2026 ASEE ANNUAL CONFERENCE June 21–24, 2026 | Charlotte Convention Center, NC
Host Sponsor UNC Charlotte
Paper ID #53422

PCB Design Education: Integrating Modern Microcontroller Platforms in Senior-Level ECE Curriculum to Improve Capstone Success

Jason Jung, University of Illinois at Urbana - Champaign
Michael Gamota, University of Illinois at Urbana - Champaign
Megan Zeng, University of Illinois at Urbana - Champaign
Arne Woolsey Fliflet, University of Illinois at Urbana - Champaign

PCB Design Education: Integrating Modern Microcontroller Platforms in Senior-Level ECE Curriculum to Improve Capstone Success

Abstract

The opportunity to design a custom printed circuit board (PCB) in a senior-level course is an impactful experience for electrical and computer engineering (ECE) students to reinforce concepts in circuit theory and system design. However, ECE students have a wide variety of experience levels with PCB design and assembly before beginning the course. Furthermore, the learning curve of electronics may reduce what can be accomplished in the course. To address this, students follow a tutorial for the design and assembly of a microcontroller-unit (MCU) breakout board at the start of the course. In this work, we detail the improvement of that introductory assignment in a senior-level course to teach essential and industry-relevant PCB design skills. Furthermore, we evaluate the effectiveness of this assignment in aiding student development of PCB skills and success in their capstone projects.

We have developed two separate PCBs for the improvement of the assignment. Both of them incorporate a more capable and modern MCU, resulting in different considerations for schematic design, PCB layout, soldering techniques, component selection, and programming. The new tutorials cover the design and assembly of a development board based on a choice of an ESP32-C3 or STM32F103 MCU [1, 2]. Students are asked to demonstrate the functionality of a circuit that does motor control, wireless communication, or sensor data reading. This provides students the opportunity to develop an understanding of basic circuits that can be leveraged to prototype their projects for applications such as Internet of Things (IoT), robotics, and wireless/radio frequency (RF).

Introduction

Students majoring in Electrical Engineering are required to take the ECE Department's Senior Design Project Laboratory one-semester capstone course in which students form 3-person teams to carry out an electronics project. The course is optional for Computer Engineering majors, but many take it. Although the objective of the projects varies widely, they all include an MCU-based circuit implemented on a PCB. During the 15-week course students learn to propose and receive approval for a project that solves a problem subject to time and cost constraints, develop a design and schedule, work together as a team to construct a hardware device, and demonstrate a working device at the end of the course. The proposal, approval, and design phase of the project occupies the first six weeks of the course. In the remaining nine weeks of the course, students acquire circuit components, test breadboard versions of circuits, write programs for the MCUs, work with the department machine shop to fabricate any mechanical designs, and receive their custom-PCBs to assemble and test. In a successful demonstration, the device satisfies high level requirements and its subsystems satisfy requirements laid out in the project design document. A significant part of the demonstration score is associated with having a successful PCB; however, many teams have difficulty achieving this. Students' prior experience with PCB design and soldering varies widely, so a PCB design and soldering assignment is included during the first weeks of the course. Student are required to complete a tutorial that guides them through schematic capture, PCB layout, soldering, and programming of a breakout board for the ATTiny85 MCU [3]. Our effort to improve this assignment to better prepare the students for a successful PCB outcome is the subject of this paper.

It is common for students to pursue projects that require significant computational power or wireless capability. However, the MCUs that are able to meet such demands require significantly different design considerations than that of the ATTiny85. To address this, we introduce two updated assignments that are designed around two different high performance, industry relevant MCUs. Students are given a choice of which design to complete for this assignment. Students are required to perform schematic capture for their selected MCU and its necessary peripherals, PCB layout, soldering, and firmware programming. In this work, we will show the design and assignment details we are attempting to implement into our capstone course curriculum. We discuss the survey method we used to evaluate the effectiveness of the assignment in assisting students in their capstone projects as well as teaching fundamental PCB concepts. Finally, the results and analysis of this survey are presented.

Assignment Description

The introductory project is split into two assignments: (1) PCB design and (2) assembly and demonstration. In the first two weeks of the course, students are in the ideation phase of their capstone projects. During this time, students follow a tutorial using an electronic design automation (EDA) tool to design a board based on their choice of MCU : the STM32F103 or the ESP32-C3 MCU. The latter MCU includes a RF front end for Bluetooth and WiFi.

