ASEE'S VIRTUAL CONFERENCE
At Home with Engineering Education
JUNE 22 - 26, 2020 #ASEEVC
Paper ID #30554

Development of a Printed Circuit Board Design Laboratory Course

Dr. Pelin Kurtay, George Mason University

Pelin Kurtay is Associate Professor and Associate Chair of the Electrical and Computer Engineering (ECE) Department at George Mason University. She currently heads the ECE Department's undergraduate curriculum development efforts and leads other departmental initiatives. She is the recipient of the 2015 Teacher of Distinction Award at George Mason University for exceptional teaching and commitment to teaching-related activities in electrical and computer engineering and Information technology. She is a senior member of the IEEE.

©American Society for Engineering Education, 2020

Development of a Printed Circuit Board Design Laboratory Course

With the increasing focus on more sophisticated design projects in electrical and computer engineering (ECE) curricula, the number, quality and complexity of projects that students complete by the time they graduate has been growing. Aside from culminating design projects usually completed during their senior year, many undergraduate programs have been incorporating design experiences throughout the curriculum, even starting as early as the freshman year. The emphasis of many institutions on entrepreneurial activities as well as the prevalence of multidisciplinary projects through student clubs have also increased the need for students to rapidly prototype their designs. As many of these activities require the design of electronic printed circuit boards, it has become increasingly important for students to learn about the tools needed to efficiently design printed circuit boards (PCBs). In many ECE curricula, PCB design is expected either to be self-taught as part of the senior design process, is offered as a workshop, or is appended to another course such as circuit design. By specifically teaching PCB design principles through a formal lab course, students are better equipped to implement increasingly complex projects during their curricular experiences. Having learned the basic design principles and tools necessary for PCB design, students also become better positioned to participate in various design competitions through student clubs and other organizations.

The lab course developed and described in this paper provides students with an opportunity to gain skills in the fundamental PCB design principles, which they can then use to build more advanced board circuitry as they progress through their curriculum. The course is offered as a full semester distance education offering and hence provides students with the flexibility to complete their weekly lab assignments without the need to physically come to campus each week. The paper provides a comparison of PCB design software and the justifications for adoption of the chosen software tool for the course. Weekly lab experiments and their learning objectives are described, together with their achieved outcomes. The approach used for providing students with soldering training as well as the benefits and challenges of teaching the course online are also included. Experiences gained from offering the course during a full 15-week semester are discussed, and recommendations for potential future offerings are presented.

Introduction

Printed Circuit Board (PCB) design has become a desirable skill to have for electrical (and computer) engineering students. Such skills are especially relevant if prospective graduates are looking for hardware related jobs in the industry. Not only does it make them more attractive to potential employers upon graduation, having PCB design experience can be a facilitator to provide more enriching experiences throughout a student’s time in an engineering program within a higher educational setting. With hands-on design experiences being increasingly emphasized within the curricula of a growing number of engineering programs, students in many institutions are expected to produce sophisticated projects as part of coursework or student organization activities. Many of these hands-on projects that may be multidisciplinary in nature involve the design and manufacturing of a Printed Circuit Board. The drawback, however, is that oftentimes students are expected to learn the design principles, select a software tool, as well as learn how to use the software for designing a PCB on their own. The importance of teaching PCB design through a formal course has been a topic explored by other faculty groups at various institutions [1], [2]. There are some higher learning institutions who offer courses on PCB design such as in the form of popup courses [3], or through student organizations such as the IEEE student chapter [4] however many of these offerings are traditionally non-credit and not typically counted towards a faculty's teaching load [5]. There are some institutions that have integrated formal PCB design as part of their curriculum, however it is usually part of another course.

PCB design activities may typically occur sometime starting in the sophomore (or later years) where classes increasingly incorporate more hands-one project implementations (though such projects at a simpler scale can even be implemented the freshman year). Even though there are a number of online resources for learning about PCB design and selecting software, students may find themselves unable to decide on a good tool to use and end up producing boards that are less efficient and do not fully meet design constraints. This causes them to spend more money and resources getting boards re-manufactured (if they are using a third party provider, for example) or having to re-manufacture the board again (if they are doing it in-house). In order to meet the need of such students in the Electrical and Computer Engineering Department at George Mason University, we developed a course that covers basic PCB design principles and enables students to earn credit towards this experience. The sections below describe the development of such a 1-credit course that students take to fulfill a major requirement in the electrical engineering program and discusses the structure of the course and its expectations, as well as the valuable lessons learned from a pilot offering that was offered.

