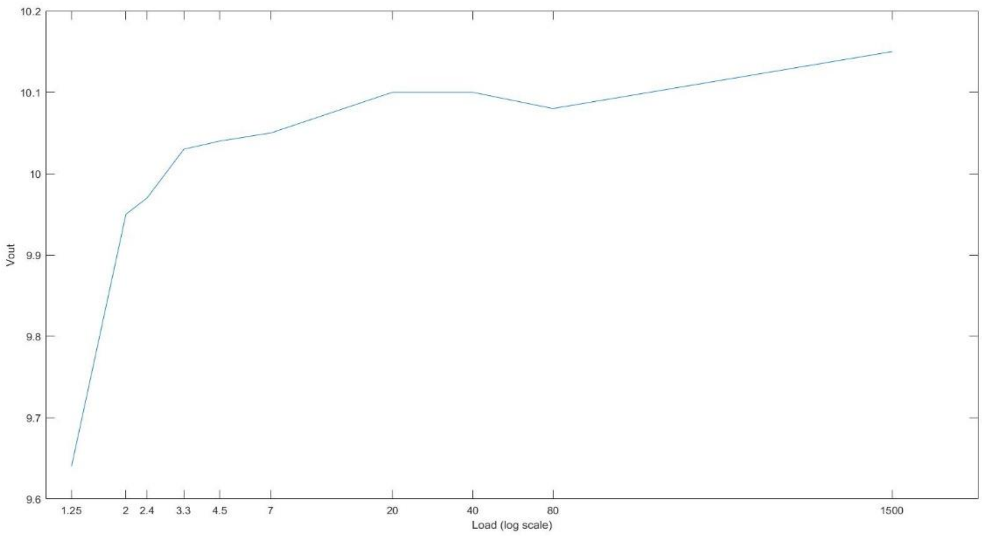


Figure 9 - Output Voltage vs. Resistance of the Load

Load vs Power

In this curve we can observe the relationship between load and the power. As we can observe both input and output power increase when the load reduces. We calculated the efficiency using those values and (below table) you can see that the efficiency increases when as we reduce the load.

PCB Design

The PCB of our power supply unit is a high-power PCB because it carries larger currents as high as $10A$. The main problem associated with the high-power PCBs is that they can easily get heated up due to the path resistances and the high-power components integrated onto the PCB. These high temperatures can damage and cause the temperature sensitive components to behave unusually reducing the reliability and the durability of the product. Although there is a number of techniques to avoid the heating of a PCB, the basic solution is to place the high-power components outside the board, most probably, attached to the enclosure, so they can be cooled with heatsinks and dedicated ventilation.

