Adjustable Power Supply (1.2V–50V @ 3A)

Adam Alshami

June 5, 2025

Contents

1 Abstract

The goal of this project is to create adjustable power supply (1.2V-50 @ 3A) with a high-efficiency, based on the LM2576HVT-ADJ. The project can convert an input voltage of up to 55 V DC into an adjustable output voltage between 1.23 V and 50 V DC, with a maximum current of 3 A. By leveraging the LM2576HVT-ADJ's built-in features such as fixed-frequency operation at 52 kHz, integrated over temperature and over-current protection, and low standby current (typically 50 A)—a compact and reliable design is achieved with a minimal number of external components.[1] The design is further build by the availability of standard components, such as inductors, capacitors and diods. The result is an efficient power module with excellent line and load regulation, well suited for a wide range of electronics prototyping.

2 Introduction

Adjustable power supply is key component in modern electronic systems. The Buck converters, or sometimes called step-down regulators, play a important role by efficiently reducing higher input voltages to lower levels with minimal energy loss. The LM2576 family and including the LM2576HVT-ADJ, have become a popular choice due to its integrated design that combines efficient switching, and ease of implementation.
Because of its broad output voltage range (1.23–57 V), its capacity to supply up to 3 A, and its integrated features that streamline the overall design, the LM2576HVT-ADJ was chosen as the main component. This device's applicability in systems with changeable supply sources is extended since it can tolerate greater input voltages (with ± 4 % accuracy), unlike the lower-voltage counterparts in the same series. Additionally, its fixed-frequency 52 kHz oscillator removes the need for external frequency tuning and lowers electromagnetic interference. In such projects, striking a balance between cost, scale, and efficiency is a crucial task. The design includes screw terminals for convenient input/output connections and a power LED to indicate input supply. To avoid and ensure a clean power delivery, the circuit features an LC filter (C5 L2) which reduces output ripple voltage caused of the switching regulator, significantly improving signal stability. Moreover, the converter includes a built in current limiting protection to ensure safety against output overload and reliable operation under fault conditions

3 Methodology

This section describes the design and implementation process of the adjustable DC supply, including schematic design, PCB layout, routing, component selection, placement, and verification. The work was carried out with a focus on achieving high efficiency, compact size, and reliable operation under various load conditions.

3.1 Schematic Desgin

The schematic design was centered around the LM2576HVT-ADJ buck converter making sure a robust and efficient voltage regulation. The design was made in KiCad, which is freely available for both Windows and Mac as well as it is an open source.
The buck regulator was implemented according to Texas Instruments datasheet for the LM2576 serie [1], using its fixed 52kHz internal oscillator to drive the switching. The output voltage is regulated via a feedback network consisting of the potentiometer (P1 50k) and resistors R2 4.7k and R3 1.21k, which define the adjustable voltage range 1.23–50V. To further reduce output ripple, an LC filter composed of L2 (12µH) and C5 100µF was placed directly before the output terminals (see Figure 2).

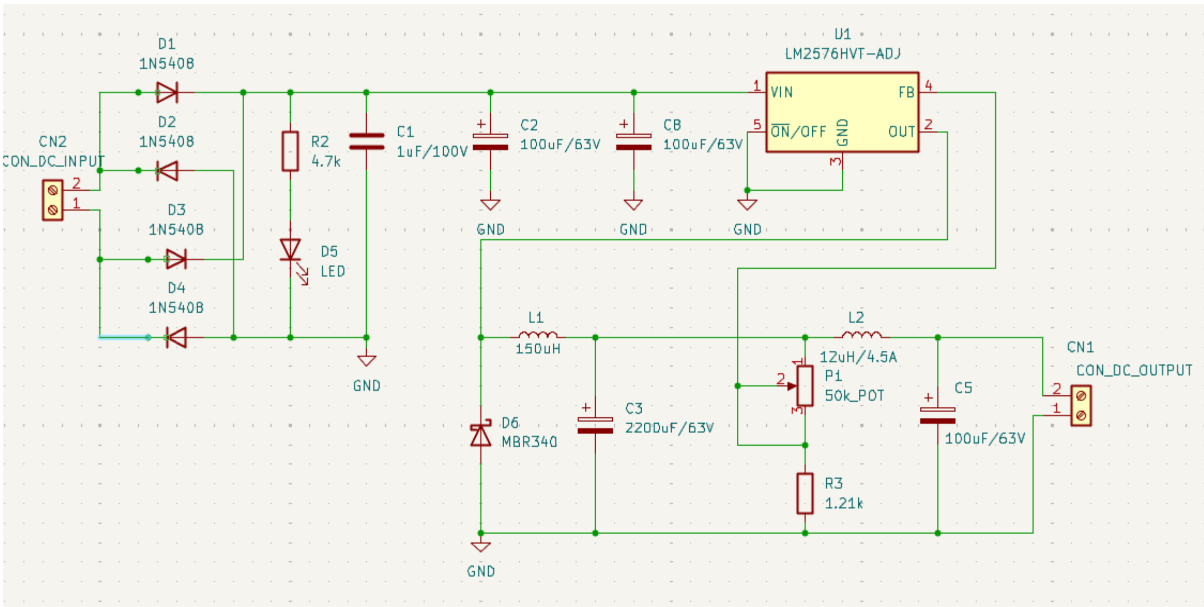


Figure 1: Schematic diagram

3.2 Ripple filter

To attenuate/damp high-frequency noise from the switching action 52kHz and reduce residual ripple, an LC filter consisting of the inductor L2 12µH and the capacitor C2 100µF was included. The cutoff frequency was calculated as

