Design Report: Regulated DC-DC Buck Converter

Author: Shravana H S **Date:** 2025-10-04 **Project:** Hardware Intern Assignment

Github documentation

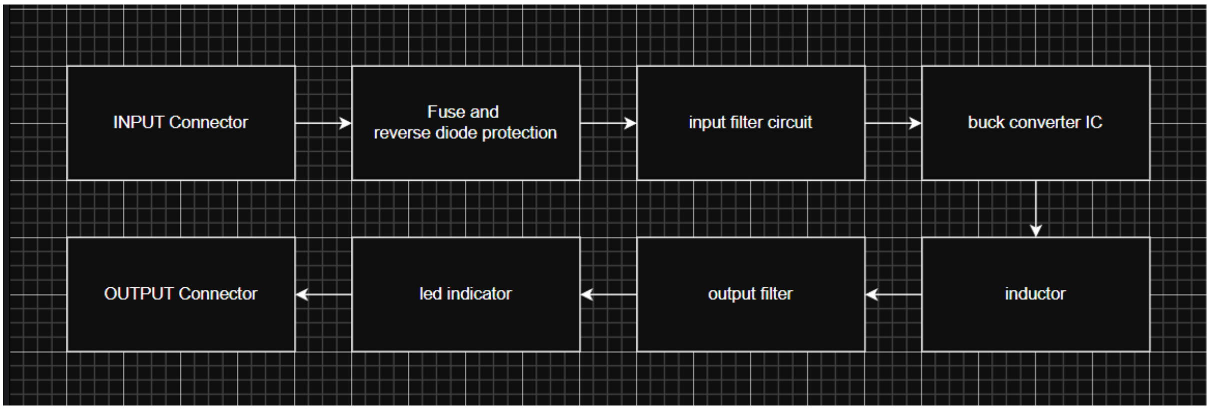
Gerber, BOM and design Files

1. Introduction and System Overview

This document details the design of a regulated DC-DC step-down (buck) converter based on the LM2596 monolithic switching regulator. The primary objective was to create a reliable and versatile power supply capable of converting a higher input DC voltage to a lower, stable, and adjustable output voltage. The circuit is designed to handle a **1A load** and features input protection and a power indication LED. The final design is a compact, two-layer PCB using primarily surface-mount components for a modern and space-efficient solution.

Key Specifications:

- **Input Voltage (Vin):** 7V to 35V DC
- **Adjustable Output Voltage (Vout):** 1.23V to 9.43V DC
- **Maximum Output Current (Iout):** 1A



---

2. Circuit Design Explanation

The circuit is divided into four main functional blocks: Input Protection, the Converter Core, the Feedback Network, and the Output Stage.

A. Input Protection: To ensure robustness, the input stage includes a **1.5A slow-blow fuse (F1)** for overcurrent protection and a **1N4004 diode (D2)** for reverse polarity protection. This prevents damage to the circuit from accidental incorrect power supply connections or catastrophic failures.

B. Converter Core: The heart of the circuit is the **LM2596S-ADJ (U1)**, a 150 kHz switching regulator. It operates by taking the input voltage and switching it on and off at a high frequency. This switched voltage is fed to a **100 µH power inductor (L1)** and a **1N5822 Schottky diode (D1)**. The inductor stores energy when the switch is on and releases it when the switch is off. The Schottky diode is crucial for efficiency, providing a low-loss path for the inductor current during the off-cycle.

C. Feedback Network: To achieve a regulated output, a feedback loop is implemented. A voltage divider, formed by a fixed **1.5 kΩ resistor (R1)** and a **10 kΩ potentiometer (R2)**, samples the output voltage. This sampled voltage is fed to the IC's feedback pin. The LM2596 internally compares this to a precise 1.23V reference and adjusts its switching duty cycle continuously to maintain the target output voltage, regardless of changes in input voltage or output load.

D. Output Stage: The output from the inductor is filtered by a **220 µF electrolytic capacitor (C2)** to smooth the DC voltage and a **0.1 µF ceramic capacitor (C3)** to filter high-frequency noise. This ensures a clean and stable DC output. An LED connected to the output via a current-limiting resistor provides a visual indication of an active power rail.

---

3. Key Component Selection

Component selection was driven by reliability, performance, and cost-effectiveness.

- **Regulator (LM2596S-ADJ):** Chosen for its robustness, integrated protection features (thermal shutdown, current limit), and high current capability (3A), providing ample margin for the 1A design requirement.
- **Inductor (Bourns SRP1245A-101M):** A 100 µH shielded power inductor was selected. Its **3.0A saturation current rating** is well above the circuit's peak current (~1.15A), preventing saturation and ensuring stable operation. Its low DCR (DC Resistance) contributes to higher overall efficiency.
- **Catch Diode (1N5822):** A Schottky diode was essential. The 1N5822 was chosen for its fast switching speed and low forward voltage drop (~0.5V), which significantly improves efficiency compared to a standard rectifier.
- **Potentiometer (Bourns 3296W):** A multiturn (25-turn) trimmer potentiometer was selected for the feedback network. This allows for very precise and stable adjustment of the output voltage.
- **Capacitors:** Low-ESR (Equivalent Series Resistance) capacitors were chosen for the input and output to effectively handle the ripple currents associated with switching converters.

---

4. Conclusion

This project successfully implements a fully functional and adjustable DC-DC buck converter. The design meets all specified requirements, incorporating robust protection features and utilizing well-chosen components for reliable performance. The final PCB layout is compact and follows best practices for power supply design, including wide power traces and a solid ground plane for thermal management and signal integrity.