WIP: Teaching Advanced PCB Design in a Collaborative, Project-Based Learning Approach

Sebastian Zisch, David Riehl, Klaus Hofmann, Ferdinand Keil
Integrated Electronic Systems Lab (IES)
TU Darmstadt
Darmstadt, Germany
{sebastian.zisch, david.riehl, klaus.hofmann, ferdinand.keil}@ies.tu-darmstadt.de

***Abstract*—This innovative practice work-in-progress paper describes an integrated course to introduce students to challenges in real world Printed Circuit Board (PCB) design. With the increasing complexity and speed of modern electronics, PCBs have become an integral part of electronic systems. Thus, acquiring knowledge in the field of PCB design is of great importance for aspiring engineers. At the same time the cost to manufacturing PCBs has fallen considerably, that even advanced processes are within reach of design courses taught at the university level. In the lectures, the PCB design workflow is introduced to establish a baseline between all learners. Part selection, schematic capture and creating a layout of a two-layer PCB is demonstrated live. Design rules are established, allowing more effective design reviews. The students acquire knowledge on advanced topics, such as multilayer technology, signal integrity and manufacturability. Exercises allow students to consolidate the most important learnings with impulse discussions. An integrated lab is successfully implemented and encourages students to practice PCB design. Using open-source software throughout the course lowers the barrier for students to get started with their designs.**

***Index Terms*—printed circuit board, electronics, integrated lab, untergraduate**

I. Introduction

The PCB is a fundamental component in modern electronic devices and can be found in nearly every part of daily life. From the smallest children's toys to process monitoring systems in industry, the size and complexity of the underlying electronic circuits vary massively. It is not only used to hold the components, but allows for freedom and miniaturization in the assembly. Deeply integrated systems, in the growing market of industry 4.0, integrate a vast range of functionality in a confined space. For example, in order to predict the lifespan of a timing belt, flexible PCBs allow integration of wireless sensors directly into the mechanical system [1]. Device assemblies can be scaled down and manufacturing costs reduced by using a rigid-flexible PCB, as they are lightweight, thin and minimize cable management [2].

The digital transformation in industry, assisted driving in the automotive sector or healthcare wearables are strong drivers for the PCB market [3]. Therefore, engineers with experience in PCB development possess skills that align well with the needs of the industry. At TU Darmstadt, PCBs are often integrated into labs as tools for experiments [4]. Other courses encourage creating a PCB to assist in solving a predefined problem. Although it is only a small part of the solution, students underestimate time consumption for a working design. Oftentimes, only one revision fits into a seminars timeframe, diminishing motivation to continue PCB layout if the desired result is not achieved. Education in advanced PCB design can help to better estimate design time and effort to successfully integrate PCBs into a project.

II. Previous Works

PCB design is already included in various electrical engineering courses. A design workflow using industry standards was developed by Kim and Schubert [5]. Given a predefined design task, the students had to optimize their layout in terms of price-driving factors. Two different tools, Multisim and Ultiboard, were used for schematic and layout respectively. The final layout was automatically evaluated for efficiency against a design created by an instructor. The course was limited in a budget, therefore not every student got to test their design after manufacturing.

A more product-oriented course incorporating PCB design was realized by Dankovic et al. [6]. The emphasis in this course was to take advantage of the interaction between mechanical and electrical Computer Aided Design (CAD) tools. Altium Designer was used to create the necessary design files for the circuit board.

A project-based course was developed and implemented by Jiao and Brakora [7]. They offer a PCB design project as part of an existing embedded interface class with given requirements. Teams of students work with Altium Designer to create a PCB, that fulfills all requirements for a self-defined use-case. By utilizing inexpensive circuit board fabricators and the university lab, the students were able to order and assemble the design.

PCB design in an interdisciplinary setting was researched by Bogdanovs et al. in 2022 [8]. The necessary design files were created using Altium Designer, which the students found hard to get comfortable in. The project-based approach was successfully implemented and resulted in good preparation for the students for their future careers.

III. Desired Learning Outcome

This paper presents a work-in-progress course that introduces undergraduate learners at TU Darmstadt to advanced concepts and processes in the PCB design space in a collaborative way using open-source tools. It was held in wintersemester 2023/2024 for the the first time. As a course offered in the 5th semester for students from the electrical engineering bachelors program, basic knowledge on electronic circuits can be assumed.