$$f_{\text{cutoff}} = \frac{1}{2\pi\sqrt{12\ \mu\text{H} \cdot 100\ \mu\text{F}}} \approx 4,6\ \text{kHz},$$

which is well below the switching frequency 52kHz, ensuring effective filtering

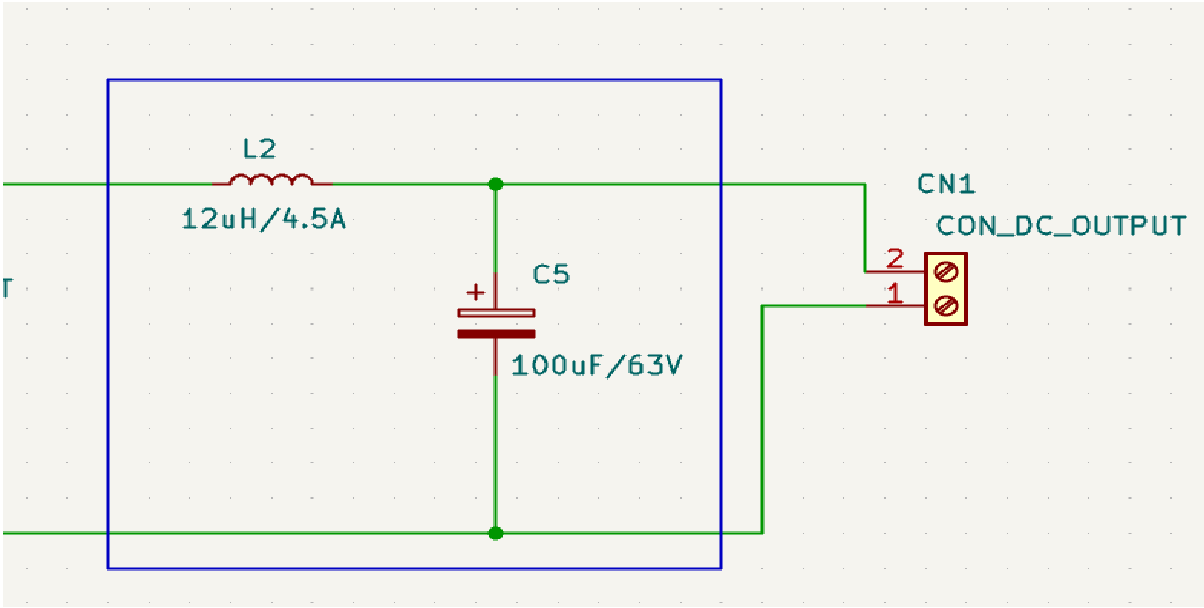


Figure 2: Ripple filter

3.3 Component selection

The LM2576HVT-ADJ was being chosen for this project for it adjustable output voltage range (1.23–50V), low operating voltage, and its ability to supply up to 3A of load current, enabling wide voltage and power handling in a single circuit. The energy-transfer elements consist of two inductors—L1 (150µH) for the main buck conversion and L2 (12µH) for the additional LC output filter—thereby reducing output ripple as specified in the project plan. Moreover, the capacitors were selected as C1: 100µF/100V, C2 and C8: 100µF/63V, and C3: 2200µF/63V to ensure stable operation under varying loads. For freewheeling and input rectification, Schottky diodes 1N5408 and MBR360 were used, providing robust over-current protection of voltage spike induced by the L1 and minimal switching losses. A heatsink will be mounted on the transistor to facilitate heat dissipation.

BOM							
NO	QNTY.	REF	DESC	MANUFACTURER	SUPPLIER	SUPPLIER PART NO	Price
1	1	CN1	2 PIN SCREW TERMENAL 5.08MM PITCH	WIEDMULLER	DIGIKEY	281-2888-ND	8.28
2	1	CN2	2 PIN SCREW TERMENAL 5.08MM PITCH	WIEDMULLER	DIGIKEY	281-2888-ND	8.28
3	1	C1	1uF/100V SMD SIZE 1206	MURATA/YAGEO	DIGIKEY	1206B105K101CT	1,62
4	3	C2,C5,C8	100uF/63V SMD	NICHICON	DIGIKEY	493-6185-1-ND	3x7,58=23
5	2	C3	2200uF/63V THT	NICHICON	DIGIKEY	493-1360-ND	27.9
7	4	D1,D2,D3,D4	1N5408	ON SEMI	DIGIKEY	1N5408G	3,94
8	1	D5	RED LED SMD	OSRAM	DIGIKEY	HSMC-C170	4.95
9	1	D6	MBRS360	ON SEMI	DIGIKEY	MBRS360T3GOSCT-ND	3.34
10	1	L1	150uH/5A 30MM DIA	BOURNS	DIGIKEY	652-2200LL-151-H-RC	41.65
11	1	L2	20uH OR 12uH/4.5A 12MMX12MM	BOURNS	DIGIKEY	SRR1260-120MTR-ND	10.21
12	1	P1	50K POT 16MM	BOURNS	DIGIKEY	PDB181-K415K-503B-ND	13.14
13	1	R2	4.7K 5% SMD 2512 OR 1210	MURATA/YAGEO			1.21
14	1	R3	1.21K 1% SMD SIZE0805	MURATA/YAGEO			1.21
15	1	U1	LM2576HVT-ADJ TO220	TEXAS	DIGIKEY	LM2576HVT-ADJ/NOPB-ND	67.33
							Tot 211 sek