Upon completion of their PCB design assignment, students are given the following two weeks to assemble and program tutorial boards using the university's senior design lab resources. After assembly, they are instructed to configure, write, and flash a program onto their MCU. ST Microelectronics has their own programming ecosystem: CUBE integrated development environment (IDE) [4]. The ESP32-C3 can be programmed using the Arduino IDE [5]. Students complete this assignment once they have made a successful demonstration of their choice of motor control, WiFi capability, or sensor data reading.

Providing two MCU choices allows students to make decisions based on their capstone needs. The ESP32-C3 allows wireless capability; however, the STM32F103 consumes less power and offers more general purpose inputs and outputs (GPIO). Regardless of choice, both boards were designed with the same goals of teaching fundamental PCB design concepts and serving as a development board that students can use to prototype their projects.

Schematic Design

The first part of the assignment guides students to perform schematic capture for their PCBs. Students use KiCAD, an open-source electronic design automation (EDA) tool, for both schematic and layout design. Since the focal point of these PCBs is the MCU, students will be referred to their respective MCU datasheet [6]. Instructions will be provided to distill the datasheet and guide the students in developing core circuit designs so that their MCU can be powered and programmed. As such, all schematics include: USB-C programming circuits, reset buttons for MCU boot-mode configurations, a linear regulator circuit for power supply, and crystal oscillator clock sources.

Both designs include the circuit to operate an L293 motor driver and GPIO fanout for arbitrary usage [7]. Students interested in robotics can use the motor driver, and GPIO can be configured for various digital communication protocols used to interface with sensors, other devices, etc.

Due to its wireless capability, ESP32 schematic design is suited for students interested in Internet of Things (IoT) and wireless/radio frequency (RF) projects. The ESP32's low noise amplifier (LNA) input is connected via a π-impedance matching network to an external ceramic chip 2.4GHz antenna. Students can use this circuit for WiFi or Bluetooth connection.

For students interested in RF projects, the antenna circuit provides the flexibility for students to connect to an external RF front-end instead of the ceramic chip antenna. As shown in Figure 1, students would use AE1 instead of AE2 for their own RF design. AE1 is selected to be a U.FL RF connector. The length of the RF network is short relative its design frequency, so a lumped assumption is made. Students that utilize this option are able to design their own RF components such as filters, power dividers, and antennas at a frequency of 2.4 GHz.

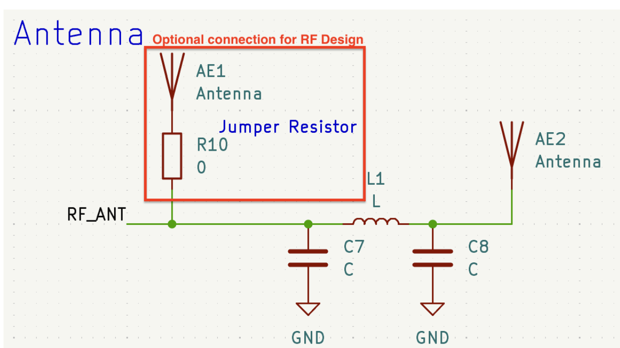
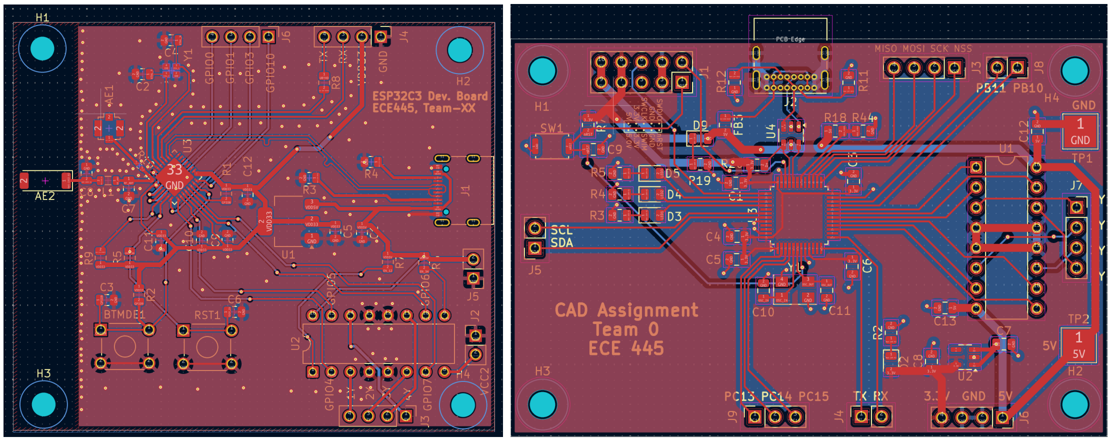