Course Requirements

The PCB Design Lab was offered for the first time during the fall 2019 semester at George Mason university where a full semester is 15 weeks. It was offered as a special topics course and fulfilled a lab elective requirement for the electrical engineering program. Both the electrical and computer engineering programs offered by the ECE department are accredited by the Engineering Accreditation Commission (EAC) of ABET. There were around 265 electrical engineering and around 230 computer engineering students enrolled as of the writing of this paper. Since this was a pilot online offering, capacity was limited. There were 21 students enrolled (full capacity) by the time the semester started. Most students in the class ended up being juniors, although there were some sophomores as well as seniors. Most of the seniors had not yet started their 2-semester sequence senior design project effort.

The course was offered as a hybrid online offering, thereby giving students the flexibility to complete majority of the work including the lab exercises outside of a physical lab. A large percentage of students in the ECE department work either part time or full time, so the partially online nature of the course made it especially accommodating to this group of students. Students were required to complete weekly lab activities and to submit items for grading at the end of each week. There were a total of 8 lab activities, a midterm exam and a project component with the following grade breakdown:

Weekly Lab Activities: 50 %

Midterm Exam: 15 %

Project: 35 %

At the start of the semester, each student was required to complete a course orientation quiz (non-graded) which was to ensure that all those enrolled were fully prepared to complete the course requirements and possessed the necessary background. Even though the course was open for registration as an online offering, the instructor wanted to ascertain that students understood they were required to be on campus for some activities such as the midterm exam, soldering training and the project presentation at the end of the semester. Other aspects of the orientation focused on confirming whether or not they had downloaded the required tools and that they would need to utilize the services of a third party PCB manufacturer in the manufacture of their board designs for their projects. The third party manufacturer of choice was a provider called OSHPARK, which provided the class with a very convenient way to upload their PCB designs directly generated through the KiCAD software (which was the software chosen for this class) and receive 3 copies of their manufactured boards. Other PCB service providers were evaluated, but eventually OSHPARK proved to have a simple user interface, fast turn-around times and met the basic requirements for the boards. Boards that were designed in the class ranged from 2-4 layers which was well within OSPHARK's manufacturing capabilities.

PCB Design Software

There is a wide array of PCB design software tools available for use both within the industry as well as for academic purposes. It can be a daunting process to choose which one to use, as many of them are very good and have very sophisticated features and capabilities. Some of the more prominent software tools used in the industry include Altium Designer, OrCAD, and EAGLE. There are a number of others, each with their own unique features, however only a couple will be discussed here, as they are the deemed to be the most widely used. The first two major tool providers listed above provide universities with software licenses to download and use the software. Altium charges a fee for university classroom or lab use but is free for students to download [6]. OrCAD PCB designer is a tool provided by CADENCE and similarly requires a fee by academic institutions participating in its academic program [7], [8]. Autodesk provides free individual educational licenses for access to the premium version of EAGLE and the software can be freely downloadable by educators and students by creating an account with the company [9]. Another popular software is KiCAD, which is the software adopted for this class. It is open source and is currently under development by a KiCAD Developers Team [10]. Since it is open source, it is freely downloadable and is constantly growing through its collection of contributors.

It was found that many industry positions that were advertised on job sites seek candidates with knowledge and experience of mainly Altium, EAGLE, or OrCAD. KiCAD, on the other hand, is considered to be more popuar among hobbyists and electronics enthusiasts. Through experiences gained from past senior design project teams and instructors, it was decided that KiCAD was going to be adopted for this class due to it being free, its ease of use, being open source, as well as being able to operate on any platform such as Windows, Linux, OS X etc. Although a specific software such as KiCAD was adopted for instructional purposes, it is straightforward for students to learn other tools once they have mastered the fundamental design principles and become familiar with the steps associated in creating a board design.