Figure 3: BOM List

3.4 PCB Layout

The PCB was designed in KiCad with dimensions of 85×60mm using this free, open-source circuit and PCB layout software. Input and output connections are implemented with screw terminals (CN1 and CN2) for easy hookup during both testing and regular use. An on-board indicator LED (D5) lights whenever an input voltage is present, providing a clear visual confirmation that the unit is powered. Special attention was paid to PCB trace widths to accommodate the maximum load current of 3A. According to standard PCB trace-width guidelines, a 3A current requires at least a 50mil (1.27mm) wide trace [4], to ensure a sufficient safety margin and reliable current delivery, all power traces were routed at 80mils (2mm) width.

The PCB layout follows Texas Instruments' LM2576 datasheet recommendations. [1] The top and bottom planes were grounded in order to minimize traces that are connected to ground so much as possible, to ensure stable operation and low noise.
Four 2.2mm mounting holes are integrated into the board to allow secure attachment inside an enclosure if needed. Several SMD pads have been increased slightly beyond their datasheet dimensions to facilitate manual soldering and improve solder fillets—especially useful in this case when stencils are not used.The stencils can increase fabrication costs and it is not necessary for the prototype. The silkscreen layer clearly labels all components to aid assembly and troubleshooting. In addition, both the input and output terminals are distinctly marked—"DC Input" at CN2 and "GND" and "V+" at the output—as shown in the figure below.

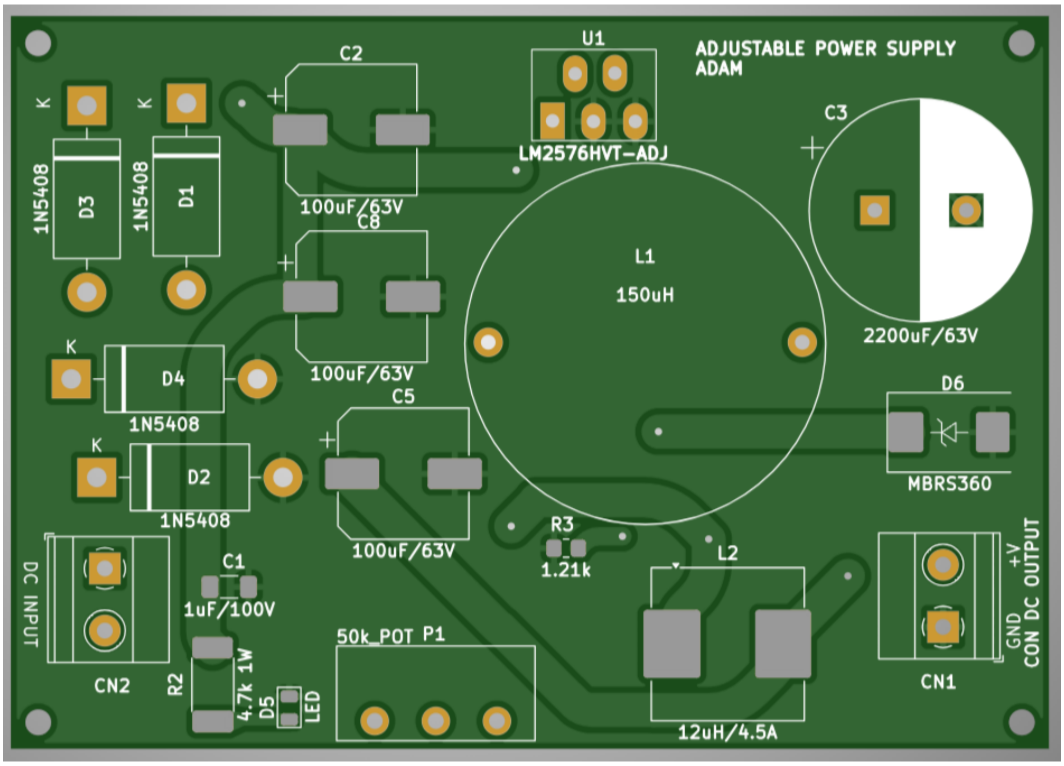


Figure 4: PCB view front

3.5 Simulation in MATLAB

Before committing to PCB fabrication and ordering components, the entire buck-converter design was first validated through a time-domain simulation. The circuit was implemented in a MATLAB script that models the LM2576-based converter with an outer PI controller and an added LC ripple-filter on the output, using the same component values as in Figure 1 and $V_{\text{in}} = 55$ V DC.

- **PI Controller:**

$$K_p = 5, \quad K_i = 500$$

  generates a control voltage compared against a sawtooth carrier to produce the PWM duty cycle.

