A Minor Project Report on
DC-DC Buck Converter

By
Jay Lokhande

INDEX

1. INTRODUCTION
    - 1.1 Preface
    - 1.2 Organization of Report
2. METHODOLOGY (PCB DESIGNING)
    - 2.1 PCB Composition & Materials
    - 2.2 Design Requirement
    - 2.3 Schematic
    - 2.4 Footprint and Layout
    - 2.5 Printing PCB
    - 2.6 Etching and Drilling
    - 2.7 Soldering
3. IMPLEMENTATION
    - 3.1 Components
    - 3.2 Constructed Circuit
    - 3.3 Principle of Operation
    - 3.4 Costing
4. RESULTS
    - 4.1 Observation
    - 4.2 Conclusion
    - 4.3 Merits

LIST OF ACRONYMS

DC: Direct Current

PCB: Printed Circuit Board

CAD: Computer-aided Design

LED: Light Emitting Diode

LPI: Liquid Photo Image-able

$FeCl_3 \cdot H_2O$: Ferric Chloride

1. INTRODUCTION

1.1 Preface

In today's rapidly advancing technological landscape, the demand for efficient and compact power management solutions is ever-increasing. DC-DC converters play a pivotal role in various applications ranging from portable electronic devices to renewable energy systems. Among these, the buck converter stands out as a versatile and widely utilized topology due to its simplicity, cost-effectiveness, and high efficiency.

This project report delves into the design, implementation, and evaluation of a DC-DC buck converter employing the MC34064 integrated circuit (IC). The MC34064 is a monolithic control circuit specifically tailored for DC-DC converter applications, offering features such as low start-up current, precise reference voltage, and adjustable output voltage.

The primary objective of this endeavour is to analyse the performance characteristics and efficiency of the DC-DC buck converter utilizing the MC34064 IC under various operating conditions. Through theoretical analysis, simulation studies, and practical experimentation, we aim to gain insights into the converter's behaviour, assess its performance metrics, and identify areas for optimization.

The report is structured to provide a comprehensive understanding of the design methodology, circuit operation, control strategies, and performance evaluation techniques employed in the development of the DC-DC buck converter. Additionally, it aims to serve as a valuable resource for engineers, researchers, and enthusiasts seeking to delve into the realm of power electronics and converter design.

By documenting our findings, challenges encountered, and lessons learned throughout the project, we endeavour to contribute to the collective knowledge base in the field of power electronics and foster innovation in energy-efficient conversion technologies. Through meticulous experimentation and analysis, we aspire to validate the efficacy and practicality of the MC34064-based DC-DC buck converter while offering insights into its potential applications and avenues for future enhancement.

2.METHODOLOGY (PCB DESIGNING)

PCB is an acronym for Printed Circuit Board. It is a flat plate or base of insulating materials that contains a pattern of conducting material and components. A PCB allows signals and power to be routed between physical devices. A PCB is a layer cake where alternating layers of different materials are laminated together with heat and adhesive such that the result is a single object.

2.1 PCB Composition & Materials

A copper-clad substrate or polyimide-based film is often used for flex circuits. A solder mask is also used, which is a technique where everything on the circuit board is coated with an epoxy-based LPI or cover coat for flex, except the contacts to be soldered.