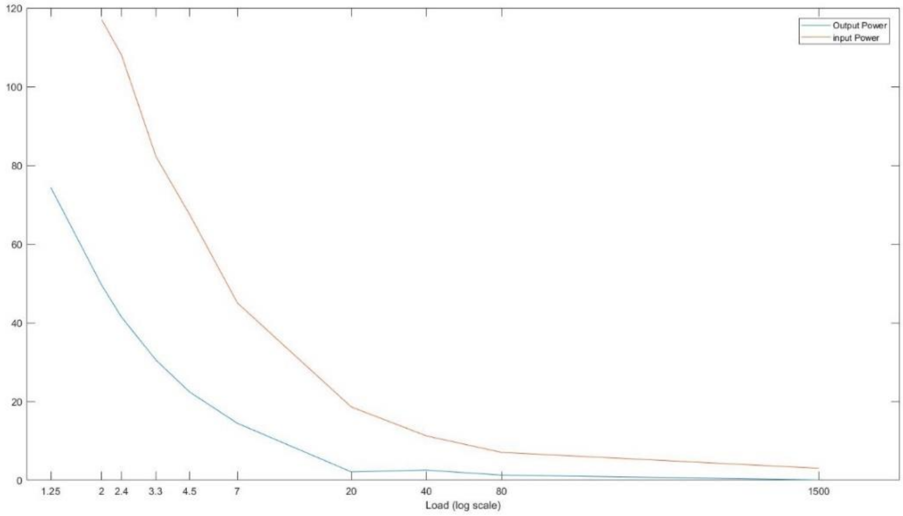


Figure 10 - Output Power and Input Power vs the Load Resistance

Reducing the resistance of the high-current PCB traces also helps to reduce the heating of a PCB. This can be achieved by designing those paths to be shorter, thicker and wider. However, since customizing the trace thickness may come with an additional cost, it is more convenient to increase the width of the copper traces. IPC-2221 standard provides some formulas to calculate the PCB trace width when the copper thickness and the maximum current rating are known. Our PCB was printed using a typical $1\ oz$ copper board (copper thickness $= 35\mu m$) and it has to support a $10A$ maximum current. For these specifications, a trace width of $7.19mm$ is suggested by the IPC standards.

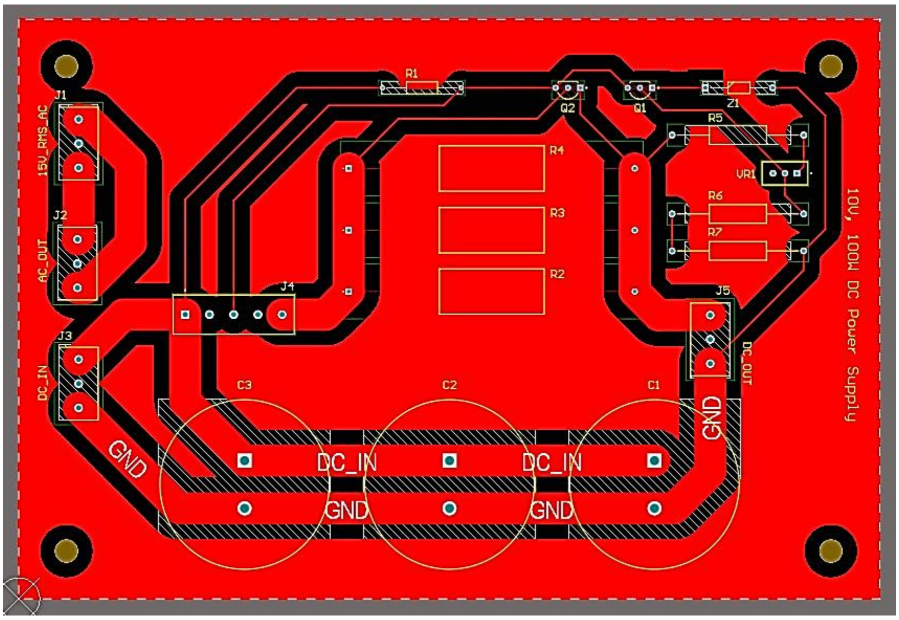


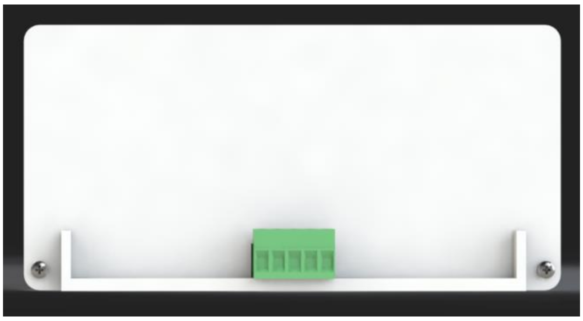
Figure 11 - PCB Design

Enclosure design

An enclosure for the Linear Power Supply was designed using SOLIDWORKS software. Following are the images of the enclosure design in different orientations.