Lab Activities

Weekly lab activities required students to design various parts of a circuit board, starting from a very basic board involving the design of a power supply. Each lab exercise taught students a new feature of KiCAD and built upon previous knowledge by utilizing the tools learned in earlier labs. The instructor adopted the e-book by Peter Dalmaris of Tech Explorations [11]. Lab exercises were directly mapped out from the activities presented in the book and aligned to match weekly tasks. Students were expected to complete their designs by incorporating the principles contained in the textbook and to follow the useful step-by-step instructions provided by the author.

By the end of the semester, the instructor's goal was to have students be able to design at least a 2-layer PCB from start to finish. The goal was also for students to be well acquainted about how PCBs were manufactured (including the raw materials used), as well as understand the various tradeoffs involved in the design process, some of which include cost, manufacturability, speed, physical board size, and component size/placement. To this end, the following lab activities were outlined, where students were required to complete a lab activity each week during the first half of the semester:

Lab 1: Getting acquainted with the general features of the software, types of electronic files associated with a board design project, starting to draw a circuit schematic and applying electrical rules checking (ERC) to the schematic design.

Lab 2: Importing netlists from schematic design, creating a board layout, positioning components, adding footprints, dimensioning, specifying board size and shape. Reviewing OSHPARK's minimum design rules to ensure board design features fall within manufacturing specifications.

Lab 3: Drawing electrical traces, trace width calculations, routing concepts, running design rules checking, implementing fill zones and keep out areas. Creating graphical and textual elements on the board. Using 3D viewer to view board in progress.

Lab 4: Understanding the various tradeoffs between single-layer and multi-layer board design, incorporating vias into the design, routing with vias and working with manual and auto-routing options. Understanding factors that impact manufacturing cost and implementing design tradeoffs.

Lab 5: Synthesizing knowledge gained in previous labs to design a simple 2-layer board. Receiving feedback on the board design by the instructor and sharing various common design mistakes made by students.

Lab 6: Learning about the various steps of the PCB manufacturing process involving chemical, mechanical, electronic and optical processes. Learning about the raw materials and equipment used in PCB manufacturing. Reviewing in-house mechanical PCB manufacturing techniques. Understanding how to generate Gerber files and what the content of each Gerber file means to a board manufacturer. Submitting a project proposal.

Lab 7: Adding symbol and footprint libraries from external sources. Designing custom symbols and footprints. Generating a bill of materials. Surveying electronic component suppliers and choosing one or more suppliers from which to order parts. Designing the circuit schematic for the project, receiving feedback from instructor on design and ordering parts for the project.

Lab 8: Understanding different component and chip packaging technologies and differences between surface mount (SMT) components compared to through-hole based boards. Comparing differences in design between SMT and through-hole design approach. Verifying project operation by simulation/constructing circuitry on a breadboard and implementing testing.

After the last lab, students were required to attend an in-person exam during the 9th week where they were given a circuit diagram of an analog electronic circuit together with some design rules, which included constraints around placement of components, dimensions as well as manufacturability. They were required to draw the schematic diagram in Eeschema, the schematic editor tool of KiCAD, and then to create a PCB design using PCBNew (KiCAD's layout editor). All students were required to bring their laptops to the exam with KiCAD installed (as well as the e-book to use as a reference) to generate individual board designs. Part of the evaluation criteria included the extent to which they utilized the built-in tools of KiCAD, the adherence to PCB design principles learned during the first half of the semester and the extent by which they avoided design mistakes. Designs were also evaluated based on the degree to which their boards adhered to the constraints given in the specifications.

Project Component

As mentioned earlier, another requirement of the class was for students to work on an individual project to be presented at the end of the semester during an open showcase. Students were asked to submit a proposal and the instructor gave feedback on the acceptability and viability of the project proposed. Evaluation criteria for projects included factors such as the realistic time-frame for completion, ease of access to parts as well as whether or not it was a workable circuit. Once the instructor approved the project, students were given the go-ahead to start work and looking into purchasing electronic, electrical, and mechanical parts as needed for their designs while making sure to review data sheets and footprint specifications of components. They were encouraged to verify their circuit designs such as through simulation and on a breadboard before placing an order for the board. Projects that were not approved went through a revision process to either modify the design or to come up with a new design. Projects ranged from analog circuit designs such as audio amplifiers and noise generators to those involving digital circuitry involving microcontrollers and embedded systems. Many students also chose to submit their board designs to the instructor for feedback prior to placing an order with the manufacturer. This reduced the number of students having to re-order their boards.