Figure 1: ESP32 schematic, RF circuit

PCB Layout Design

After schematic capture, students are required to do PCB layout. An example of the completed layout for both boards are shown in Figure 2. The boards are 2-layer designs with 1 oz copper thickness.



(a) (b)

Figure 2: (a) ESP32 PCB layout (b) STM32 PCB layout

The following layout practices and skills were taught in this portion assignment:

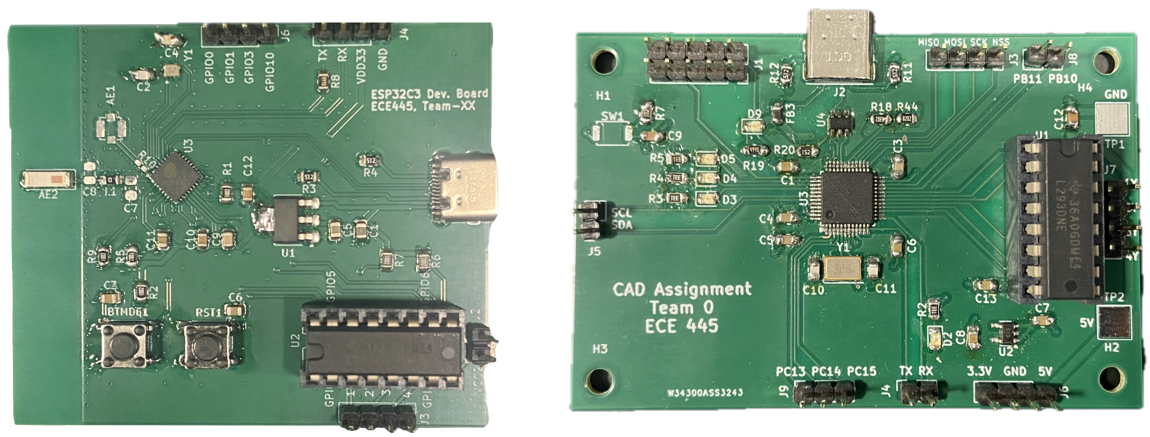
- Component placement with soldering and routing in mind.
- Calculating appropriate trace widths
- Best practices for grounding and via placement
- Routing practices for signal, power, and differential pairs.

Soldering and Assembly

After schematic capture and layout, students are required to assemble their PCBs. To accommodate a wide variety of soldering comfort among students, a tutorial of the following soldering techniques is included and demonstrates:

- Soldering with an iron
- Stencil, solder paste, and a reflow oven
- Hot air gun soldering
- Desoldering techniques using hot air or using wick.

The instructors guide the students through the use of a stencil, solder paste, and a reflow oven to assemble their boards. Students may use the other soldering techniques as needed. A microscope station is provided for students to perform visual inspection. After successful assembly, students will have PCBs that mirror those shown in Figure 3.



(a) (b)

Figure 3: (a) ESP32 PCB assembled (b) STM32 PCB assembled

PCB Programming and Demonstration

The final portion of the assignment requires students to program and demonstrate their PCBs. Students are given options on what to demonstrate: using the 2.4GHz antenna on the ESP32-C3 board to set up a WiFi access point, control a unipolar stepper motor using the L293 motor driver, or using GPIO on either of the boards to interface with an external sensor. An external accelerometer module and a stepper motor are provided for students demonstrations.

Instructions for environment setup in the Cube IDE or Arduino IDE for firmware programming are provided for both the STM32 and ESP32 MCU respectively. Furthermore, working demonstration code is provided to the students so that they can focus diagnosing hardware-related issues in their debugging.

