Electronics – A First Course for Printed Circuit Board Design

Cheng Liu

Cheng Liu is a Professor in the Computer and Electrical Engineering Program at University of Wisconsin Stout. He taught courses in electrical, computer engineering, and engineering technology programs.

Jian Chang

Jian Chang is an Analog IC Designer with Texas Instruments in Dallas, Texas. He began working in circuit design for temperature sensors, power management ICs, Serializer and De-serializer in 2006. His current role is to support mixed-signal design verifications and design verification methodology for Texas Instruments.

Electronics – A First Course for Printed Circuit Board Design

The design of printed circuit board (PCB) is an essential aspect in learning circuits and systems in the electrical and computer engineering (ECE) curricula. In fact, many institutions have allowed for this opportunity in 3D printing labs for students to rapidly prototype their circuit designs and other 3-D printed projects. Further, we see a high importance for students to learn practical aspects of circuit design in the sophomore year, and to retain ECE students who feel a greater level of accomplishment in computer science courses or plan to drop out. In this article we discuss what can be done in an electronics course to possibly retain students in ECE programs.

Microprocessors, field programmable gate arrays (FPGAs), and microcontrollers are all essential components in electronic systems. The electronics course offers two levels of design experiences to students: 1) circuits and electronics and 2) PCB designs for manufacturability. These two experiences incorporate microcontroller and environmental sensor interfaces, in addition to Bluetooth connectivity and motor driver integration. As a result, this allows students to use the circuit interfaces on their PCB to build complex systems for smart home, automation, mobile, and industrial applications.

Herein, we detail the incorporation of PCB development in the electronics course offered in year-two of the ECE curriculum. The course is organized in a 15-week schedule that covers traditional circuit designs with diodes, transistors, operational amplifiers, but also includes electronic circuits design in schematic capture, PCB layout, soldering electronic components and microcontroller programming on a custom PCB. This paper also describes an upload process of the Arduino bootloader to a custom PCB and the use of the Arduino IDE to write code and upload it to the PCB. The approach used for soldering training to students and challenges of teaching the printed circuit design are also presented.

Introduction

Electronic circuit design has undergone changes over the last decades. Tremendous advances in semiconductors have catapulted electronics into all aspects of our lives. This offers new opportunities and challenges in teaching circuits and electronics to excite and motivate today's students [1]. The design focus on electronic circuits is more motivating than the conventional bottom-up approach, addressing a common dissatisfaction of students with circuits classes [2]. The design focus on a custom PCB can spark student's interest in electronics and expose students to physical aspects of circuit designs.

For the ECE students, this course is the first exposure to electronic circuits hidden in the mobile devices they own. However, most are familiar with computers and have programming experience with the open-source Arduino software and hardware, providing a natural path to connect their experiences to printed circuit boards. Many electronic circuits can be electromechanical in nature that involve with component selection, circuit design, and PCB production. Due to the nature of PCB designs, without any previous experience, students can initially be overwhelmed by their complexity. Therefore, the importance of teaching PCB designs has been a topic of interest by faculty at several institutions. Substantially similar courses in the past have trained students to

design PCBs but would not be fabricated [3], [4] or chose to build primitive circuit boards in-house [5]. There are courses that offer project-based PCB design experiences to senior students [6] or look into whether a PCB course can be made a requirement to benefit students [7]. There are a wide variety of courses that teach PCB designs with a special topic in wireless network [8], PCB antenna [9], power electronics [10], and many other topics, but these special topics associated with PCB designs are not intended for teaching fundamental circuits and electronics. Further, most PCB courses are offered to students in junior or senior year, but we see a high importance for students to learn practical aspects of electronic circuits in the sophomore year, and to retain the ECE students who feel a greater level of accomplishment in computer science courses.

In order to introduce students with circuits, electronics, and PCB design, we revised a 4-credit hour course to include PCB along with the following topics: diodes, bipolar junction transistors (BJTs), field-effect transistors (FETs), operational amplifiers/active filter designs, time-frequency domain analysis in a 15-week schedule. These lab experiments were revised for a teaching lab that excites and motivate today's students [1].

The following sections describe the enrollment and organization of the course along with valuable lessons learned from the course offerings.

Course Enrollment

The PCB lab were added as part of the electronics course for the first time during the fall 2018 semester at University of Wisconsin Stout. This 4-credit hour electronics course contains 6-hour combined lecture and lab meetings each week. Both ECE programs have been accredited by the ABET, and PCB design was included in the student learning outcomes of the course. There were 180 ECE students enrolled as of the writing of this paper. The full capacity of the electronics course was 22 students. In our program plan, the students are sophomore students taking the electronics class concurrently with the University Physics II course.