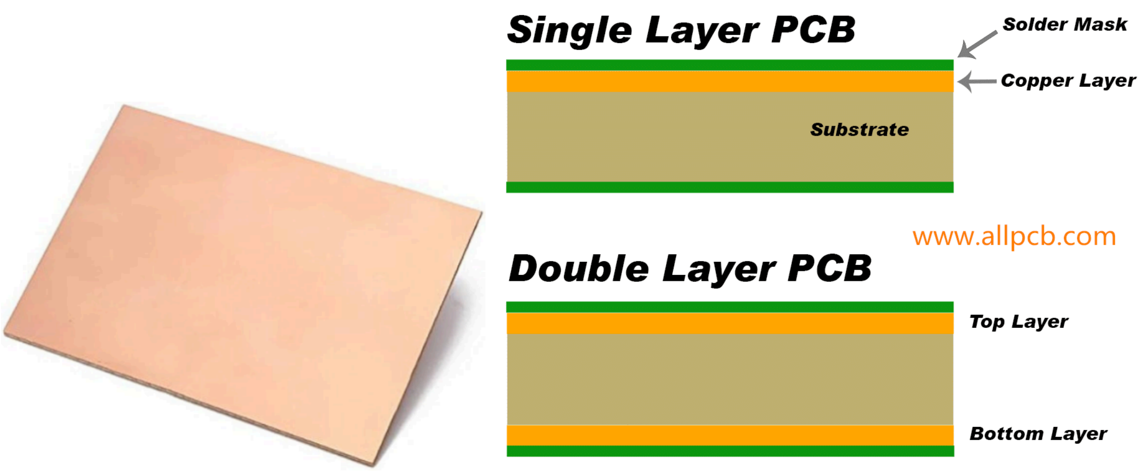

Fig 2.1 PCB Layer

2.2 Design Requirement

Before starting the design of the board within the CAD tools, it is necessary to ensure that the library part designs are completed. For the schematic, this means creating logic symbols for the parts that will be implemented; resistors, capacitors, inductors, connectors, and integrated circuits (ICs). These are combined to create more sophisticated devices such as logic gates, precision amplifiers, adders, and multipliers. Logic development is the key factor that define the outcome of the project.

The design of the project consists of 1 resistor, 5 capacitors, 2 inductors, 1 8-pin IC, 2 screw terminals, 1 potentiometer, 1 diode. All the components we use for designing are through hole footprint based.

2.3 Schematic

A schematic, or schematic diagram, is a representation of the elements of a system using abstract, graphic symbols rather than realistic pictures. In an electronic circuit diagram, the layout of the symbols may not resemble the layout in the circuit. With these parts ready for use, begin by organizing them on schematic sheets within the CAD tools. At this stage we develop a schematic with a software platform such as Tinker-CAD, EasyEDA and KiCAD. These are the few open-source platform to design PCB. We opted for EasyEDA an open-source software.

