

Procedure of designing and producing a boost converter PCB

Felix Sjöqvist and Olle Olofsson

MDH — School of innovation, Design and Technology

Västerås, Sweden

Email: fst17001@student.mdh.se, oon17003@student.mdh.se

I. ABSTRACT

Boost converters is an important tool for many applications including many small battery powered devices and hybrid electric vehicles. And for the engineer its function is of great importance. But so is it to be able to make one. This report explores the design and production of making a boost converter on a printed circuit board with the purpose to supply the reader with information about the experience that was gained doing this project. This includes what hardware and software was used, the choice of components, the manufacturing process and how the circuit finally got tested. The goal of the project was to successfully produce a working boost converter which actual measured voltage output is in pair with the simulated value. This is found to not be the case, possibly because of an unwanted short in the circuit.

II. INTRODUCTION

Boost converters are widely used in numerous electrical applications throughout the world, all the way from small battery powered applications to hybrid electric vehicles. Its frequent use makes it impossible to ignore and very important to understand. But understanding the boost converter is worthless if you cannot make one yourself.

The subject of this report is the procedure of designing and producing a printed circuit board (PCB) step up converter with the constraint of using a 1mH inductor. This report will explore the procedures required for developing such a PCB. The goal of the project was to successfully design and produce a step up converter using the tools at hand, but also to get the understanding of some of the difficulties one can encounter in the process.

Before this project we knew how a boost converter worked in theory and had a small amount of experience of how it worked in practice. We knew how to design and simulate circuits using Multisim but had no experience of producing PCB's.

When the circuits' design had been made, components chosen, and simulations had been performed, we knew what to expect from the final outcome. The simulated output voltage of the circuit at 10000Hz and a duty cycle of 0.9 was 15V. This value will be compared to the actual output voltage of the circuit to evaluate if the project was a success or not.

III. CONTROLLING OUTPUT VOLTAGE

The output voltage is controlled using the duty cycle of the input signal. The duty cycle is the fraction of the period in which the signal is active, and therefore takes on values between 0 and 1. A low duty cycle, say 0.25 will produce a low boost effect. This is because, the lower the time the mosfet is on i.e. letting current flow through it, the lower the time the inductor's magnetic field is charging up which boosts the output voltage during the signal off time. If the duty cycle instead were to be 0.75, the inductor would have a larger magnetic field to boost the output voltage. The theoretical output voltage can be expressed as the input voltage and the duty cycle d :

$$V_{out} = V_{in}/(1 - d) \quad (1)$$

The output voltage is also dependant on the frequency. Since there are inductive and capacitive components in the circuit, the total impedance is changing with the frequency. It is hard to know the actual total impedance of the circuit, and therefore finding an optimal frequency, but a good estimation of optimal frequency can be done by calculating the resonance frequency of the inductor and capacitor. This can be done using the following formula:

$$f = 1/(2\pi\sqrt{LC}) \quad (2)$$

In this case, with the components of this project, the resonance frequency is 5032Hz, which would make that a good starting point if one tried to maximize the voltage output.

IV. HARDWARE

A note on replicability: Although the hardware-related equipment used in this paper could be considered to be of a level of sophistication unreachable or undesirable, a substantially more rudimentary setup would suffice. For testing purposes, a simple PWM-signal generating device such as a function generator, or even simpler, an Arduino UNO [1] could be used. Since the circuit is designed to run on 5V DC, even an ad hoc power supply comprised of a cut USB-cord plugged into a wall adapter for phone chargers, or an unused USB-port, could make due. As for measuring, any old multimeter would do.

NI MyDAQ is a simplistic and small all-in-one solution providing all the necessary power and measuring capabilities used in this paper. It provides a steady 5V output for power, a

flexible, software-controlled signal generator and a multimeter capable of measuring 60V[2]. Far more than our predicted output, and far beyond what our capacitor would be physically able to produce.

Voltera Printer was used to realize the circuit for testing purposes[3]. It is a desktop-sized device designed for rapid PCB-prototyping, allowing for a short idea-to-test pipeline, similar to what FFF 3D-printing technology has come to provide for mechanical applications design.

V. SOFTWARE

Accompanying the earlier note on replicability, a similar description of suitable software substitutes will be provided, since the software choices relies heavily on the hardware used. The only crucial software, if a setup similar to what was described above was to be used, would be the IDE used to program the Arduino board. One option is the official Arduino IDE, available as a web-based editor and as a desktop application[5].