- **Switch-Node Dynamics:** The switch node toggles between V_{in} and $-V_{\text{diode}}$ based on the duty signal, and the inductor-capacitor state equations are integrated using Euler's method.

- **Results Logging:** Capacitor voltages, final output voltages are recorded over a 50 ms simulation windows.

Plots generated at the end of the simulation confirmed: This simulation step ensured correct regulation, adequate voltage delivery, and ripple control before any hardware was manufactured.

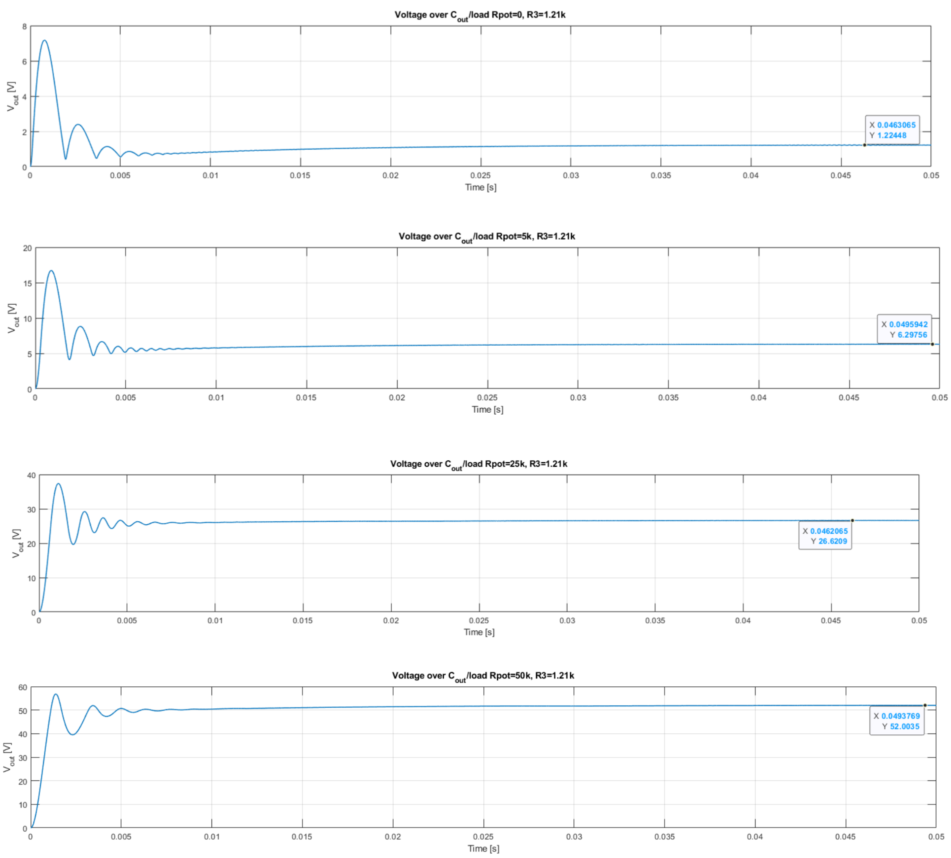


Figure 5: Results of simulations.

4 Results

When the PCB was received from manufacturing, a thorough inspection was carried out to verify component placement and solder pad alignment. All components were found to be correct, except for one issue: the power resistor R2 (4.7k, 1W), intended to protect the indicator LED, was missing. The originally planned surface-mount resistor had a larger footprint than standard SMD sizes and could not be mounted properly. To resolve this, a standard through-hole resistor was used as a replacement. Its leads were trimmed and shaped to fit the SMD pads, allowing it to be mounted on the PCB without functional compromise. This workaround ensured that the LED indicator could operate safely and reliably, despite the deviation from the original SMD specification.