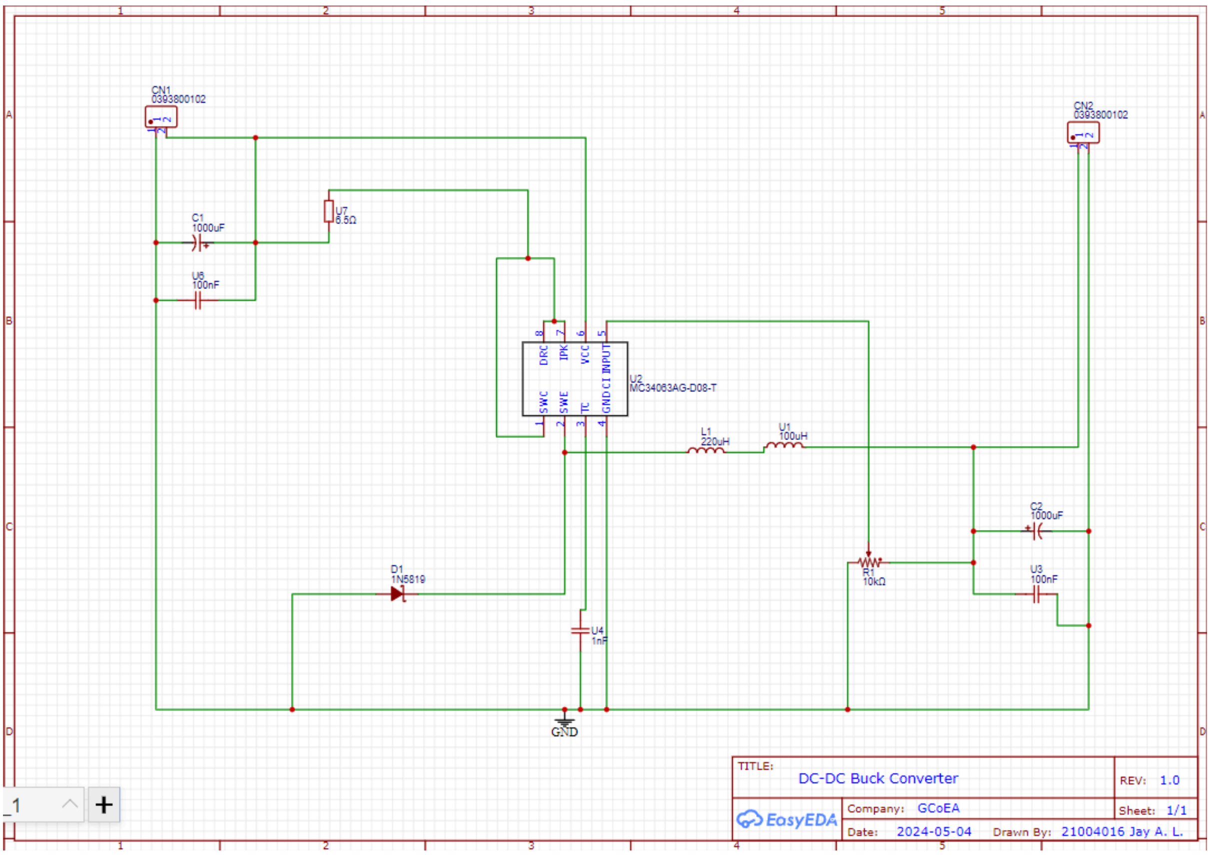

Fig 2.2 Schematic Diagram

2.4 Footprint and Layout

The PCB footprint is defined as the physical interface among electronic components or land pattern and printed circuit boards which also comprises of the information of documentation such as reference, polarization mark, and outline. Every component that will appear on a circuit board layout will have a PCB footprint. Footprint pad guidelines are rules or suggestions that stipulate how pads should be spaced from other pads, other surface elements, drill holes and the board edge. They also define how pads for a single component should be arranged. All PCB routing need to have copper that connects components on the surface layer or internal layers, known as traces. Some of the important design routing requirements:

1. The current carrying capacity of traces, as high current boards can require large traces or even polygons.

2. Trace width to be used in the board, which will ensure manufacturability and will affect crosstalk.

3. Any controlled impedance signals, which require specific width that must be set based on the PCB stack-up.

4. Routing PCB topology, which will determine how traces branch connect to multiple components.

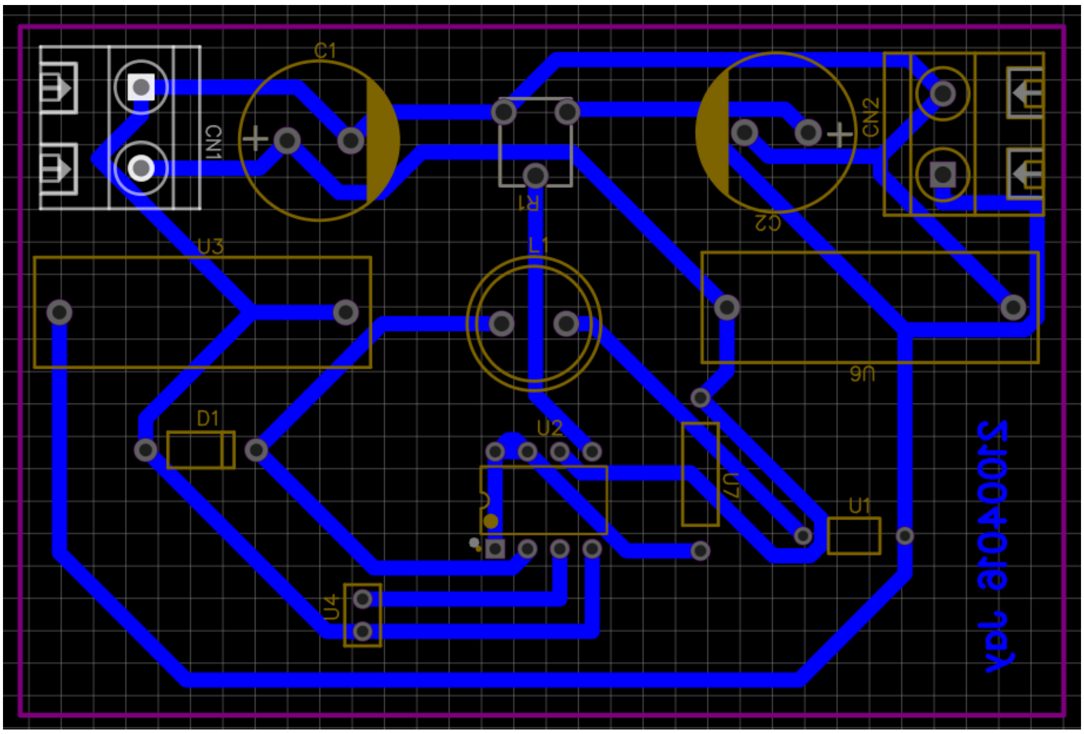

Fig. 2.3 Tracks and Component position on EasyEda software

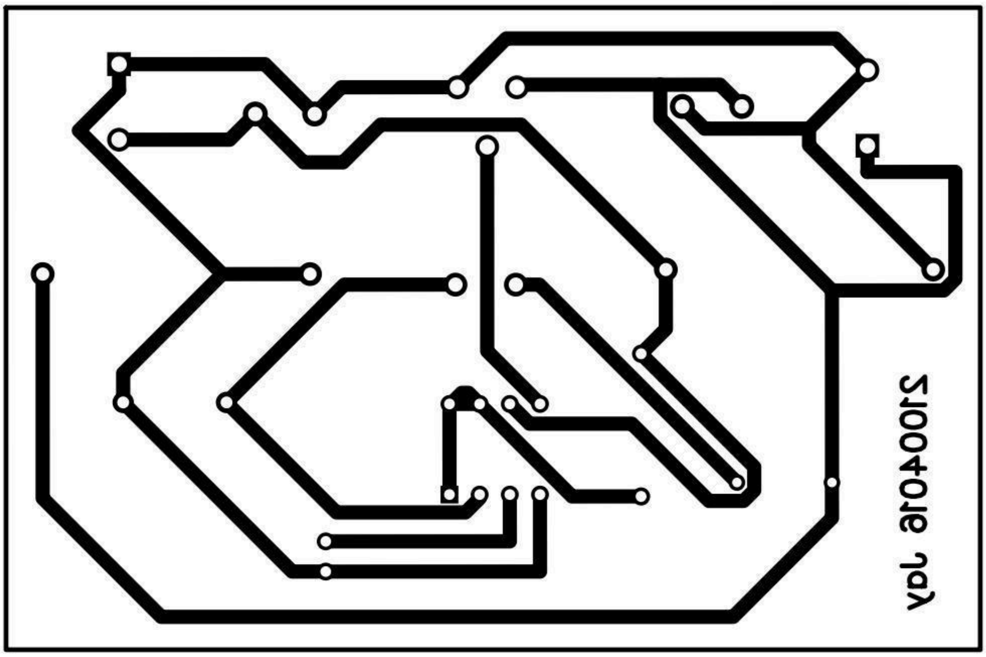

Fig. 2.4 Layout to be printed on Copper Clad PCB Board

2.5 Printing PCB

Printing a circuit over a copper-clad can be done by different methods. The industries standard uses laser or ink-jet printers to print the layout over clad. This process is also termed as photo lithography. Convenient methods of printing are Press and Draw.