Students participating in project-based seminars in microelectronics, sensor systems and other areas regularly have to design custom circuit boards. Although it is often just an aid to support the research, it requires many revisions before a sufficient result is achieved. At the end of this course, every student should be comfortable with common tools in a PCB design workflow. With more in-depth knowledge on PCB design, the students will be capable to create more complex circuit boards successfully. Some of the desired learnings of the lecture and excercise are:

- Challenges in integrating electronics and connectivity on a PCB level can be approached methodically.
- Students have the ability to generate readable schematics by adhering to consistent design rules and develop functional layouts.
- They can conduct design reviews, identify problematic areas in layouts and propose appropriate measures.
- They can argue when to use multilayer stackups and choose appropriate materials for a functional layout.
- Signal integrity and trace geometries can be calculated to fulfil transmission line requirements.
- The students can design for manufacturability and reliability.

At the end of the voluntary integrated lab, the students will hold a manufactured and assembled PCB in their hands. The assembly is done by an external manufacturer. Every student taking part in the lab will have practical experience in a complete PCB design process. By analyzing the finished design, they can improve their workflow and result of the next design.

IV. Course Description

The course is divided in three parts

- A lecture to introduce advanced topics in PCB design,
- An accompanying excercise, providing more in-depth knowledge on selected topics,
- An integrated lab, which allows to gradually put the knowledge to reality.

Per week, a 1.5 h lecture is held in a lecture hall with a powerpoint presentation as the main teaching method. The equally long excercises are mostly intended as impulse discussion problems for selected topics. Lastly, the practical lab is integrated into the excercises in form of design reviews.

A. Teaching with Open-Source EDA-tools

The knowledge of the students ranges from no experience in PCB design to some experience in high-speed design. Hence, the first four lectures build a common base of knowledge. An introduction to the history and development of PCBs is given, to motivate creative projects and benefits of a good design. This is accompanied by fundamentals in a PCB design workflow, namely drawing a schematic, routing the layout and finally sending it off for manufacturing. Today, this is done using Electronic Design Automation (EDA) tools, which integrate all necessary tools. Most of the industry standard tools are tied to a licensing model, raising the barrier to practice PCB design. Alternatively, open-source tools such as LibrePCB or OsCAD [9, 10] are available for free, but come with limited capability or a complex user interface.

KiCAD is a powerful open-source EDA tool that provides all necessary functionality and is consistently used throughout the lectures and excercises. First released in 1992, its development picked up pace since 2012 after investments of the open hardware foundation from CERN [11]. It is very well suited for undergraduate studies at a university for several reasons:

- Lightweight compared to commercial products such as Altium Designer, with an easy to learn user-interface.
- Schematic capture, layout editor, symbol and footprint library manager
- It runs, free of charge, on Linux, MacOS and Windows equally.
- Large development community to add more complex functionality to provide a complete EDA suite, with a 3D viewer, a BOM manager and a PCB calculator.
- Good online and offline documentation [12, 13].

A big advantage is the library, which already contains a lot of standard components with fitting footprints and 3D models. This is beneficial to get started on a schematic design, because this part of the work is already done beforehand. At the same time, the libraries are suited as a template in case own footprints are to be drawn. KiCAD also integrates several footprint wizards to, for example, generate BGA footprints or even capactitive touch slider. Not too long ago such features were the realm of high-end commercial design software, which underlines the progress that has been made recently in the open-source EDA software space.

B. Schematic and Layout Demonstration

The schematic presents the functionality of a design on a high level [14]. In the lecture, a set of guidelines similar to grammar in written language is established to create a uniform visual, collaborate on the same design more efficiently and prepare the students for working in an industrial environment.

Reading rules can be tiresome and, without practical application, they are often quickly forgotten. Therefore the design rules are explained and demonstrated live by going through a schematic design process for a simple headphone amplifier, allowing discussion with the audience. The rules are standardized for schematic diagrams in the IEC 61082-1 [15]. The final look of a design cannot be standardized, as every engineer has their own logical way of organizing it. Hence, in industry, commonly used components are often already included in internal libraries. In Altium Designer for example, a new project has an empty symbol and footprint library by default. Students with no prior education in PCB design, might find it difficult to start with even a simple design, because they first have to manage component libraries.

C. *Advanced Topics*

Signal integrity is an essential part in high-speed PCB design and is often underestimated by novices. In the lecture and excercises, common signal integrity problems are discussed with a real-life example. A layout containing an ESP32 microcontroller, which sends data over an ethernet adapter inherited a signal integrity error, resulting in a failed connection. Using a real-world measurement from an oscilloscope and the circuit diagram, the issue is discussed with the students in the lecture.

Crosstalk was known to some students already, however the cause and implications are often not. For this reason, a demonstration board with different trace topologies was created to showcase signal integrity (fig. 1). A pulse source with a $30\,\mathrm{ns}$ rise-time was utilized to create a measureable crosstalk. Return currents can be visualized using a magnetic probe by Aim-TTi [16].