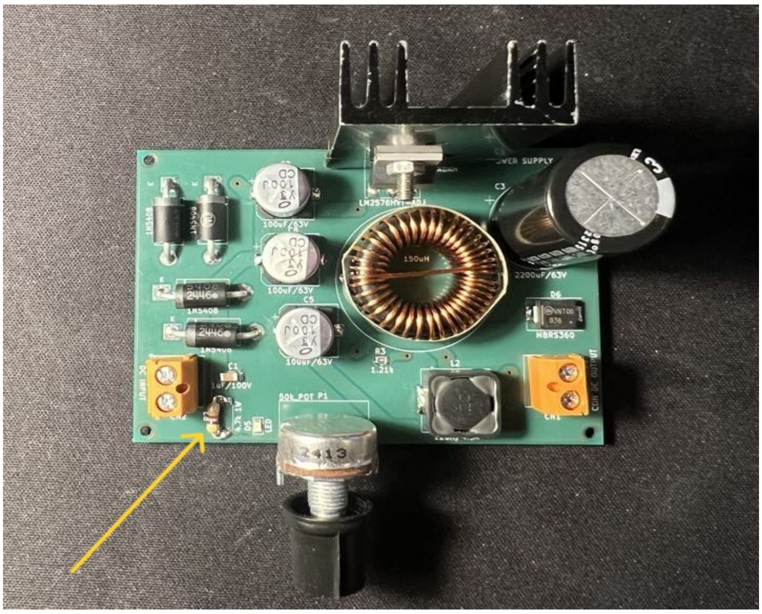


Figure 6: The project front view

An input full-wave bridge rectifier was mounted at the PCB's power input to protect the converter and enable either AC or DC input. The converter was first tested with 12 AC and 12 DC to verify that everything works properly, futhermore was it tested both with a 40V DC source and with a 40V AC source. In both cases, the converter started up correctly, regulated the output, and sustained stable operation up to a 3A load. This confirms that the bridge arrangement reliably protects the circuitry while providing full compatibility with AC and DC supplies.
With a 40V AC input, the output voltage was measured under varying load conditions from 0.1A to 3A. Table 1 and fiures 8 summarizes the load regulation data: the output remains within 0.55prcent its no-load value, exhibiting only a slight voltage drop as current increases—an outcome consistent with the converter's inherent losses.
Throughout all tests, no thermal run-away or voltage oscillations were observed. Visual inspections and oscilloscope measurements demonstrated that the LC filter effectively getting a clean DC output. Furthermore, the minum output voltage when turning the potentiometer to its lowest resistansce I get a voltage of 1.237 V and ann aproximatly 38 V when te input is 40 V which indicate a bit drop on the output votage as espexcted beacuase of the voltage drop due to the diods and LM25 Figure 8 illustrate the regulator's performance at selected load currents (0.5A, 1A, 2A, and 3A).

4.1 Measurements

Output Voltage Calculation

The LM2576HVT-ADJ contains an internal reference voltage, V_{ref}, of exactly 1.23V according to the Texas Instruments datasheet [1]. This reference voltage is used internally by the error amplifier to compare against the feedback voltage on the FB pin. The controller adjusts its switch until the FB pin voltage equals $V_{\mathrm{ref}} = 1.23$V.

When the potentiometer is set to zero ohms, the feedback network forces the output voltage also to fall to 1.23V. The general formula for the output voltage is:

$$V_{\mathrm{FB}} = V_{\mathrm{ref}} = \frac{R_3}{R_{\mathrm{Pot}} + R_3}\, V_{\mathrm{out}} \overset{!}{=}$$

Solving for V_{out} gives

$$V_{\mathrm{out}} = V_{\mathrm{ref}}\, \frac{R_{\mathrm{Pot}} + R_3}{R_3} = V_{\mathrm{ref}}\left(1 + \frac{R_{\mathrm{Pot}}}{R_3}\right).$$

The feedback network uses:

$$R_{\mathrm{pot}} \in [0,\ 50\ \mathrm{k}\Omega], \quad R_3 = 1.2\ \mathrm{k}\Omega, \quad V_{\mathrm{ref}} = 1.23\ \mathrm{V}.$$

The general formula is

$$V_{\mathrm{out}} = V_{\mathrm{ref}}\left(1 + \frac{R_{\mathrm{pot}}}{R_3}\right) = 1.23\left(1 + \frac{R_{\mathrm{pot}}}{1.2\,\mathrm{k}\Omega}\right).$$

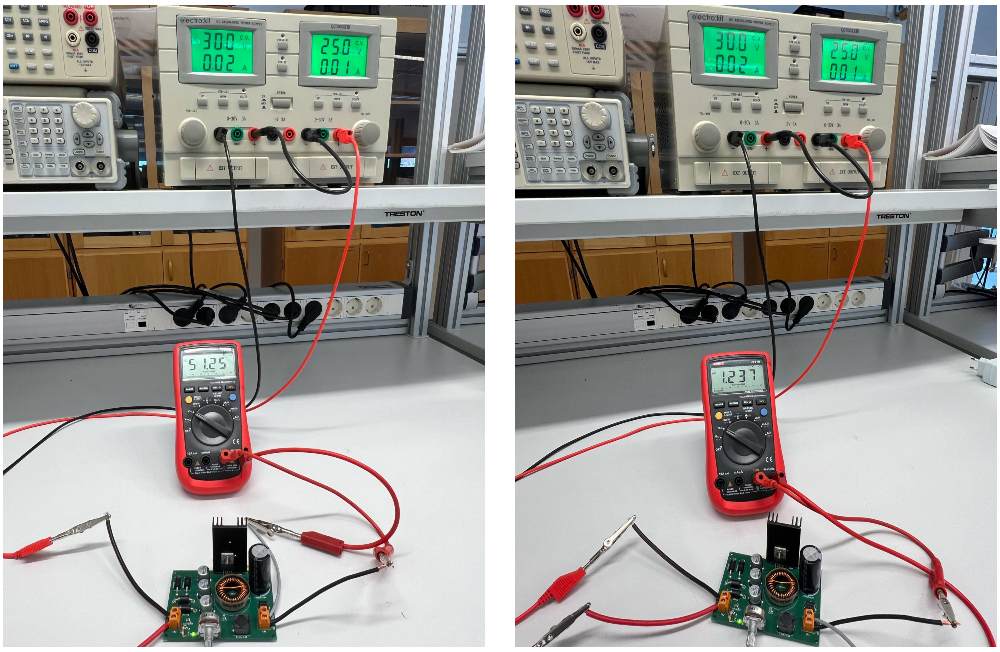
Case 1: $R_{\mathrm{pot}} = 0$

$$V_{\mathrm{out,min}} = 1.23\left(1 + \frac{0}{1.2\,\mathrm{k}\Omega}\right) = 1.23.$$

Case 2: $R_{\mathrm{pot}} = 50\ \mathrm{k}\Omega$

$$V_{\mathrm{out,max}} = 1.23\left(1 + \frac{50\,\mathrm{k}\Omega}{1.2\,\mathrm{k}\Omega}\right) = 1.23 \times \left(1 + 41.667\right) \approx 52.0.$$

The theoretical predictions agree closely with the experimental results: when the PCB was powered from a 55VDC source, both the minimum output of 1.237V and the maximum output of approximately 51.25V matched the calculated values.