Multisim[4], by National Instruments, was used to design and simulate the circuit, a software based environment allows for fast and easy experimentation, which is convenient in electronics design due to its trial-and-error nature.

Ultiboard is another peice of software by National Instruments, with out-of-the-box integration with Multisim. It was used to design the actual PCB using the already-designed circuit imported from Multisim.

Voltera for Windows 64-bit was used to interact with the Voltera printer.

VI. CHOICE OF COMPONENTS

The choice of componets was experimentally produced though the starting point were an erlier lab where a working boost converter was already designed. This design was of course not directly applicable to our project since this was not a surface mounted design. Different components were tested and simulated untill a desirible result was reached, see Tab I. Simulations were done using a 50 Ω resistor in series before the inductor to simulate a real inductor, which has a resistance. While small components often are desirible, the size can compromie circuit stability, espesially regarding capacitors. Therefore a smaller capacitor was not necessary.

TABLE I
CHOICE OF COMPONENTS

Component	Name	Value	Footprint (%)
Capacitor	-	1uF	CAPC2012X145N
Inductor	-	1mH	INDC4520X140N
MOSFET	2N7002K	-	SOT95P230X110-3N
Diode	D1N4148	-	SOD2513X110AN
Resistor	5000	4700 Ω	RESC1608X63N
Pin strip	Pin strip	2x7	Custom

VII. MANUFACTURE PROCESS

This section will describe the full process of developing the step up converter PCB. That is simulation, PCB layout, and printing.

A. Design and simulation

Firstly, simulations were made to experimentally come up with a theoretical solution to the problem. In this stage, total freedom was at hand which made the ability of finding errors and differentiate good solutions from bad ones simpler. The simulations were made in Multisim using the build in function generator and multimeter to measure the outcome. When the desired function had been simulated and verifired, the netlist, which is a description of the connectivity of the circuit, was carefully verified to maintain the desired function going into Ultiboard.

B. PCB layout

The Multisim file was transfered to Ultiboard where the last design choices were to be made. In Ultiboard, two crucial files were constructed.

The first one "*Copper Top*", which contains the information of where the printer shall print the silver traces, connecting all the components. Thanks to the netlist from Multisim, Ultiboard automatically connects all components making the process significantly easier. The components were placed as compact as possible, with simplicity in mind, see Fig.1. Next trace width and trace clearence was chosen to handle the voltage, current and frequency the circuit was designed for.

The next file "*Solder Mask Top*", contains information about where the printer shall print the soldering masks that the physical components later can be placed and soldered on to. This information is automatically generated from the footprints of the components, being chosen in the Multisim file.

C. Printing

After connecting the Voltera printer to the computer, the gerber file "*Copper Top*" form Ultiboard was exported to the Voltera software to begin the printing process. The conductive ink "*LaughingBear*" was used as the conductive trace material. The printer flow was calibrated to ensure good quality traces and pads with neither too much flow, resulting in possible shorts, nor too less, resulting in possible broken traces. The traces were printed and inspected to watch for possible faults. The board was then baked for about 30 minutes, using the ink specific, Voltera baking program, to harden the printed silver ink.

After the baking, the next gerber file "*Solder Mask Top*" was loaded to begin the second printing phase. Here, the solder paste "*FuriousAnt*" was used for the solder masks. The solder masks were printed and inspected to watch for possible shorts.

With solder paste on the pads, the components were ready to be placed on the board. A Small amount of extra solder paste was manually put on the pads of the pin strip footprint to strengthen the otherwise delicate structure. All the components expt the pin strip were carefully placed on the board using the component placing machine. The pin strip was manually placed on the board. Then the board was reflow soldered using the Voltera reflow program, this again took about 30 minutes.

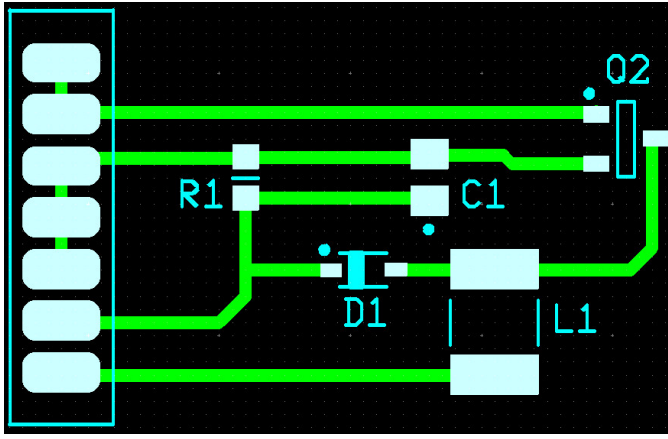


Fig. 1. Picture of the layout of the components on the PCB. R1-resistor, D1-diode, C1-capacitor, L1-inductor, Q2-mosfet.