Simulation is introduced as a way to analyze a problem in a controlled environment. Using the *tline* element in LTSpice, the impact of termination on a signal travelling a transmission line is shown. In the excercise, the students calculate a reflection diagram of an arbitrarily chosen transmission line, which helps to develop an understanding how a signal traverses a copper trace. Transferring this knowledge to PCBs, different trace geometries are discussed.

Diving deeper into more advanced PCB technology, flexible circuit boards (FCB) are a continuously growing market [17]. In the lecture, the manufacturing details are presented and material choices discussed. A complementary excercise, dives deeper into applications and pitfalls. KiCAD has no specific flexible PCB mode. On the other hand, Altium Designer offers a specialized rigid-flex stackup mode, with which the process of creating and optimizing a FCB layout can be shown.

In industry, reliability of electronics is an important factor. Failing electronics can lead to safety problems or process interruptions and often result in high cost for maintenance and production losses. Research regarding lifetime of LED-Drivers showed, that a faulty PCB layout can lead to catastrophic failures [18], which is an important factor considering security. Due to these reasons one lecture will present failure mechanisms of PCBs, for example dielectric breakdown and electrochemical migration. In the excercises, this is further

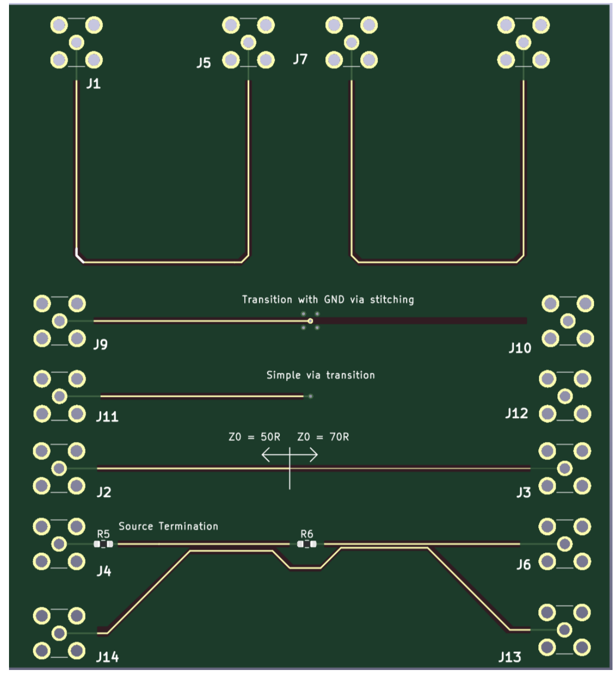

Fig. 1. Demo-board for signal integrity measurements. Each track between SMA connectors shows a different trace characteristic. On the top, return currents for high frequency can be demonstrated. Three straight tracks inherit different impedance transitions. Lastly, differently spaced tracks allow crosstalk visualization.

analysed with a root cause analysis. The students are presented a defective PCB and are to identify the cause for the failure.

V. COLLABORATIVE PCB DESIGN

It takes practice to master the design of printed circuit boards. Therefore the students are encouraged to put their own idea into reality with the knowledge gained in the lecture and excercise. The integrated lab is divided in four timeframes, in which different project tasks have to be finished:

1) Block diagram
2) Schematic capture
3) Layout
4) Send-off to manufacturing

The timeframes offer guidance to aid with time management. In each of the phases, the results will be discussed in design review sessions instead of an excercise. In the review sessions, each design that was uploaded by students was shown on the beamer. The student designer can present the ideas and thought process that lead to the result for other students. This way, discussions and questions with other fellow students are effected. Simultaneously, other students can learn from more experienced students and incorporate new ideas in their design.

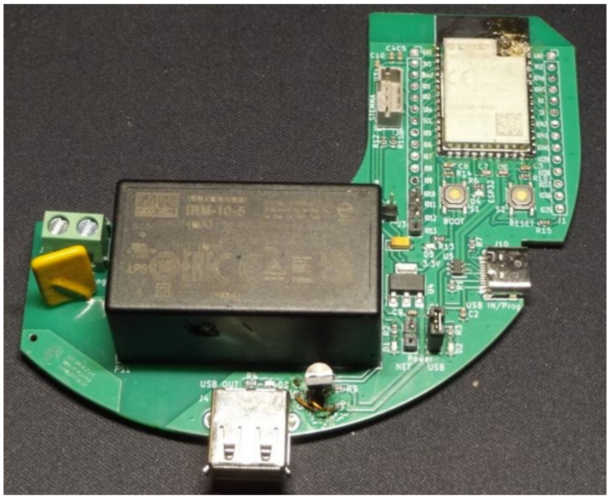

Fig. 2. Smart charging adapter for upcycling of an old electric lamp, integrating high-voltage and low-voltage components on a small space.