(a) Maximum Output at 55V Input (b) Minimum Output at 55V Input

Figure 7: Measured output voltages under a 55VDC input.

Output Voltage with load

Furthermore, testing and verifying the Buck converter under various load conditions is a crucial aspect of evaluating its performance. The converter was tested with different load currents and up to its maximum rated load of 3A. The figure and table below illustrate the results. The output voltage remained relatively stable throughout the load range, with a slight voltage drop observed as the load increased. This behavior is consistent with expectations and indicates that the converter performs well under load, maintaining regulation without significant voltage drop.

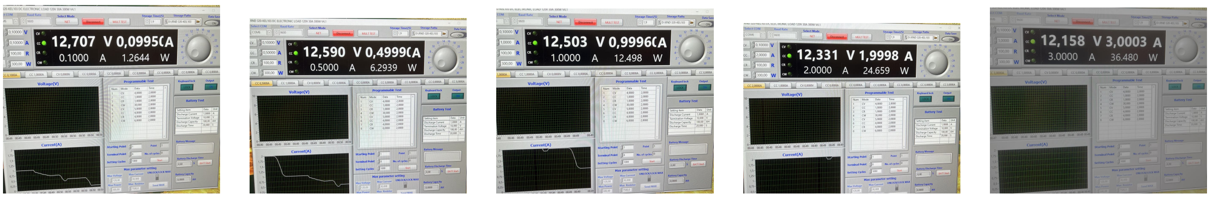


Figure 8: Results of load regulation.

The output voltage remained relatively stable throughout the load range, with a slight voltage drop observed as the load increased. This behavior is consistent with expectations and indicates that the converter performs well under load, maintaining regulation without significant voltage drop.

Table 1: Load Regulation at 40V Input

Load Current [A]	Output Voltage [V]	Power Output [W]
0.1	12.707	1.264
0.5	12.590	6.294
1.0	12.503	12.498
2.0	12.331	24.659
3.0	12.158	36.480

4.2 Thermal Performance

The circuit board was tested with a thermal camera to verify the heat dissipation performance under maximum load. The overall results, the thermal performance is satisfactory. The 1N5408 diodes (D1 and D3) at the input ware measured at around 70 °C, and the inductor L2 at the output were measured at approximately 85 °C, which both are well below its maximum ratings: 125 °C for L2 [5] and 175 °C for the 1N5408 [6]. And accuring to the 1N5408 datasheet, the diod can operate safely in ambient temperature up 75 °C withtout any risk of thermal runaway [6]. Since the measurments are below the below limits can the temperature dissipation be considered to be exptable and operating within safe thermal limits.

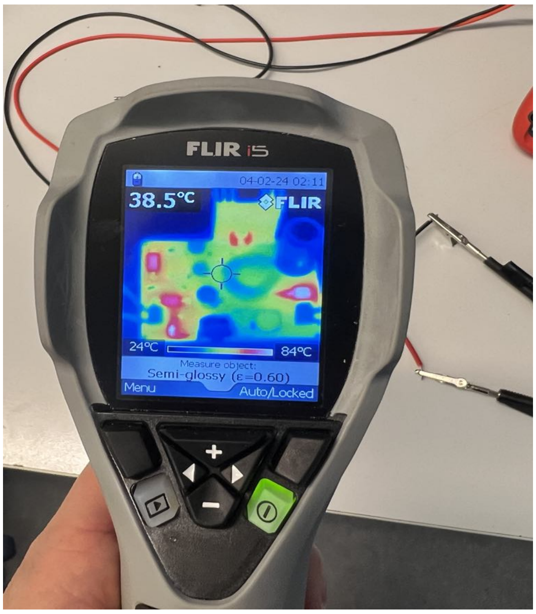
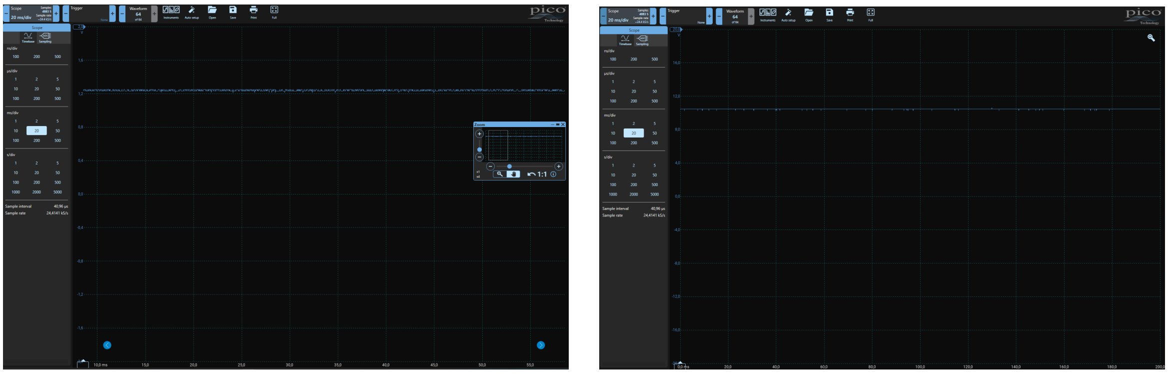