PCB Design Software

Altium Designer was the main software use in the class, due to their simple and intuitive graphical interface [11]. OrCAD PSpice was also adopted to allow the students to create their own schematics and run dc, ac, and transient analysis for their PCB designs.

Alternatively, PCB Artist, Easy EDA, and KiCAD are all free software that can be used. Other high-end PCB software used in industry include Cadence OrCAD PCB Designer and Autodesk EAGLE, both of which offer free student licenses for educational use.

PCB Fabrication

The primary goal was to create an environment that simulated an industry-like atmosphere where students could submit their own PCB design directly to a manufacturer. Thought we had initially proposed that each students create their own individual design for fabrication, we were constrained by the extra time needed to ensure that the design met the manufacturer's fabrication

requirements [12], [13]. To solve this, our approach was to define a single circuit problem by the instructor and require each student to submit a PCB design based on that specific problem. In order to include other educational aspects of PCB designs, students also submitted their schematics, PCB layouts, two-/three-dimensional images of their PCB design- all of which were created using Altium Designer. The instructor then evaluated students' work and provided feedback. At the end, each student received a revised and optimized PCB design from the instructor. That design was then used for production.

The advantage of this approach was to submit one optimized design for PCB production, and this turned out to be more time efficient and cost effective. Digi-Key Electronics was the third-party PCB vendor who provided the class with a convenient web interface to select an online PCB fabrication service [14]. Other online PCB fabrication services also provided the class with quick PCB delivery times [12], [13].

Method of Evaluation

There were thirteen lab exercises, homework assignments, exams, and project. The course grade was evaluated based on the following breakdown:

Weekly Lab Exercises: 50%
Homework Assignments: 5%
Exams: 20%
Project: 25%

Lab Exercises

The course goals were to cover circuits, electronics, and PCB design. The following lab exercises were outlined where students completed a lab exercise each week:

Lab 1: Reviewing the usage of oscilloscope, function generator, and multimeter

The lab involved with functional and signal integrity tests using probes for circuits on PCBs. Performing current and voltage measurements for a resistor-capacitor-inductor (RLC) circuit. Identifying the differences between root mean square (RMS) and instantaneous voltage from a multimeter or oscilloscope. Impedance analysis with a software tool used for PCB designs

Lab 2: Getting acquainted with diodes, ac and dc diode models, LED, Zener diode, reverse polarity protection circuit, and H-bridge motor driver circuit

Lab 3: Bipolar Junction Transistors (BJTs), Field Effect Transistor (FETs), ac and dc models of the transistors, and use of a BJT as a switch or a current amplifier

Lab 4: Use of BJTs to design transistor oscillators. Designing a H-bridge circuit to control the direction and speed of a dc motor

Lab 5: Getting familiar with operational amplifiers. Designing an inverting and non-inverting amplifier. Measuring DC and AC outputs. Plotting frequency response graphs

Lab 6: Designing active filters including a low-pass, high-pass, and band-pass filters

Lab 7: Getting started with the PCB design software (Altium Designer). Installing symbols and footprint libraries. Getting familiar with schematic capture and PCB layout

Lab 8: Creating circuits with schematic capture. Defining a board shape. Positioning components and footprints. Generating PCB layouts with auto routing tool. Reviewing design rules to ensure board design features fall within manufacturing capabilities. Understanding how to design PCB and generate gerber files and drill files for manufacturability