Survey Methods

A survey was offered to all students in the course (N=196) 10 weeks into the semester to evaluate the impact of the new assignments on student learning outcomes and capstone success. Students were awarded extra credit for their participation, and the collected responses reflect a response rate of 74% for the entire course. The survey questions evaluate the impact of the assignments on students' PCB skill development, the assignment pertinence to industry-relevant PCB skills, the effectiveness of the assignment in aiding their capstone development process, and student confidence in delivering a successful and fully-functional capstone project.

While both assignments were tested by an independent reviewer with beginner-level experience, only the ESP32-C3 PCB design and soldering assignments were offered during the testing semester to a small batch of randomly selected students (N=25). This was due to the limit on hardware resources available to the course. The tutorial documents created for the introductory assignment were made available to the entire course to serve as a design reference.

Collected responses are divided into two groups based on whether students completed the newly developed assignment (i.e. the pilot assignment) or the current assignment. Both introductory exercises were administered during the semester to support the transition of assignments. The responses are further partitioned into beginner, novice, and advanced categories based on a respondent's prior experience with PCB projects.

To assess student PCB skill development, Likert-scale questions were administered, covering 5 categories: 1) design skills, 2) familiarity with EDA tools, 3) common design considerations for a particular circuit, 4) design of a PCB around a certain MCU platform, and 5) assembly and test. Examples of the types of questions asked are shown in Table 1.

Subcategory	**Example**
Design Skills	"Creating a PCB schematic"
EDA Tool Familiarity	"Importing a footprint"
Design Considerations	"Impedance matching"
MCU Platform Selection	"Designing a PCB based on an ESP family microcontroller"
Assembly and Test	"Hand soldering SMD and THT components"

Table 1: Prompt: "Rate your confidence in the following PCB-related tasks. The rating scale is from 0–4."

The students are then asked if the skills learned in the assignments have helped them in any hardware design interviews. This is used as a proxy to determine if the assignment was valuable for their industry endeavors. Students are prompted to provide details.

To estimate the effectiveness of the assignment, we ask students how the difficulty of completing their project would change if they did not do the assignment and to provide justification.

Finally, to evaluate capstone success, students were asked about the design choices they made for their projects as well as their confidence in delivering a fully-functioning project by the end of the course. We subsequently ask for an explanation.