Figure 9: Thermalcamera with maximum load

4.3 Ripple & Noise

When it comes to the noise, there is a picScope 2000 series which was used to measure the ripple noise at the output connector. It is clearly that the noise is bit slight when the output voltage is low compared to higher output voltage, it results very low noise level when the output voltages increased as the figures a and b below. Thanks to the ripple filter att the output which filtering noise and give a nice DC outout level.

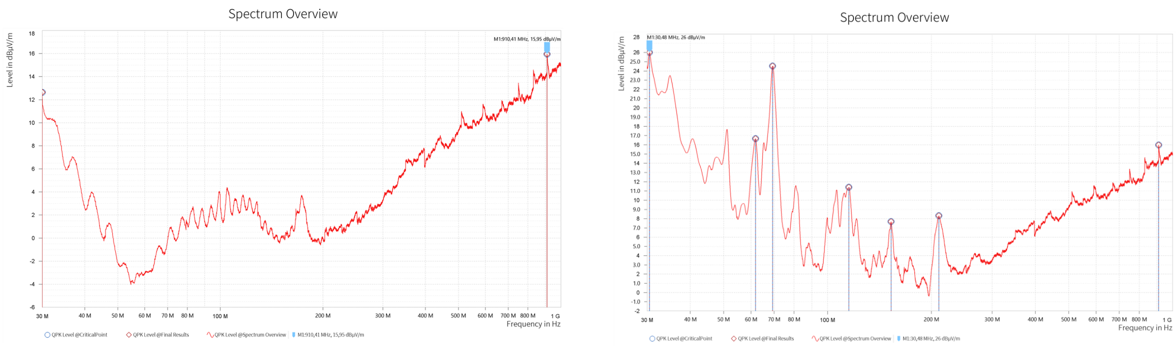


(a) Noise with no load at output 1.23V (b) Noise with no load at outout 11V

Figure 10: Measured output voltages wiht a input of around 12V AC.

4.4 EMC measurment

The PCB was tested for Electromagnetic Compatibility (EMC) in an EMC Maxwell chamber at Halmstad University. The PCB was placed at a distance of 3 m from the antenna at a height of 1.5 m. The antenna automatically switches between horizontal and vertical polarization. Radiated emissions were measured over the frequency range 30 MHz to 1 GHz using a Peak detector (QPK) with a measurement bandwidth (BW) of 120 kHz.



(a) Unit on without load (b) Unit on with load 3A

Figure 11: Spectrum of radiated emissions without load and with 5V/3A load.

From the results, it is not possible to assign a specific EMC class (e.g Class A or Class B) because no reference limit lines or target standard were used during testing. Each class has its own set of frequency-dependent emission limits, defined under a particular standard. Since the setup did not include those limit lines or curves, and the PCB prototype was not targeting any specific standard or application (e.g., commercial or industrial),there was no need to reference any class specification. Moreover, the switching frequency of the transistor was well below the spectrum's band (30 MHz–1 GHz). The main purpose of the EMC measurements was primarily to become more familiar with EMC testning and measuring.

5 Conclusion

The project set out to build a compact, effecient adjustable power supply as the project plan descriped by using LM2576HVT-ADJ buck regulator, capable of delivering an output range from 1.23 V up to 50 V at max current up to 3A,through component selection, PCB layout, LC ripple filter and in practice, the converter achived stable voltage regulation and performed remarkably well.

5.1 Voltage regulation and load performance

The buck converter, with an input voltage up to 55 V, produced an output range from 1.23 V to approximately 51 V. With a 40 V input, load regulation was tested from 0.1 A up to 3 A, and the output voltage dropped slightly under load, which is expected in such cases. Overall, the regulation results were satisfactory. However, during testing, I noticed a small delay in the output response when adjusting the potentiometer. In other words, when the potentiometer was turned to a new value, the output took about one second to respond. A possible reason for this delay is that capacitor C3 does not charge and discharge quickly enough. While this behavior is acceptable in many situations/cases, it can be improved by placing a small resistor in parallel with L2 and C3 to ensure that the capacitor charges and discharges quickly.

5.2 Thermal test:

All components stayed within a aceptable tempeature range at a continous 3A max load, the parts that heat up the most were diode at the input which did not climed above 70 °C and the power inductor that reached around 85 °C during the thermal test. The results were compared to the components datasheet, and according to it, these temperaturen meausrments are still below the maximum allowed temperature values- bothe in terms of operating temperature and ambient limits. When implimaning or designing an enclosure like a box for the PCB, it is important to consider thermal mangenment. Thermal vias and effective cooling methods should be carefully implemented to ensure proper heat dissipation to prevent any overheating during usage. Bad heat dissipation can lead to overheating which in turn may shorten the lifespan of components and decrease the PCB's optimal functioning.