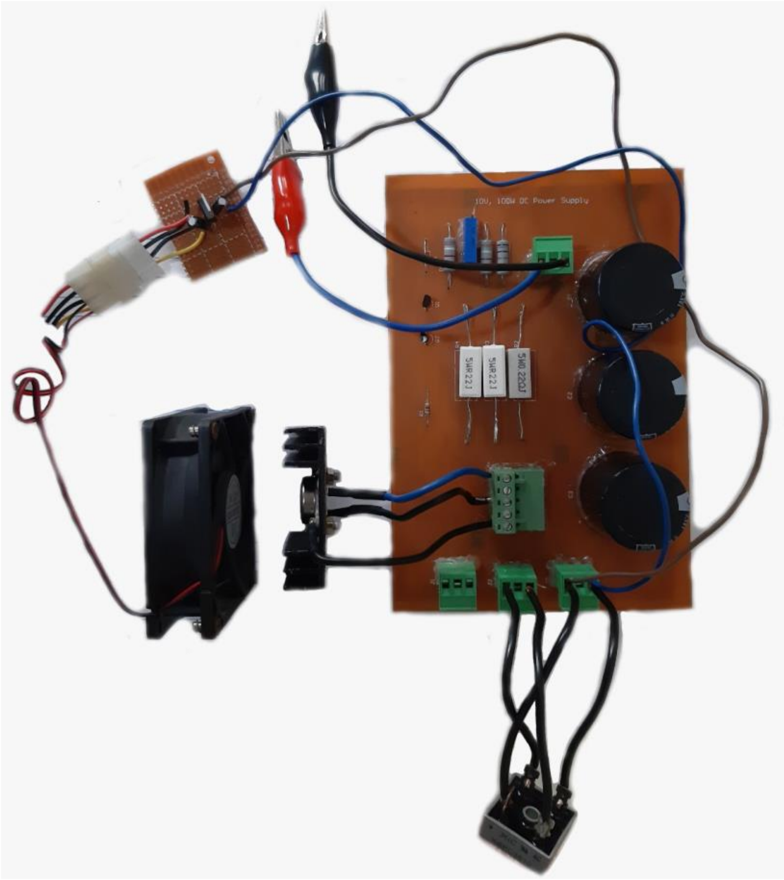
Several key factors were considered when designing the enclosure,

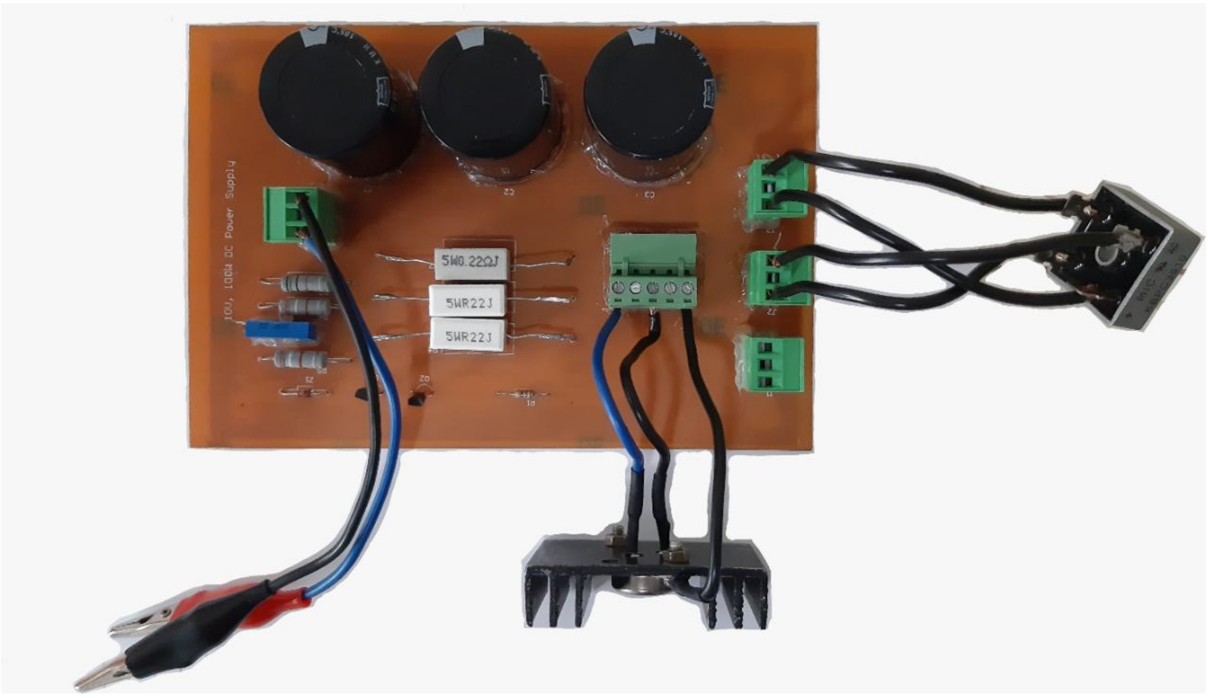
1. The enclosure is a two-part design. The design easily allows working on the PCB without the top part. The top part can be screwed to the base using 4 screws.
2. Tests done on the final design showed intense heat dissipated from the power transistor and bridge rectifier. Heatsinks were connected to both components to improve heat dissipation. The enclosure was designed to allow proper heat flow using vents. A vent was included on the top of the enclosure to house the 12V DC fan that cools both the components said above. Vents were included in the sides and bottom to improve the heat flow.
3. The exterior design of the Linear Power Supply was designed with minimalism in mind. A 5-way screw connector was used to connect the 230 V AC input and 10V DC output. The screw at the center is Not Connected to ensure space between AC and DC connectors. Front of the design contains only the fuse and switch.