Lab 9: Re-designing the Arduino UNO Board to a specific board shape as shown in Fig. 1
The Lion board was built with electronic components different from the official Arduino UNO board but used the same Atmega328P microcontroller and bootloader. The bootloader can be programmed to the Lion board with the USBtinyISP programmer with Arduino IDE software [15], [16], [17]. More information about component selection, schematics, and layout created for the Lion board is available from the link: https://osf.io/fyjgn/
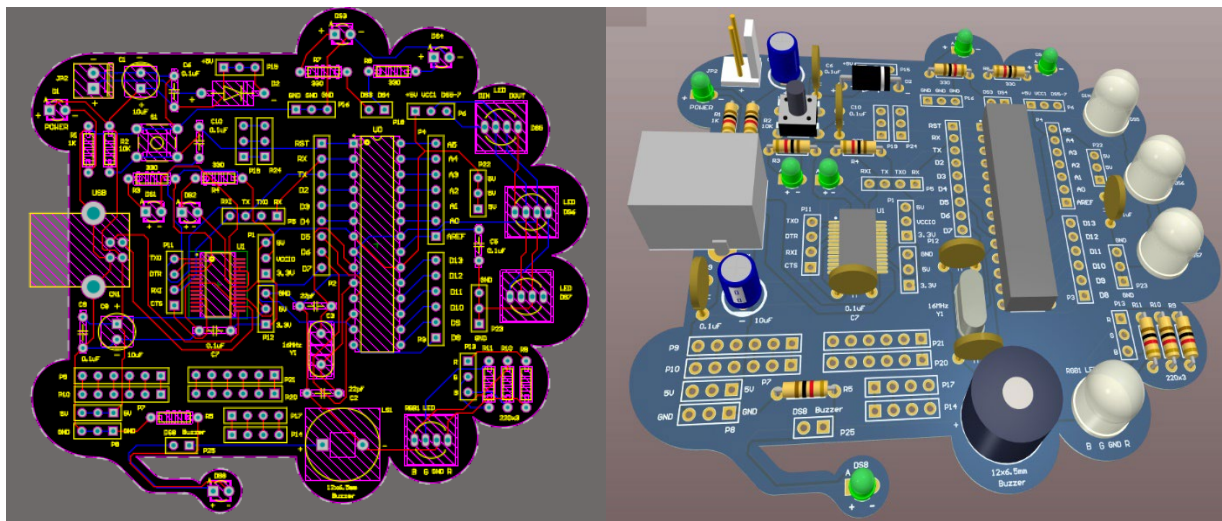

The Arduino UNO Board

Fig. 1 The Lion Board 2-D and 3D Views

Lab 10: Identifying through-hole and surface mount device (SMD) and soldering components onto the Lion PCB

The only SMD used on the Lion board was FTDI chip [18]. The FTDI chip is a USB to serial UART interface for the Lion board to provide in-system programming (ISP) to the microcontroller with a computer.

Lab 11: Testing the Lion board with a simple LED blinking program

Lab 12: Building a smart LED Bulb. Using block programming in the MIT App Inventor to create a phone app [19]. Changing color and luminance of a tri-color LED with a phone app. Designing slide bars on the phone app to control the brightness of a tri-color LED as shown in Fig. 2

Fig. 2 The Phone App Created with MIT App Inventor

Lab 13: Soldering wrap-up. Providing remaining steps to complete the soldering tasks

Project

After lab 13 was completed, students were introduced to the mobile robot (M-Bot) project. The project was for students to design a board shape to fit onto a ROMI chassis as shown in Fig. 3 [20]. The PCB for the M-Bot is shown in Fig. 4.
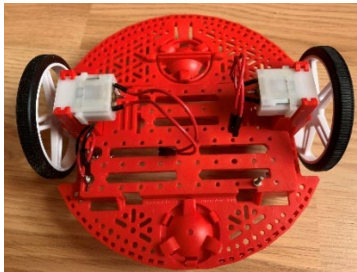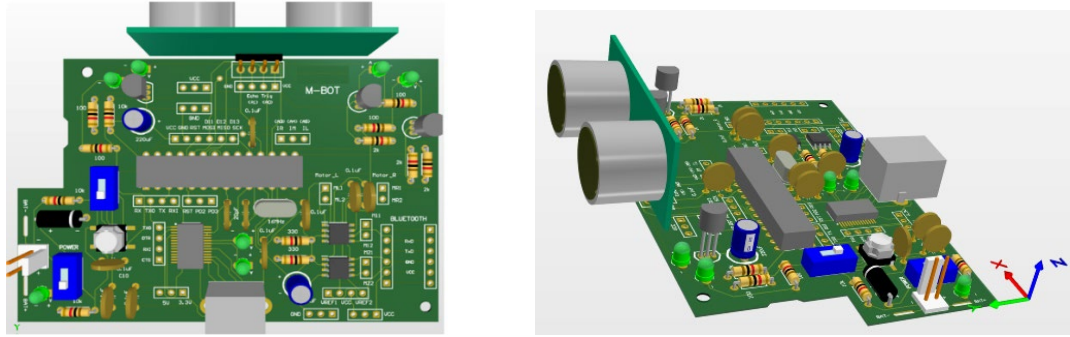

Fig. 3 M-Bot Chassis

Fig. 4 M-Bot PCB

The project required students to re-use the circuits on the Lion board and add motor control and collision avoidance circuits to the M-Bot PCB. One requirement was for students to select components suitable for motor control and collision avoidance with M-Bot. The project also required students to add two clear light emitting diodes (LED) and two yellow LEDs to the M-Bot as they were used as headlights and turn-signal lights in the driving mode. The headlights and turn-signal lights were marked with circles to illustrate their location on the PCB in Fig. 5.