A. Project outline

The project is only vaguely introduced to allow freedom in the concept phase while stating their own requirements. Similar to [7], the project is intended to be worked on alongside the lecture and excercise. With no extra laboratory time, the students are expected to work without direct supervision. This is encouraged by offering a bonus to the final grade of up to 0.4.

As a creative framework, two options are proposed. For more experienced students, the small, well-known "Feather" formfactor [19], pioneered by Adafruit, poses a challenge for a dense design. Alternatively, an outline of maximum $100\text{x}100\,\mathrm{mm}$ offers more creative freedom. The designed four-layer PCBs are ordered fully assembled from a manufacturer. For simplicity, the part selection is limited to what is available from the manufacturer. Due to optimized manufacturing processes, simple four layer boards can be purchased starting at 7$ for five pieces excluding shipping [20]. This enables universities to offer project-based courses under industry conditions with a physical product.

B. Collaboration approach

As the e-learning platform, moodle LMS [21] was used, because it also provides the platform for course management. An exchange between students outside of regular lectures and excercises is possible through a standard forum. In order to preserve the knowledge from students for future courses, a glossary is maintained.

In contrast to laboratory sessions, the collaboration is not limited by group size. The *Peer Review* function of moodle is well suited for collaboration. For each design phase, a separate section is instantiated. In the first two phases, a PDF for a block diagram and schematic is sufficient. The students can download each others schematic and highlight problematic sections, write comments or ask questions.

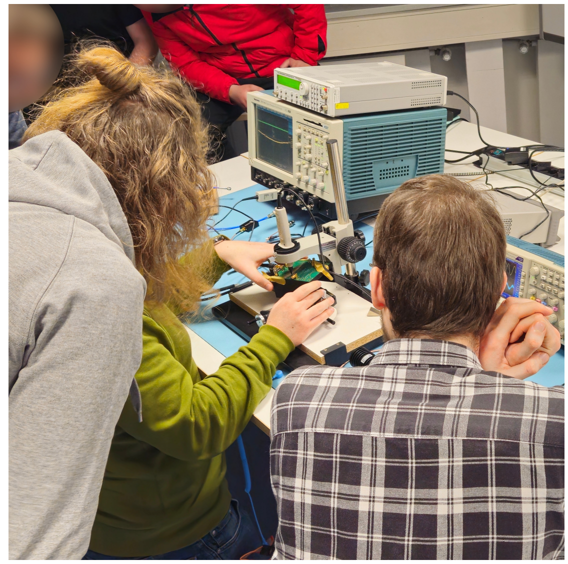

Fig. 3. The students collaborating in measuring EMI phenomenons on the demonstration board

The layout review is more challenging, essentially tow options for collaboration are possible. A simple, but inflexible approach is to print each layer individually, similar to schematics. The better option is to share the KiCAD PCB file, which embeds all footprints. In case of multi-layer designs, it allows a better overview and feedback can be given with screenshots.

VI. Conclusion

This integrated course presents an innovative approach to teach advanced topics in PCB design to undergraduate students. The use of open-source software lowers the barrier to get started in PCB design. Live demonstrations, experiments (fig. 3) and impulse discussions help students to consolidate their knowledge. An integrated lab encourages students to put their knowledge into practice. A finished assembly from the manufacturer saves time and is less prone to assembly mistakes. Students can visually and physically inspect their design, thus aquiring valuable design experience.

Many creative solutions were presented, for example upcycling of old electronics (fig. 2). A working PCB design was handed in by 80% of the students that attended the first instalment of the lecture. The design rules and guidelines were well received and helped to create productive design reviews for the schematic and layout. The most discussed topics were part selection and connection diagrams. More experienced students could shared their insights and helped others incorporate these ideas in their design. The e-learning platform was not frequently used, instead the impulse discussions in the excercises were more popular. Feedback from students on the course design and content was very positive, especially from those, who had very little experience beforehand.

References

[1] D. Großkurth and G. Martin. "P2.14 Intelligenter Zahnriemen". In: *Tagungsband*. Nürnberg, Germany: AMA Service GmbH, Von-Münchhausen-Str. 49, 31515 Wunstorf, 2019, pp. 738–743. DOI: 10.5162/sensoren2019/P2.14. (Visited on 05/06/2024).