Appendices

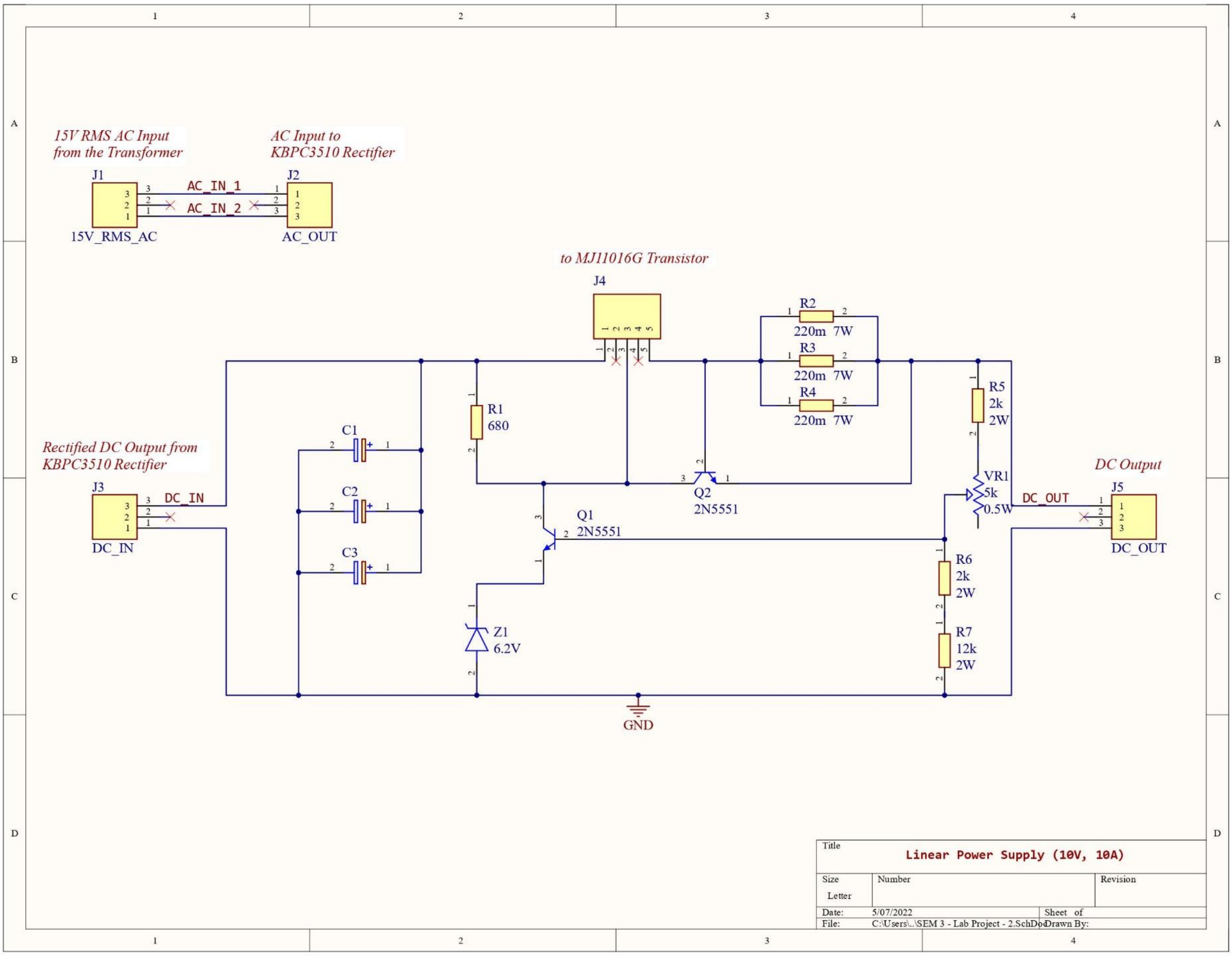
Appendix A – Our Product



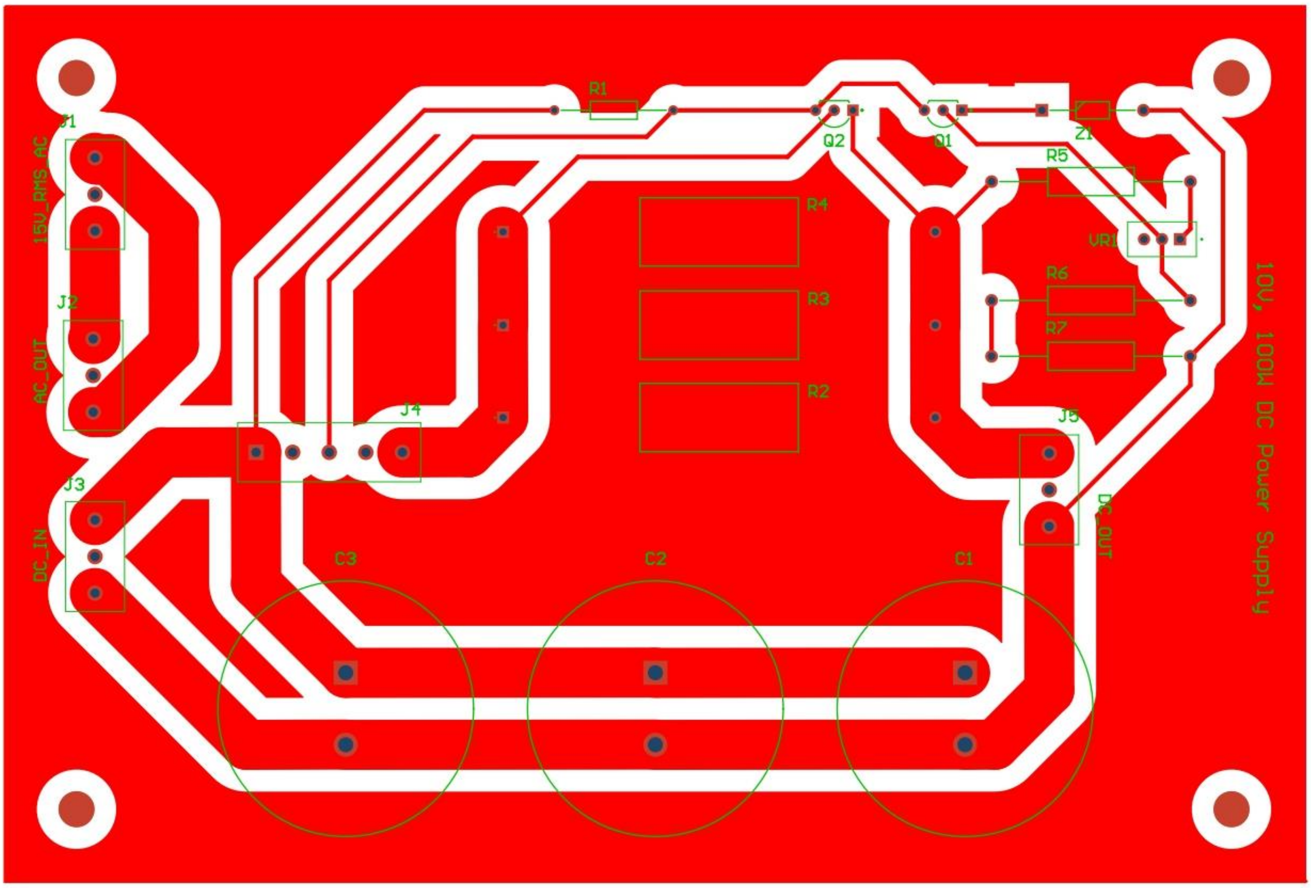


Appendix B – PCB Design

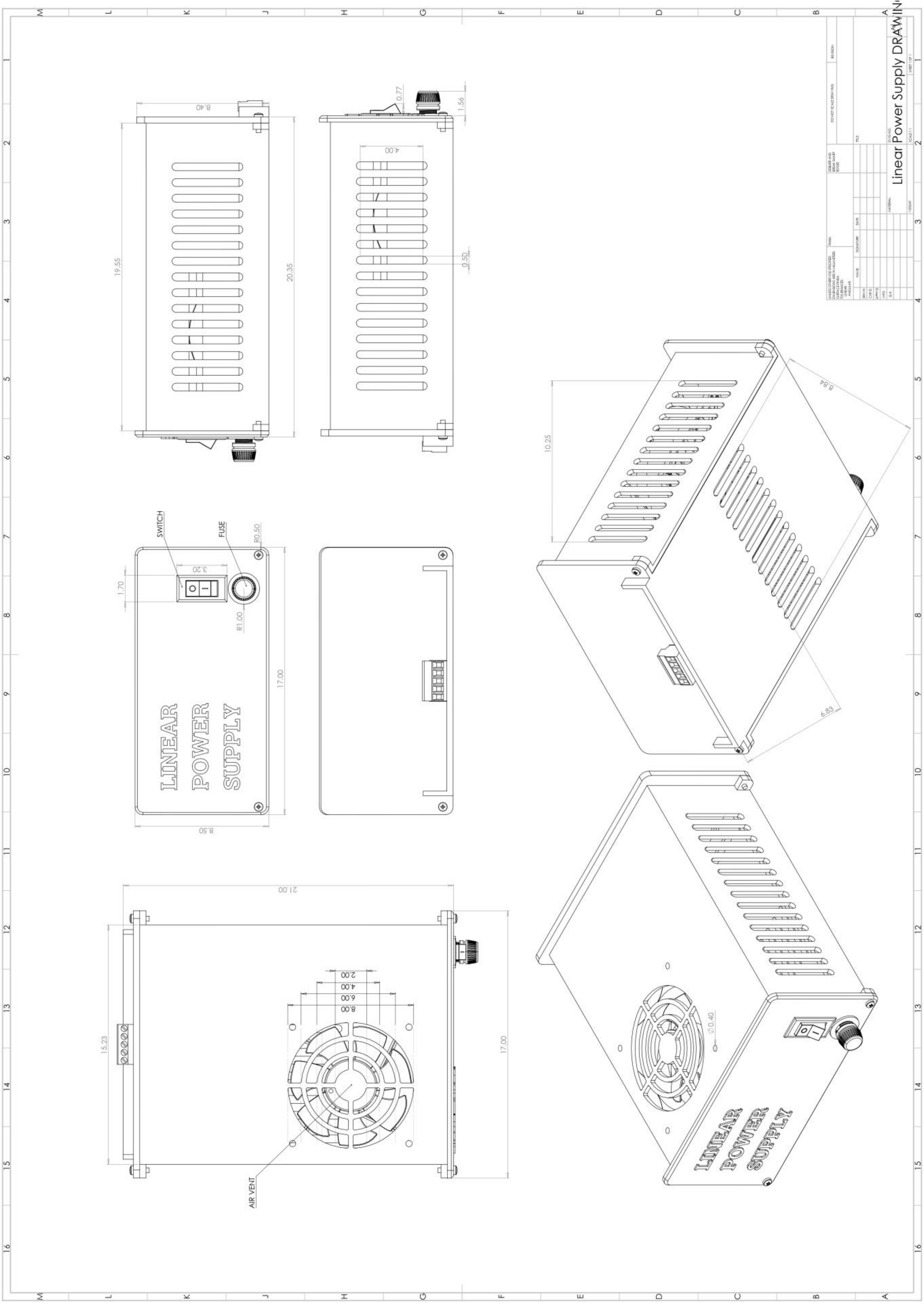