5.3 The costs

In the BOM list (figure 3), the components prices for a single prototype from the distributor DigiKey come to roughly 211 SEK, which is acceptable for a single PCB. However, there are several ways to reduce the few percent savings depending on the amount of units, e.g. ordering a big amount of components and trying to change standard kondensatorer like C1, C2, C3, C5, C8 and L1 and L2 can be changed. It is often possible to choose another major brand (e.g. Coilcraft or TDK) that offers equivalent specifications at a lower price. The other components such as LM2576HVT-ADJ is specified and is quiet hard to replaced. On the output side, adding the 12 µH/100 µF LC filter right at the switch node paid off. Across the entire load range,as it showed in (figures 10), which is good enough for most lab equipment or embedded electronics. At very low output voltages (around 1.23 V), the ripple was a slight noticesable at low voltage output and almost nonexistent at higher voltages.

5.4 Simulation

The simulations in MATLAB are useful before ordering and manufacturing the prototype, as they help ensure the unit will function correctly. In this case, detailed simulations were not strictly necessary because the PCB layout was based directly on the reliable Texas Instruments datasheet for the LM2576HVT-ADJ [1]. However, whenever new components (such as diode bridge diodes, a power-on LED, or ripple-filtering elements) are introduced into the circuit, it is advisable to run simulations to verify proper operation. My MATLAB simulations (see Figure 5) showed the expected output voltages for various potentiometer settings. There was a small overshoot at startup, which damped out within approximately 5 ms. Although I am not certain that such a transient is ideal, it appears acceptable for my application. After soldering and testing the prototype in practice, however, no such transient was observed.

Overall, the design meets the expected results outlined in the project plan, as well as the accuracy requirements for creating a functional PCB prototype. During the design process, I paid close attention to key factors such as thermal management, trace widths, and overall functionality.
Moreover, there are several important considerations when designing a buck converter PCB. For example, it is beneficial to minimize the number of vias in order to reduce manufacturing cost and improve current flow. It is also recomanded to avoid any sharp trace corners—in this case, I avoided any 45° angle traces when designing the PCB, because it helps maintain signal integrity and the flow ofcurrent. Traces that carry high current should be kept as short and straight as possible, which lowers resistance and improves efficiency.

6 Discussion

Firstly, a resistance could be placed between C3 and the potentiometer in the circuit because there is a small delay in the millisecond range when reducing the output voltage from a higher level to a lower one, which can improve the discharge of C3 when I reduce the output. It appears when no load is connected to the prototype.
Furthermore, there is potential to reduce the size of the PCB by a few centimeters in both dimensions. The layout can be optimized by grouping components more closely and making better routing, as well as paying attention to thermal heat dissipation. In my opinion, a smaller PCB can save both material costs and allow for a more compact end product.
Another area that can be improved is the EMC part. By introducing reference lines or limits (for example, according to CISPR classes) in the measurement setup, it is difficult to clearly indicate which EMC class the prototype should meet or correspond to, due to the absence of such a reference limit. This is important because any improvement may

be needed to meet either industrial or commercial requirements.
When it comes to simulations, I have simulated my design in MATLAB, which is not as good as OrCAD. I used a PI regulator in MATLAB because there were no simulation models for the LM2576HVT-ADJ in OrCAD. Next time, instead of switching from OrCAD to MATLAB for simulation, I could search for a replacement component with the same specifications as the LM2576HVT-ADJ for which OrCAD has ready-made simulation files. This way, the simulation results will be more representative, and I can have more confidence in the transient analysis and in the final PCB layout. The results from the MATLAB simulation were acceptable, apart from the initial transient that occurs in a very short time. Since the schematic is based on data from the Texas Instruments datasheet, this issue could be overlooked for the PCB design part, but for future versions it would be desirable to have a complete OrCAD simulation.

What I have learned from the project beside designing the PCB layout, preparing footprints, the silkscreen, checking component datasheets before use, and managing the budget and costs, is that I gained a solid understanding of how a buck converter works and selecting component values such as inductors, capacitors, and feedback components to achieve a stable output. As well, the project has given me practical skills in measurement steps and simulation, insight into how to document and follow a structured reporting process, and the ability to identify and prioritize improvement actions, especially under a set schedule. These lessons will be valuable in my future engineering projects, whether they involve similar power circuits or other types of electronics.

7 References

[1] https://www.ti.com/lit/ds/symlink/lm2576.pdf
[2] https://en.wikipedia.org/wiki/Buck_converter
[3] https://www.youtube.com/watch?v=ywFpNdvSC7A&ab_channel=MyVanitar
[4] https://resources.altium.com/p/pcb-trace-width-vs-current-table-high-voltage-desig
[5] https://www.bourns.com/docs/Product-Datasheets/SRR1260.pdf
[6] https://mm.digikey.com/Volume0/opasdata/d220001/medias/docus/1043/1N5401-08_Rev_2.5.pdf