Pressing: Printing the image file of the layout on a bond parchment. The parchment heated via “heating lamp” or “Electric Iron Press” while pressed face down on copper-clad in order to imprint its ink onto the other surface. And sudden treatment in cold bath lets the ink settle on the clads surface. Thus, printing circuit routes on the PCB. This forms the blueprint that decides which pieces of copper must be removed from the board. Drawing: An alternative to pressing method is to draw the tracks and footprint on the PCB according to the routes. the drawing of the routes on the PCB are done by permanent marker so that the ink doesn’t spread or either get breaks while Etching.

2.6 Etching and Drilling

PCB Etching is the process of removing unwanted copper from a printed circuit board. There are two type of etching these are dry etching and wet etching. We opted for wet etching.

Usually carried by treating the copper clay with $FeCl_3 \cdot H_2O$ solution for 30 minutes to an hour. Ferric chloride is a corrosive, acidic chemical compound that will eat away all copper on the board that is not protected by the marker's ink Once all of the excess copper has been removed from the PCB, only the required circuit track remains. 1 equivalent of Cu requires a minimum of 2 equivalent of $FeCl_3 \cdot H_2O$. Drilling the hole so that through hole components could be mounted on the PCB. The imprinted pad or through hole footprint on the layout locate the point to drill. Holes are drilled with a drill bit of size 1 mm point.

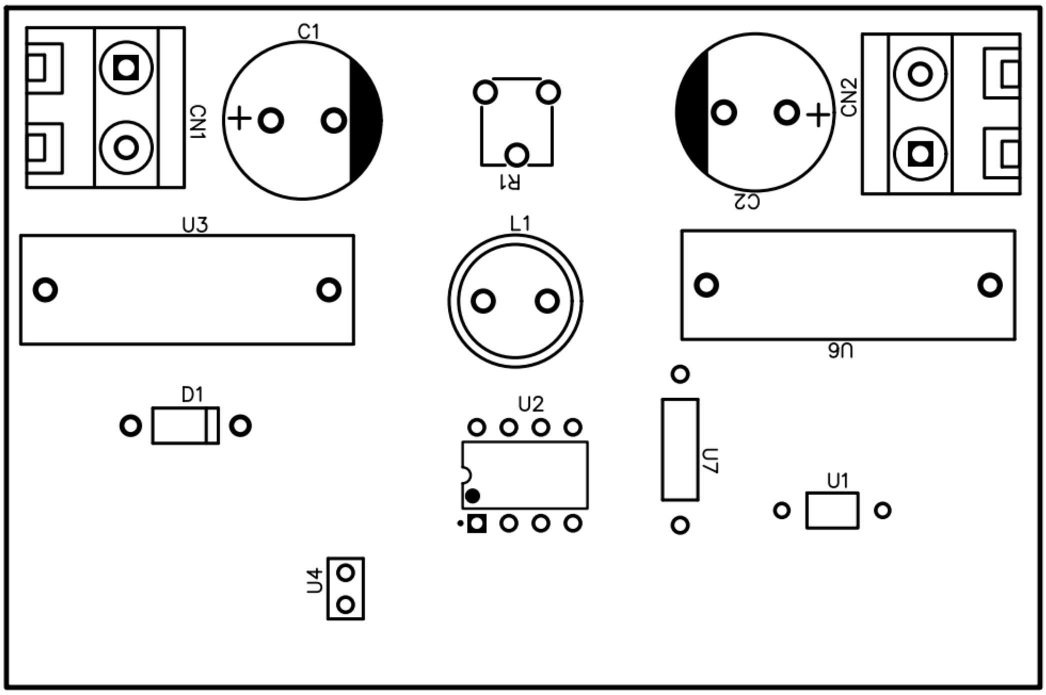

Fig.2.5 Top Silk Layer

2.7 Soldering

Soldering is a process used for joining metal parts to form a mechanical or electrical bond. It typically uses a low melting point metal alloy (solder) which is melted and applied to the metal parts to be joined and this bonds to the metal parts and forms a connection when the solder solidifies. the leads of the component are passed through holes in the PCB and then soldered to a `pad’ on the reverse side of the PCB. Soldering wire is a metal alloy wire used to join two or more metal surfaces together by melting the wire to form a permanent bond. The composition of the soldering wire typically includes a combination of tin, lead, and other metals such as silver, copper, and zinc. The most common wire is `Leaded Solder Wires’ having a diameter of 1mm, having melting point of 180 degree Celsius and composition of Sn60, Pb40.