Appendix B.1 – PCB Schematics



Appendix B.2 – PCB Layout



Appendix C – Enclosure Design



Appendix D – Data Sheet of the Linear Power Supply

Data Sheet Linear Power Supply

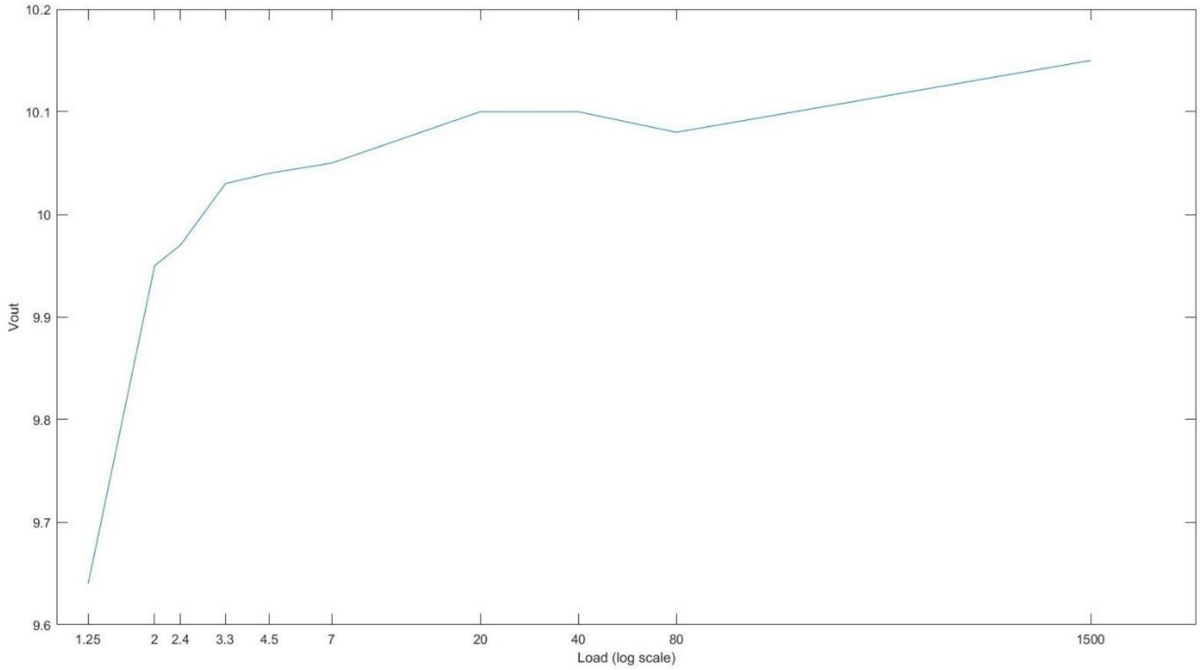