Project grading criteria were as follows:

-Successful demonstration of working project-50%

-Successful completion of soldering training-10%

-Quality/neatness of soldering -10%

-Adoption of PCB design principles used in the design of the board (e.g. proper usage of vias, avoidance of PCB design mistakes, usage of silkscreen elements, connectors, etc.)-20%

-Summary of lessons learned during the PCB design/manufacturing/testing/implementation process: 10%

-Bonus: Up to 10 points: Additional design elements/complexity used in the implementation of the project

Once every student received their board from the manufacturer and completed soldering training, the rest of the weeks in the semester were spent soldering components and troubleshooting boards. Some boards were sent to be re-manufactured due to irrecoverable design errors (such as boards from students who had not chosen to receive instructor feedback). During this process, students also learned how to reconfigure circuit connections using techniques such as the application of enameled magnet wire to circumvent around erroneous paths.

Project presentations occurred during the last class of the semester in the form of an open session/showcase. Each student was given the opportunity to describe and demonstrate the operation of their project in front of their peers and shared their individual experiences and lessons learned. All students in the class were able to see each other's board designs and ask questions during this open session. Everyone was encouraged to share what they would be doing differently the next time around and discussions ensued around this topic.

Soldering Training

After the boards were ordered through OSHPARK, there was a mandatory in-person training on campus. Multiple soldering sessions were offered and students were allowed to sign up for one of the 5 sessions offered. This gave them the flexibility to choose a day/time that best fit their schedule. Given that this was an online course offering, there was no guarantee that every student would be able to attend the session if only one were made available, so multiple sessions were made available.

Soldering training also included a tour of the PCB protoyping lab and a review of how the machines and other equipment found in the lab were used to create 2-layer boards. Although students were not required to print them in-house, it was important for them to see first-hand how a PCB was created, starting from its basic raw material of a 2-sided copper board to a finished PCB. The lab tour also discussed the use and soldering of SMD components via a demonstration using a hot reflow oven. After the lab tour, students were given access to soldering stations and issued components to practice their soldering skills. Prior to any soldering activity, a safety training was provided, as well as a review of the variety of chip and component packages commonly found on PCBs. After a soldering demonstration a teaching assistant (TA) assisted the students, together with the instructor in developing their soldering techniques. Various items were soldered, including adjoining of bare wires, soldering items onto a proto-board, soldering through-hole components as well as soldering of discrete SMD components onto PCBs. The soldering training culminated with each student having to demonstrate examples of various components they soldered onto sample PCBs. Criteria for evaluating soldering success included the proper utilization of soldering paste, the amount of solder used for each connection, neatness and shape and the visual inspection of potential cold solder joints, among other considerations. Although many students had soldered before, the training was an opportunity to refresh and refine these skills. All students who had prior soldering experience indicated they benefited from the training.

Lessons Learned by Students

As part of the project experience, one requirement was to talk about and submit a summary of lessons learned after the demonstration session. This gave each person in the class an opportunity to reflect on their experience to critically evaluate what they would be doing differently the next time they designed a board. Much of the feedback received indicated that students did not fully simulate or spend sufficient time building the circuit on a breadboard first before proceeding with the manufacture of their design. Others commented on ordering the required components/parts first and then to select the appropriate footprints on the board (rather than selecting a footprint first and then trying to find the appropriate part). Some project designs contained footprints that were hard to find corresponding components for, so those students indicated they would order parts first and then proceed to selecting the footprint either from the existing footprint libraries or designing their own footprints. Other feedback included not starting the design process early enough and not sufficiently optimizing the real estate space. Since PCB manufacturers typically charge by the square inch of board space for 2-4 layer boards, some students ended up ordering boards with empty real-estate space and hence paying for more than they needed. It became apparent that component placement optimization was one aspect of the designs that a number of them needed to pay more attention to.

Modifications for future offerings

The pilot offering of the class during the fall 2019 semester yielded very encouraging results as demonstrated by the success of the project demonstrations and all the positive feedback from the students received at the end of the semester. Students found the book easy to follow and the software easy to use. They appreciated the opportunity to design boards on their own and to gain experience ordering parts from various suppliers discussed in the class. Overall, they felt more confident going in to senior design. Since some of the students in the class were also involved in student organization activities such as the student chapter of the Institute of Electrical and Electronics Engineers (IEEE), the class gave those students the tools needed to work on prototyping designs for group projects.