Results

PCB-Related Skills

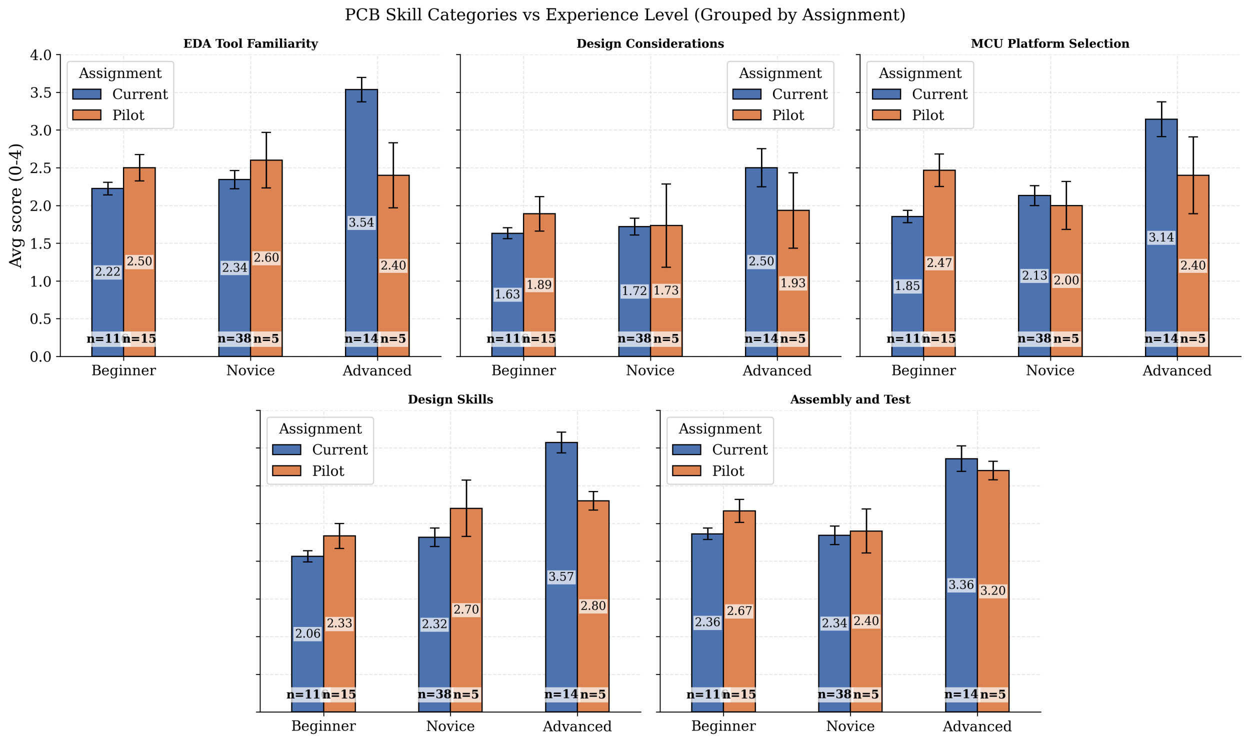


Figure 4: CAD and soldering skill levels after assignment completion

The results of the Likert-scale questions that evaluate skill retention and student learning are shown in Figure 4. Across all measured skill categories, beginner students who completed the pilot assignment report greater confidence in their PCB-related skills than beginner students who completed the current assignment.

Since the assignment was catered to beginners, we are most concerned with comparing beginners from each assignment group. We expect novice and especially advanced students to find less utility in these assignments, particularly if they are already familiar with the tools and PCB design in general.

Interview Impact

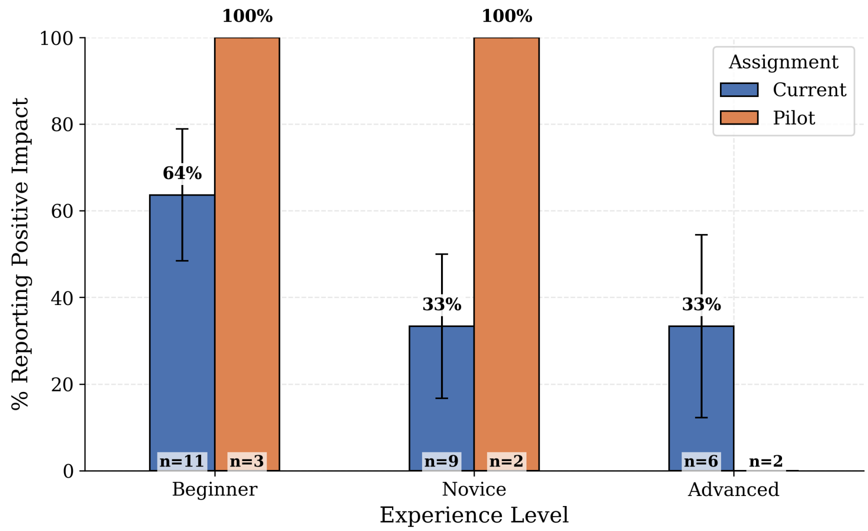


Figure 5: Distribution of responses to a question asking if the PCB assignment had a positive impact on an interview

To assess the value of the assignment experience on industry-relevant PCB skills, the survey participants were asked if the skills learned throughout the assignment had helped them during an interview process. Figure 5 shows the distribution of responses based on student experience-level. All of the beginners and novices who completed the pilot assignment and subsequently had a PCB design related interview report that the assignment helped them in the interview.

Students were asked to specify which skills that they learned were directly useful in their interview process. Sample responses are shared.

- *"Understanding what islands, vias, ground planes, silkscreen, reasoning for trace width, etc were useful in one-off questions about some simple PCB questions."*
- *"I recently completed a second interview for a PCB design/debugging position. The assignment helped me get an introduction to PCB design which was helpful"*
- *"In interviews, I have been asked about the importance of placement (to prevent EMI/EMC interference), schematic design, and other power component design decisions that I have directly worked on in this class with the KiCAD assignment."*

Many of the theoretical bases for PCB design practices are learned by students in courses (ground planes, impedance matching, EMI, etc.), but completing a PCB design assignment provides a tangible experience that can be called upon in an interview.