Made in Sri Lanka

University of Moratuwa

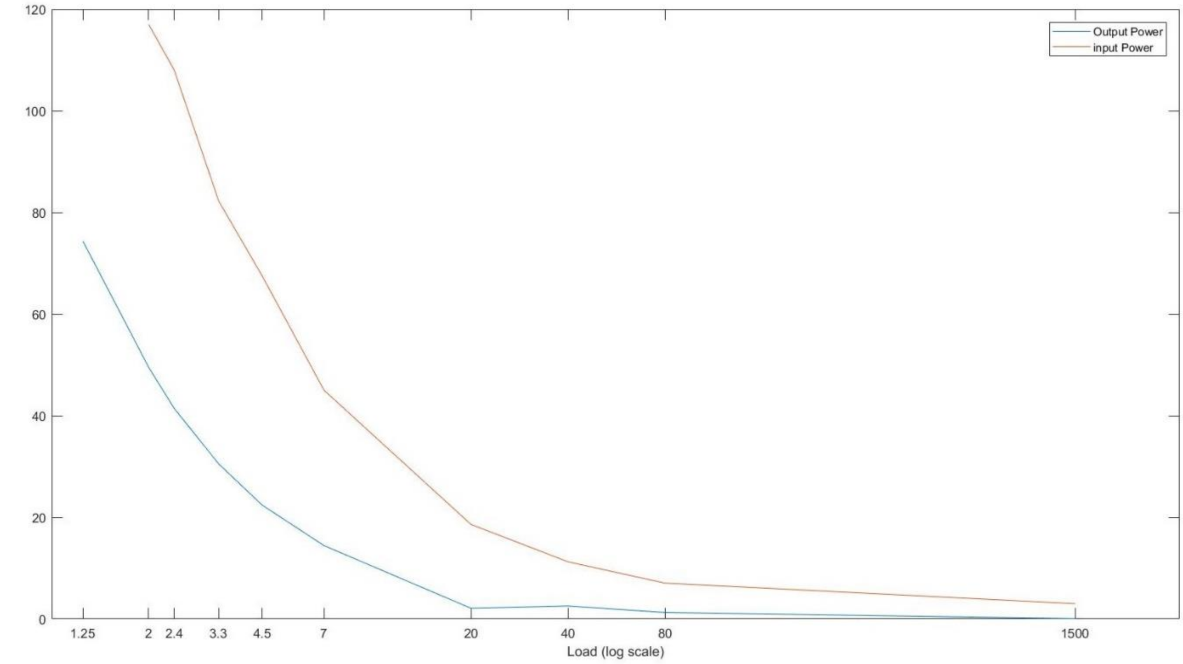
Specifications – Linear Power Supply

VAC INPUT • 230 V • Frequency Range 50 HZ	**OVERVOLTAGE PROTECTION** • Automatic Voltage