Student research project

Educational DC-DC Buck Converter for Student Lab Purposes

DC-DC converters are omnipresent in modern electronic systems such as renewable energy systems, electric vehicles, and portable electronics. Consequently, DC-DC converters are also an essential topic of any beginner's course on power electronics.

The objective of this student research project is to design, prototype and test an educational buck converter for future student laboratory sessions. Based on the output of this thesis work (documentation, design guidelines), future undergraduate students should assemble and test their own buck converter prototype and gain hands-on experience (soldering, testing, debugging). The focus of this work is therefore not pushing towards high integration levels or performance optimization, but fully on educational aspects and practical usability.

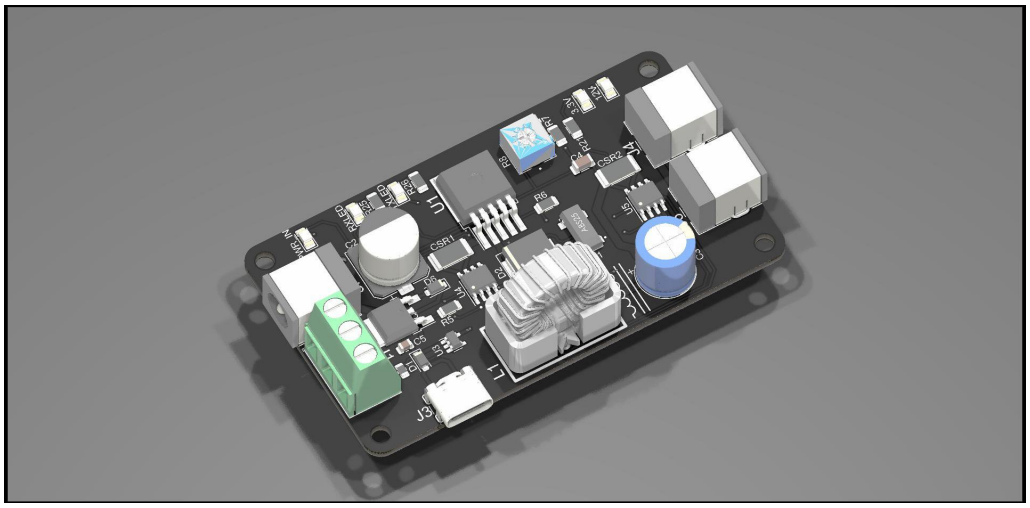

Figure 1: Example buck converter PCB (source: hackaday.io)

The technical specifications of the educational buck converter are as follows:

- Input voltage (nominal): 12 V
- Output voltage range: 0 V to 12 V
- Duty cycle: adjustable via potentiometer (open-loop voltage control)
- Output current (nominal): 1 A
- Switching frequency: 5 kHz to 50 kHz (adjustable via potentiometer)
- Test points for scope probse access (input / output voltage, switching pattern, output current)
- No complex ICs (discrete components only)
- Discrete inductor mount (for self-wired inductors)
- Safety: status LED, fuse and silkscreen labeling
- Connectors: banana plugs

Necessary requirements:

- Finished course work on power electronic fundamentals
- First practical experience with PCB design and soldering

WP 1: Component selection and schematic **[2 weeks]**
The project thesis starts with calculating suitable inductor, capacitor, and semiconductor ratings and choosing market available components. Moreover, the inductor should be self-winded, i.e., a calculation guideline for the core dimensions and winding scheme must be provided. This leads towards the creation of a detailed schematic diagram in KiCad.

WP 2: PCB layout **[2 weeks]**
Based on the schematic from WP1, a PCB layout will be created in KiCad. The layout must consider practical usability aspects, such as test points for oscilloscope probes, discrete inductor mount, and safety features (fuse, silkscreen labeling). After the layout is finished, the PCB and the discrete components to populate it will be ordered from the chair.

WP 3: Hardware assembly and testing **[2 weeks]**
The goal of this WP is to assemble the hardware components of the buck converter and test its functionality. This includes soldering the components onto the PCB, connecting the necessary power supplies, and setting up the test environment. Various tests will be conducted to verify the compliance with specifications. Insights into possible improvements will lead to revisions of the design and layout.

WP 4: Documentation **[1 weeks]**
All above steps should be decently documented in a technical report which includes the design process, the chosen components, the PCB layout, the assembly process, and the testing results. This report will serve as a guide for future students working on similar projects.

Gantt chart

The planned timetable is shown in the manual Gantt diagram below.

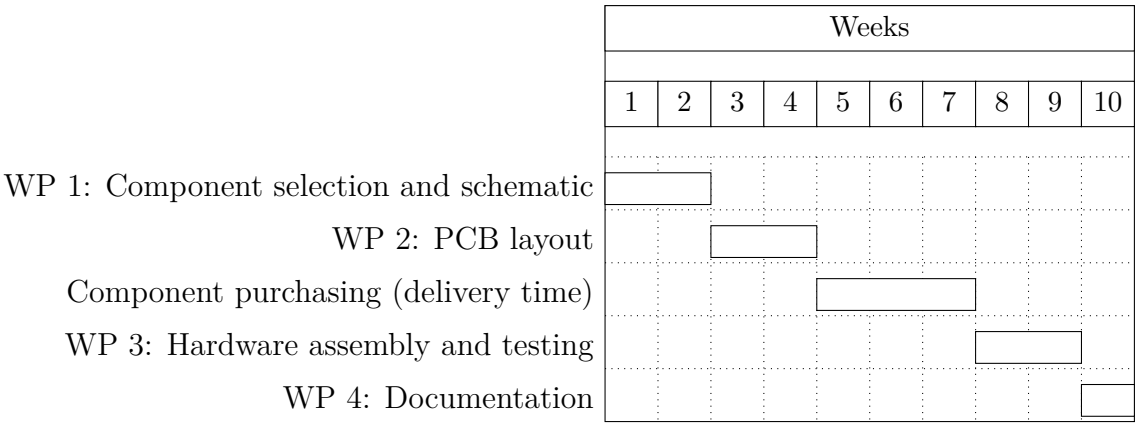

Figure 2: Gantt chart for the project (assuming student full time work on project).