Perceived Difficulty of Project

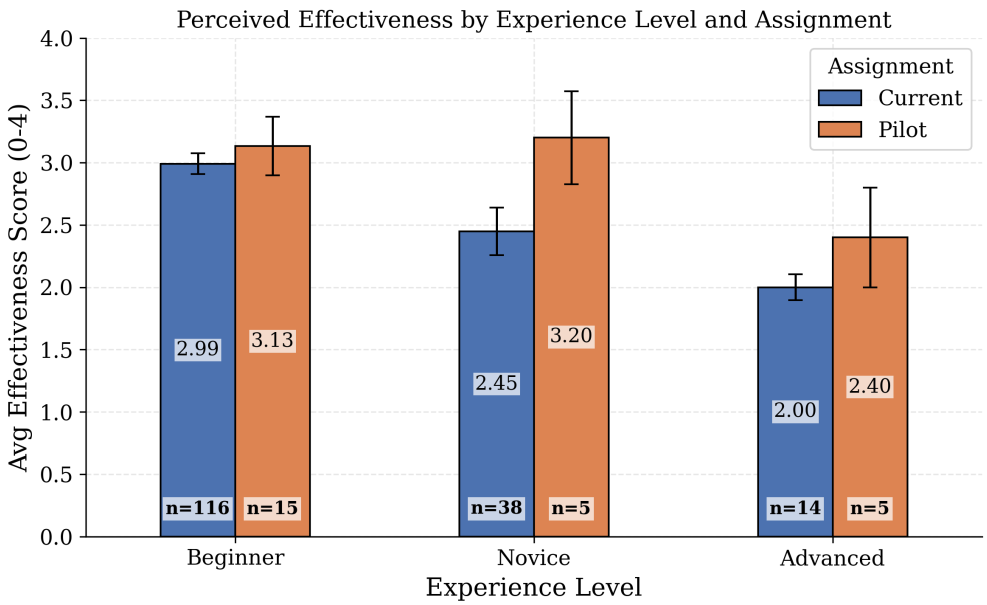


Figure 6: Student responses to a question evaluating assignment effectiveness. Higher score indicates the assignment was more helpful.

To further evaluate learning outcomes, student were asked the impact on the assignments on their perceived difficulty in their senior design project. In a Likert-scale question, they were asked if the capstone project would be more or less difficult to complete without having done the assignment. A higher number means it would be more difficult without the assignment. As shown in Figure 6, requiring students to complete either assignment appears to be most effective among beginners and novices.

Responses from the beginners group often cited specific skills that directly benefited their project.

- *"I have never soldered or really designed a PCB, and my team and I referred back to the soldering assignment page when doing our own PCBs many times for guidance."*
- *"Knowing how to use KiCAD was a significant help, while the soldering was a nice review for me"*
- *"The pilot assignment didn't go well in the sense my PCB didn't work so it helped me know what to look for in my senior design project when things didn't work"*

A common response that advanced students gave was that both the current and new assignments were too simple based on their previous experiences.

- *"I didn't get too much out of the assignment, I already have a lot of experience with solder-*

ing and PCB design"

- *"I was able to reuse my intuition for soldering/creating circuit for my esp-c3, but ended up consulting the datasheets/Espressif manuals/other past projects of mine anyways"*

Regardless of the assignment, having an introductory exercise has shown to be valuable in providing beginner-level students the requisite skills to succeed in the course.