limit **OVERLOAD PROTECTION** • Automatic Current limit **SHORT CIRCUIT PROTECTION** • Fusing
VDC OUTPUT • 10.2 – 9.6 VDC • Current Rating – 10A	**FUSING REQUIREMENTS** • Fuse Rating – 12A (AC) • 12A 250V Fast Blow Glass Fuse
OUTPUT RIPPLE • Negligible	**EFFICIENCY (TYPICAL)** • Provided Below

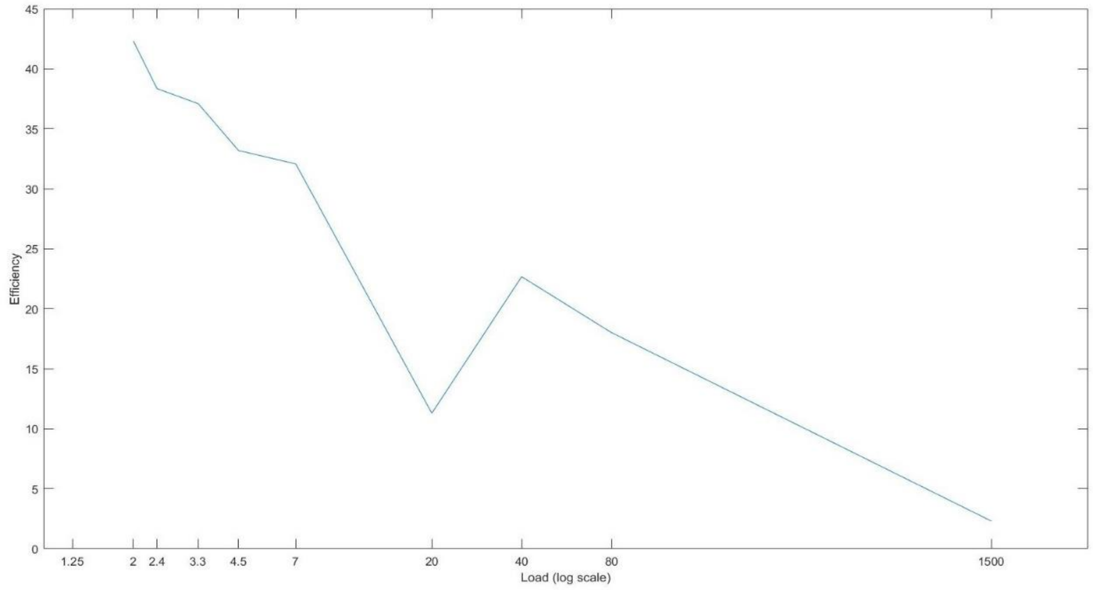
Load vs Output Voltage



Load vs Power



Load vs Efficiency



Case Size