The plans for modifications during future offerings of the course include the potential incorporation of a second software tool such as EAGLE. Once the basic design principles are covered, the plan is to have students design the same board using two different software: One with KiCAD and one with EAGLE (or another tool). This is hoped to enrich the engineering design experience and highlight the differences as to how the same result can be achieved with different tools.

Another potential modification is to have students manufacture their boards in-house. This will require the department to potentially invest in some additional equipment and tools, however it will enable students to more thoroughly grasp concepts related to materials and mechanical aspects of board manufacturing. One additional consideration is to incorporate more time to reviewing data sheets; specifically how to read and interpret dimensions of components more thoroughly and match them to footprints on the board. Spending time on footprint issues and dimensioning will also benefit students in their component selection process.

Additional hands-on and theoretical concepts are also planned to be covered such as issues related to electrical decoupling, Design for Testing (DFT), Design for Manufacturing (DFM), thermal considerations, as well as other techniques such as high speed board design and digital board design. The instructor will also require all students to utilize simulation as an integral part of the design verification process.

One of the challenges of this hybrid online class was the limited interaction between the students. The interaction between the instructor and the students was widely available and supported through email, WebEx, Skype, Zoom and other tools. Although students had plenty of opportunity to interact with the instructor at each step including the verification and feedback on their designs, future offerings of the class will potentially have to build more opportunities for students in the class to interact with each other. This may potentially be in the form where students can comment on each other's designs including component selections, footprints etc. or by incorporating group projects where team members can work on a design collectively (including working remotely).

Conclusions

This paper described a course developed for equipping electrical engineering students of the ECE department at George Mason university with the techniques and tools needed to efficiently design PCBs using an open source software that is freely available and having boards manufactured through an external provider. By taking this class an earning formal credit, students are more motivated and better prepared to tackle more sophisticated design projects such as for senior design. The department is currently in the process of looking into whether this course can be made a requirement to benefit all students (including those in computer engineering) and how it can be made available in the earlier years (such as in the sophomore year). The online nature of the class gave students a lot of flexibility and encouraged self-paced individual learning experiences. Future modifications will involve the incorporation of additional technical concepts as well as more opportunities for students to interact with each other.

References

[1] G. Aranguren, J. Etxaniz and L. A. López-Nozal, "Design of printed circuit boards in university?," *2012 Technologies Applied to Electronics Teaching (TAEE)*, Vigo, Spain, pp. 6-10, 2012.

[2] E. M. Kim and T. F. Schubert, "A low-cost design experience for junior-level electronics circuits laboratories through emulation of industry-printed circuit board design practice," *The International Journal of Electrical Engineering & Education*, *54*(3), 208–222, 2017.

[3] Texas A&M University Popup Courses Site

https://fedc.engr.tamu.edu/pop-up-classes/ (accessed Apr. 29, 2020)

[4] Hands-On PCB Engineering at University of California-Berkeley-IEEE student Branch

https://ieee.berkeley.edu/blog/hope/ (accessed Apr. 29, 2020)

[5] https://www.aps.org/careers/guidance/webinars/popupclasses.cfm

[6] Altium Academic Programs

https://www.altium.com/solutions/academic-programs/education-programs (accessed Feb. 3, 2020)

[7] OrCAD Academic Programs

https://www.orcad.com/orcad-academic-program (accessed Feb. 3, 2020)

[8] OrCAD Academic Programs (Through EMA)

https://www.ema-eda.com/products/cadence-orcad/orcad-academic-program (accessed Feb. 3, 2020)

[9] Autodesk-EAGLE Academic Licenses

https://www.autodesk.com/education/free-software/eagle (accessed Feb. 3, 2020)

[10] KiCAD Software for Electronic Design Automation

https://kicad-pcb.org/about/kicad/ (accessed Feb. 3, 2020)

[11] *KiCad Like a Pro 2e: A comprehensive hands-on guide for learning the world's favourite open source printed circuit board design tool*, Tech Explorations Publishing, 2018 [ebook]. Available: https://techexplorations.com/product/kicad-like-a-pro-2nd-edition-book/

[12] OSHPARK PCB Services

https://docs.oshpark.com/services/ (accessed Feb. 3, 2020)