Success in Senior Design Projects

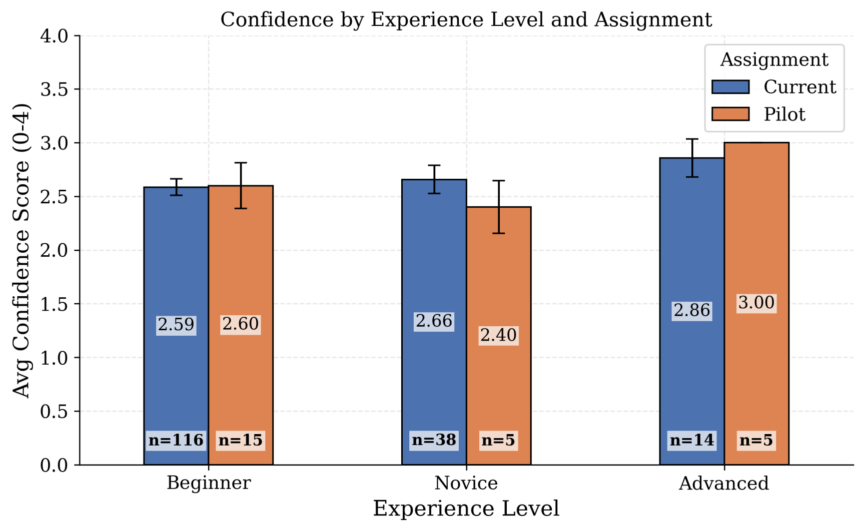


Figure 7: Confidence in project functionality by skill level

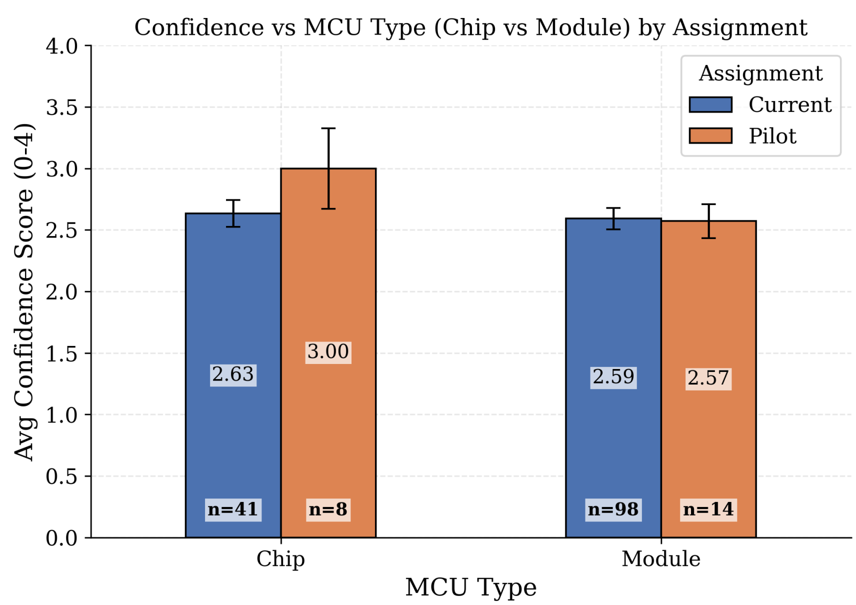


Figure 8: Confidence in project functionality by MCU type

Students were also asked to give a level of confidence that their senior design project would be fully functional. Figure 7 shows that the new assignment did not have a significant impact on confidence in project functionality among beginner students.

Nearly 75% of survey respondents opted to use an ESP32 family MCU for their senior design projects. We believe that the wireless capability commonly offered by the ESP-family of MCUs is the main reason why. Furthermore, 65% of respondents indicated that they used an ESP32 MCU module (e.g. ESP32-S3-WROOM). A module integrates multiple components such as the crystal oscillator and WiFi capability into a single package.

Despite the fact that the introductory pilot assignment guides the students through designing a PCB based around an MCU using discrete parts, MCU modules saw a higher usage rate. Two potential reasons for this is the perceived lower risk of designing and assembling fewer components for their projects.

We then explored the relation of MCU choice to confidence in project success. As shown in Figure 8, we do observe a higher average confidence among students that used a discrete MCU chip and completed the pilot assignment compared to students who completed the current assignment.

Students were then asked to justify their answer to this question. When comparing the students between the beginner and advanced groups, similar themes emerged. Regardless of choice, several beginners noted their inexperience as a major reason to their uncertainty in believing that their final project would work. Some examples include:

- *"PCB design seems like it would work after doing some breadboard testing and looking at application circuits, but I still have low confidence because of my inexperience"*
- *"Due to my lack of exposure to PCB design before this class, I'm unsure if all the connections in the PCB design process are correct and will function for the scope of our project"*
- *"I am still worried there will be some errors that were missed that will prevent our PCB from working. This is also my first PCB design so I am not very confident in my debugging abilities."*

A substantial amount of experienced students would cite technical reasons or relay their previous experience as a major reason why their senior design projects would work.