Evaluation criteria for the project included factors such as circuit design, component symbol and footprint creation, as well as whether or not students designed workable circuits on their PCB. Once the instructor provided feedback to the students for their PCB design, a M-Bot PCB with an optimized design were provided to the students.
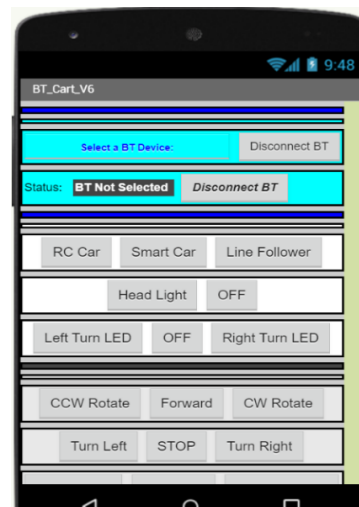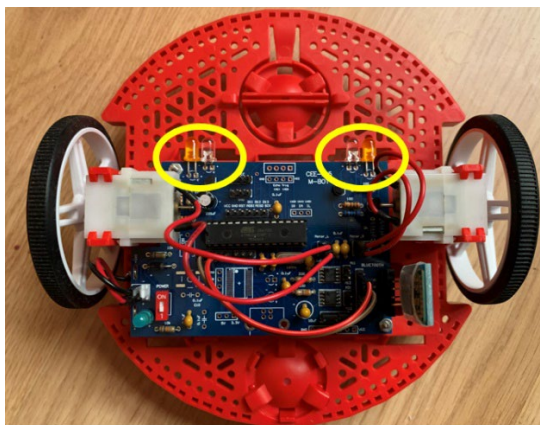

Fig. 5 Assembled PCB with M-Bot (left) and Phone App (right)

Another requirement of the project was for students to design a phone app to drive the M-Bot around. The phone app can trigger three modes: radio control, smart car, and line follower. The phone app was also used to control the headlights and turn-signal lights on the M-Bot. The turn signal blinker lit up automatically when the M-Bot made a left or right turn.

Soldering Equipment and Training

Our soldering stations were equipped with hot air reflow tools, safety glasses, solder fume extractors, solder paste, de-solder wicks, 0.8mm and 0.15mm diameter solder wires.

Prior to the first soldering activity, a safety training was provided to the students. Our focus was to design a two-layer PCB with through-hole and surface-mount components. Therefore, the students were given access to these components to practice their soldering skills.

After a soldering demonstration, the instructor assisted students in developing their soldering skills. Soldering the through-hole devices was straight forward for most students but soldering a surface mount device (SMD) was a challenge. There were SMD with two different package types available to the class. They were Quad Flat Package (QFP) and Quad Flat No-leads package (QFN). The soldering training was focused on the Quad Flat Package (QFP) because it was selected for the Lion board and M-Bot PCB.
Based on our experience, a QFP can be hand soldered with ease using an ultra-thin 0.15mm diameter solder wire. A sample soldering work for a QFP is illustrated in Fig. 6. Although a Quad Flat No-leads (QFN) device is similar to a QFP, the leads of a QFN were not extended out to the package side, so it was difficult for students to hand-solder a QFN with a hot air reflow tool.
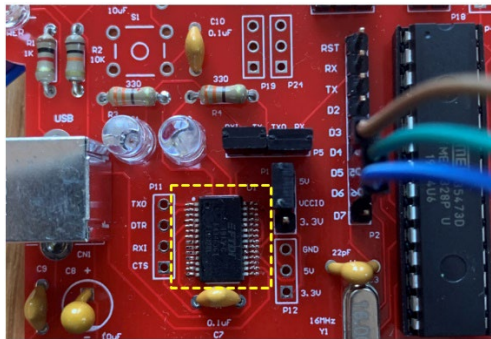

Fig. 6 Quad Flat Package Soldered on the Lion Board

Several components were soldered by the students during the training, including through-hole components and a QFP. One of the common soldering problems for the students was the amount of solder to use for a solder joint. Some students didn't use enough solder for a through-hole connection and had a connection with a gap without solder. Other students applied to much solder that resulted in solder bridges and required the use of a de-soldering wick to remove excessive solder.

Although a few students had soldered experiences, most students indicated that the training opportunity was very helpful as they can learn about the soldering equipment and practice their soldering skills.

Course Assessment

A course assessment was conducted in the Fall 2019 and Fall 2020 respectively. In the student surveys, one student expressed " we learned a large variety of cool labs with PCB design."; One student expressed " tests anxiety was highest in the course because of the style of learning. This probably has less to do with the instructor and more to do with the style of learning with circuits." One student expressed " the exams were a bit difficult to prepare for." which we interpreted this as requests for providing guidance to prepare for tests/exams. One student expressed " the lab was both fun and challenging, I really learned a lot and all of it can be directly applied to industry."; One student expressed " These experiments have the students understand the outcome and see the piece of technology in action. This is valuable experience considering we are here to work in the outside world of industry.