[2] Keith Araujo-Epec LLC. *Cost Benefits of Using Rigid-Flex PCB vs. Rigid PCB and Cable Assembly*. https://www.epectec.com/articles/cost-benefits-of-using-rigid-flex-pcb-vs-rigid-pcb-and-cable-assembly.html. (Visited on 05/17/2024).

[3] *Printed Circuit Board Market Size, 2022-2030 — Analysis PDF Report*. https://www.gminsights.com/industry-analysis/printed-circuit-board-pcb-market. (Visited on 05/06/2024).

[4] David Riehl, Ferdinand Keil, and Klaus Hofmann. "From Breadboard to Complex Electronic Systems - Introducing a Heterogenous Group of Undergrad Students to Design and Analysis of Electronic Circuits". In: *2023 IEEE Frontiers in Education Conference (FIE)*. College Station, TX, USA: IEEE, Oct. 2023, pp. 1–5. ISBN: 9798350336429. DOI: 10.1109/FIE58773.2023.10343516. (Visited on 05/07/2024).

[5] Ernest M Kim and Thomas F Schubert. "A Low-Cost Design Experience for Junior-Level Electronics Circuits Laboratories through Emulation of Industry-Printed Circuit Board Design Practice". In: *The International Journal of Electrical Engineering & Education* 54.3 (July 2017), pp. 208–222. ISSN: 0020-7209, 2050-4578. DOI: 10.1177/0020720916673650. (Visited on 05/02/2024).

[6] Danijel Dankovic et al. "An Electromechanical Approach to a Printed Circuit Board Design Course". In: *IEEE Transactions on Education* 56.4 (Nov. 2013), pp. 470–477. ISSN: 0018-9359, 1557-9638. DOI: 10.1109/TE.2013.2257784. (Visited on 05/13/2024).

[7] Karl Brakora and Lihong (Heidi) Jiao. "A Project-based Printed Circuit Board (PCB) Electronics Course". In: *2020 ASEE Virtual Annual Conference Content Access Proceedings*. Virtual On line: ASEE Conferences, June 2020, p. 34037. DOI: 10.18260/1-2--34037. (Visited on 05/02/2024).

[8] Arturs Bogdanovs, Alexander Bubovich, and Ilya Galkin. "Interdisciplinary Project-based Learning Approach Implementation for Undergraduate Electrical Engineering Students". In: *2022 IEEE 63th International Scientific Conference on Power and Electrical Engineering of Riga Technical University (RTUCON)*. Riga, Latvia: IEEE, Oct. 2022, pp. 1–7. ISBN: 978-1-66546-558-8. DOI: 10.1109/RTUCON56726.2022.9978862. (Visited on 05/02/2024).

[9] *LibrePCB*. https://librepcb.org/. (Visited on 05/17/2024).

[10] *Oscad: Open-Source Computer-Aided Design Tool*. https://dl.acm.org/doi/fullHtml/10.5555/2631585.2631589. (Visited on 05/17/2024).

[11] *KiCad, CERN and Component Libraries – MyriadRF*. https://myriadrf.org/news/kicad-cern-and-component-libraries/. (Visited on 05/07/2024).

[12] *Documentation — KiCad*. https://docs.kicad.org/. (Visited on 05/08/2024).

[13] Peter Dalmaris. *KiCad Like a Pro*. Tech Explorations. London: Elektor, 2018. ISBN: 978-1-907920-74-5.

[14] Gerald Zickert. *Leiterplatten: Stromlaufplan, Layout und Fertigung*. 3., aktualisierte Auflage. München: Hanser, 2023. ISBN: 978-3-446-47858-9 978-3-446-47583-0.

[15] *DIN EN 61082-1 VDE 0040-1:2015-10*.

[16] *I-Prober 520 Meter Datasheet Details*. https://resources.aimtti.com/datasheets/AIM-I-Prober_520_meter_data_sheet-Iss1A.pdf. (Visited on 05/13/2024).

[17] *Flexible Circuits Market Size To Hit USD 65.24 Bn By 2032*. https://www.precedenceresearch.com/flexible-circuits-market. (Visited on 05/08/2024).

[18] Ferdinand Keil. "Accelerated Lifetime Testing of Off-Line LED Drivers". In: (2023). DOI: 10.26083/TUPRINTS-00023782. (Visited on 05/08/2024).

[19] *Introducing Adafruit Feather*. https://learn.adafruit.com/adafruit-feather/feather-specification. (Visited on 05/13/2024).

[20] *JLCPCB Order*. https://jlcpcb.com/. (Visited on 05/07/2024).

[21] *Startseite — Moodle.org*. https://moodle.org/. (Visited on 05/13/2024).