VIII. METHOD

A. Problem formulation

The PCB was tested using the National Instruments myDAQ, by powering the PCB with the myDAQ's constant 5V output and then imposing a square wave on the input pin with following characteristics:

- Constant 5V amplitude.
- Constant 2.5V positive offset.
- Variable frequency $100\text{Hz} - 10\text{kHz}$
- Variable duty-cycle $10\% - 90\%$

The voltage across GND and Output was measured whilst modifying the variables above, see Fig.2 for more details.

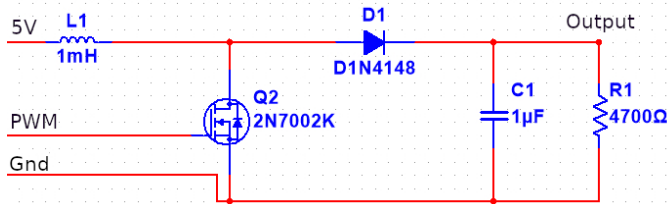


Fig. 2. Multisim diagram of the circuit used in this report

IX. RESULTS

The technique outlined above generated the results in II.

TABLE II
STEP-UP OUTPUT VOLTAGE WITH VARYING FREQUENCY AND DUTY CYCLE

V_{out}	Freq. (Hz)	Duty cyc. (%)
4.87	100	10
4.88	100	50
4.87	100	90
4.87	5000	10
4.86	5000	50
4.85	5000	90
4.87	10 000	10
4.88	10 000	50
4.89	10 000	90

X. CONCLUSION

As evidenced by the results above, our method did not produce the predicted results of 15V at 10000Hz and 90% duty-cycle.

XI. DISCUSSION

Since the method outlined in this paper did not produce the predicted results and our hypothesis hence remains unproven, a deeper discussion of the apparent failure is of high interest. A correlation between $V_{in} = 5\text{V}$ and $V_{out} \approx 5\text{V}$ was quickly made, leading to the assumption that some sort of short between V_{in} and V_{out} had occurred, the investigation ensued. The thick layer of (non-conducting!) hot-glue, earlier applied as a mechanical safety procedure after repeated pin header related failures, was carefully removed using a flat head screwdriver and rigorous amounts of determination. On closer inspection of the PCB, a short between V_{in} and V_{out} was observed, see Fig.3. The reason as to how that short circuit came to be remains unknown. One possible explanation would be that too much solder paste was applied, and the pressure applied during assembly was great enough to smear the paste too far outside the pads, or that it occurred during the reflow-phase. A solution to that issue could be to simply use less solder paste, however that raises another issue specific to the Voltera printer; the reliability of the solder paste extrusion suffers greatly on thinner layers, leading to fatal levels of inconsistencies. A better solution would be to extrude the solder paste only on the center of the footprint, instead of printing a solid layer on the whole footprint area. This approach would, as of the current state of the Voltera software, require a custom gerber file with smaller pads used exclusively for the solder paste. A hassle indeed, but maybe necessary if reliability and repeatability is of any concern.



Fig. 3. Picture of a short on the pin header. The uppermost pad is V_{in} and the one below is V_{out}

REFERENCES

- [1] "abhiV4", *Arduino Buck-Boost Converter*. Instructables.com: <https://www.instructables.com/id/Arduino-Buck-Boost-Converter/>, 2019-03-28.
- [2] National Instruments, *NI myDAQ Specifications - National Instruments*. ni.com: www.ni.com/pdf/manuals/373061f.pdf
- [3] V-One, Voltera.io: <https://www.voltera.io/product/technology/>
- [4] Multisim, ni.com: <http://www.ni.com/en-us/shop/electronic-test-instrumentation/application-software-for-electronic-test-and-instrumentation-category/what-is-multisim.html>

[5] Arduino IDE, arduino.cc: <https://www.arduino.cc/en/Main/Software>