A sample student report for the M-Bot project is available from the link: https://osf.io/zq78f/

Conclusions

The course demonstrates the possibility of offering an electronics course with PCB designs where students learn circuits, design, build, and test PCBs in both hardware and software aspects. The quality of the PCBs that students produced was encouraging indication that students were greatly motivated. Further, the PCB design experience appeared to reduce a common dissatisfaction of students with a circuit course. From the student surveys, the students seemed to enjoy learning circuits and electronics through the lab exercises and PCB project, but not for tests or exams. The deficiency would be insufficient reviews to assist the students to prepare for tests/exams. This can be improved by managing the laboratory time and tests more efficiently. The course is offered in 2022 and another student survey results will be evaluated as a part of the continuous improvement of the course.

Reference:

[1] Y. Tsividis, "Turning Students On to Circuits," in IEEE Circuits and Systems Magazine, vol. 9, no. 1, pp. 58-63, First Quarter 2009.

[2] B. E. Boser, "A first course in electronics," 2012 IEEE International Symposium on Circuits and Systems (ISCAS), 2012, pp. 2929-2932.

[3] Kim, E.M., Schubert, T.F., "A low-cost design experience for junior-level electronics circuits laboratories through emulation of industry-printed circuit board design practice," *International Journal of Electrical Engineering & Education*, Vol. 53, Issue 3, 2017, pp. 208-22.

[4] J. Song, K. E. Hoover and E. Wheeler, "Effectiveness of PCB Simulation in Teaching High-Speed Digital Design," 2007 IEEE International Symposium on Electromagnetic Compatibility, 2007.

[5] Taylor, R. A., Jackson, D. J., "A Custom-PCB Design for Microcontroller Education", Proceedings of the 2012 ASEE Annual Conference and Exposition, San Antonio, TX, 2012.

[6] Karl Brakora, "A Project-Based Printed Circuit Board (PCB) Electronics Course", Proceedings of the 2020 ASEE Annual Conference and Exposition, Virtual Conference, 2020.

[7] Pelin Kurtay, "Development of a Printed Circuit Board Design Laboratory Course", Proceedings of the 2020 ASEE Annual Conference and Exposition, Virtual Conference, 2020.

[8] G. Isaakidis, A. Spiropoulos and M. Drakaki, "A wireless three-dimensional printer for printed circuit board applications," 2017 Panhellenic Conference on Electronics and Telecommunications (PACET), 2017.

[9] W. Ren, "Design of Compact Monopole PCB Antenna for UWB Applications," 2009 International Conference on Wireless Networks and Information Systems, 2009.

[10] F. Hou et al., "High Power-Density 3D Integrated Power Supply Module Based on Panel-Level PCB Embedded Technology," 2018 IEEE 68th Electronic Components and Technology Conference (ECTC), 2018.

[11] Altium, La Jolla, California. Available at https://www.altium.com/solutions/academic-programs/student-licenses (Accessed: Jan. 19, 2022)

[12] JLPCB PCB Capabilities. Available at https://jlcpcb.com/capabilities/Capabilities (Accessed: Jan. 19, 2022)

[13] Advanced Circuits PCB, Maple Grove, Minnesota. Available at https://www.4pcb.com/blog/manufacturing-capabilities-for-custom-pcb-requirements/ (Accessed: Jan. 19, 2022)

[14] Digi-Key, Thief River Falls, Minnesota. Available at https://www.digikey.com/en/pcb-builder (Accessed: Jan. 19, 2022)

[15] Arduino Development Board and Software. Available at https://www.arduino.cc/ (Accessed: Jan. 19, 2022)

[16] Arduino bootloader. Available at https://docs.arduino.cc/hacking/software/Bootloader (Accessed: Jan. 19, 2022)

[17] USBtinyISP programmer. Available at http://www.ladyada.net/make/usbtinyisp/ (Accessed: Jan. 19, 2022)

[18] FTDI Chip. Available at https://ftdichip.com/wp-content/uploads/2020/08/DS_FT232R.pdf (Accessed: Jan. 19, 2022)

[19] MIT App Inventor. Available at http://appinventor.mit.edu/ (Accessed: Jan. 19, 2022)

[20] ROMI Chassis. Available at https://www.pololu.com/category/202/romi-chassis-and-accessories (Accessed: Jan. 19, 2022)