3. IMPLEMENTATION

3.1 Components

Table 3.1 Components used

Components	Value	Quantity
IC MC34063	-	1
Capacitor	1000µF	2
Capacitor	100nF	2
Capacitor	1nF	1
Inductor	200 µH	1
Inductor	100 µH	1
Diode 1N5819	-	1
Potentiometer	10K	1
Resistor	6Ω	1
Screw Terminals	2-pin	2
Total Components		13

3.2 Constructed Circuit

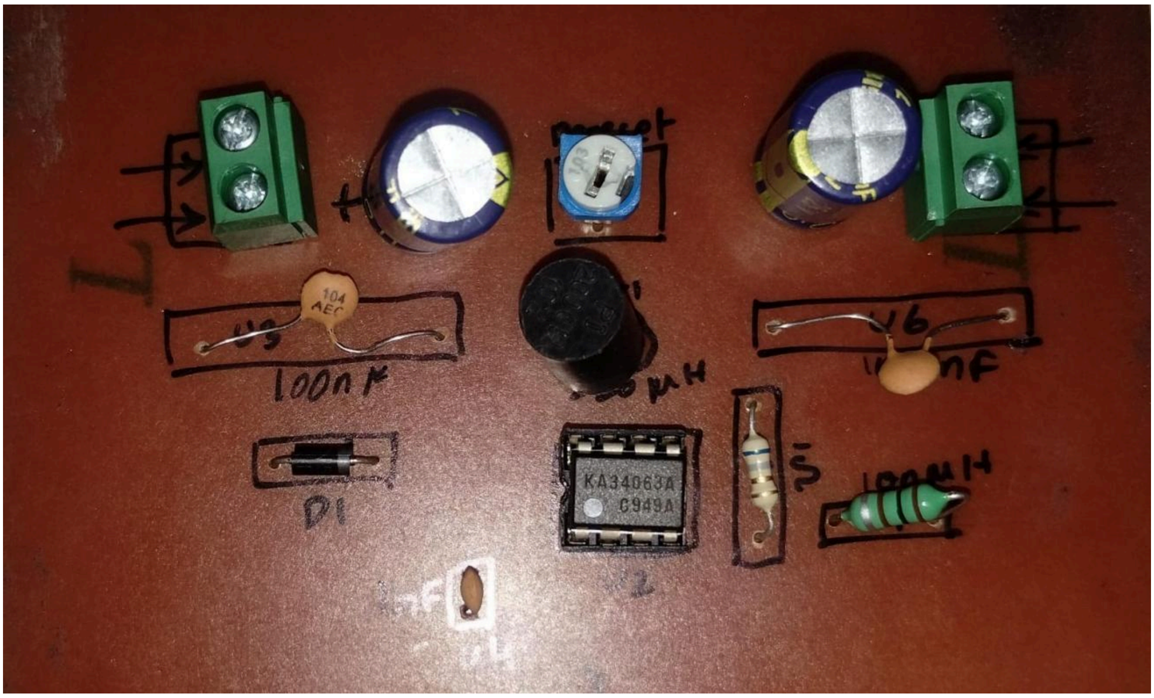

Fig. 3.1 Components soldered on PCB from above

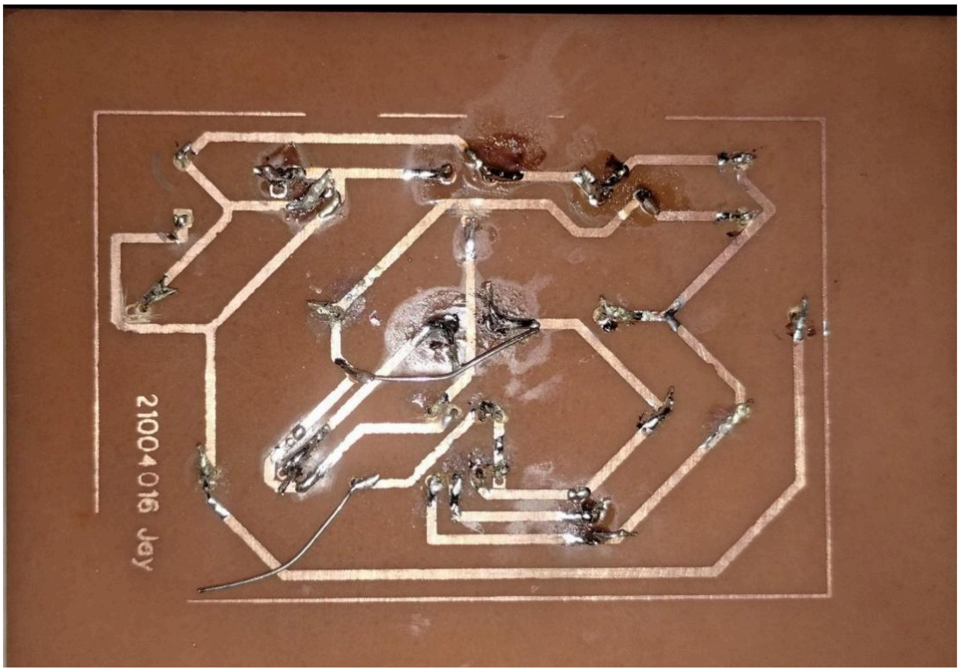

Fig. 3.2 Components soldered on PCB from below

3.3 Principle of Operation

The increase and decrease in voltage are controlled by the variable resistance i.e. the potentiometer.

Upon turning the knob of potentiometer clockwise the voltage increases and upon turning it anti-clockwise it decreases.

As stated by Ohm's Law, the increase in resistance by potentiometer the voltage increases as well, at constant current.

3.4 Costing

Table 3.2 Cost of Components used

Components	Value	Cost
IC MC34063	1	20
Capacitor	1000µF x 2	10
Capacitor	100nF x 2	4
Capacitor	1nF x 1	2
Inductor	200 µH x 1	6
Inductor	100 µH x 1	6
Diode 1N5819	1	2
Potentiometer	10K x 1	4
Resistor	6Ω x 1	2
Screw Terminals	2-pin x 1	13
PCB	1	24
Photo Paper	1	10
Ferric Chloride	1	50
Total Cost		153

4. RESULTS

4.1 Observation

Table 4.1 Results observed at constant current

	Input Voltage	Potentiometer Resistance	Output Voltage
1	3.2 V	$\sim 0.0\Omega$	1.4 V
2	3.1 V	$\sim 5.0K\Omega$	2.3 V
3	3.3 V	$\sim 10.0K\Omega$	3.2 V

4.2 Conclusion

Thus, we have implemented and observed the results of a DC-DC Buck Converter using IC MC34063, and observed how it provides variable output voltage.

4.3 Merits

Efficiency: The DC-DC buck converter, utilizing the MC34064 IC, offers high efficiency in power conversion, ensuring minimal power loss during voltage regulation. This efficiency translates to optimized energy utilization, making it an ideal choice for applications requiring precise control over motor speed and LED intensity.

Voltage Regulation: The buck converter provides stable and regulated output voltage, facilitating precise control over the speed of the motor and the intensity of LEDs. This ensures consistent performance and prevents voltage fluctuations, which can adversely affect the operation of sensitive electronic components.

Dynamic Response: The buck converter exhibits rapid transient response, enabling quick adjustments to changes in load conditions or input voltage variations. This dynamic response capability is particularly advantageous in motor control applications where rapid acceleration or deceleration is required, as well as in LED lighting systems that demand instantaneous brightness adjustments.