- *"We passed all ERC and DRC, we passed DFM checks, and we do not have any majorly challenging signals for which we need to be concerned about reflectance and high speed clocks"*
- *"From our breadboarding demos I have a similar schematic if not very similar by using dev boards"*

Based on these short-answer responses, students may still feel uncertain about their designs even after completing the assignment because of the guided nature of the assignment. The uncertainty of making a *custom* PCB cannot be overcome by a tutorial.

Examples of Integration of MCUs into Senior Design Projects

To illustrate the connection between the introductory assignment and the integration of an industry-relevant MCU into students' senior design projects, the following are examples of senior design projects:

Motorized satellite tracker

A motorized satellite tracker, which uses an RF down-converter front-end that down-converts receiving signals at S-band frequencies (2.2 GHz) to a frequency that is within the bandwidth of an affordable software-defined radio (SDR) module. With this down-converter, students send control signals to their azimuthal and elevation motors to direct their antenna to an optimal position to capture S-band signals from a moving satellite.

In this project, the students designed multiple PCBs. Their PCBs serve as a motor controller using the ESP32-S3-WROOM. They also designed RF PCBs for their low-noise amplifier, mixer, and intermediate frequency filter. Their PCBs are as shown in Figure 9(a). The assembled down-converter is shown in Figure 9(b); as shown, the students also created a helicon antenna to capture signals.

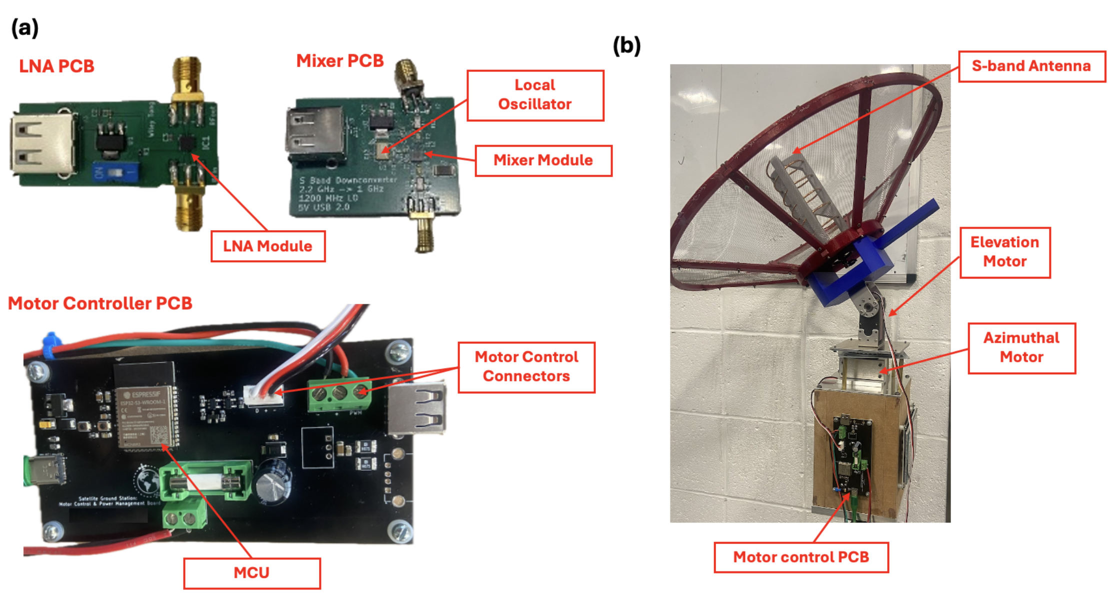


Figure 9: RF downconverter project (a) The LNA, mixer, and motor controller PCB that the students designed. (b) The assembled satellite-tracker. The motors, motor control PCB, and antenna are shown. The RF down-converter electronics are housed on the backside of the assembly.

Ant-weight Battlebot

The senior design course hosts ant-weight battlebot competition for teams that are interested in robotics for their capstone project. Each battlebot must weigh under 2 lb. As an example of a battlebot, one team built a battlebot that has plastic wheels for mobility and a lifting wedge as a weapon. To design their PCB, the team chose the ESP32-S3-WROOM which provides Bluetooth connectivity for wireless control, interfaces with an H-bridge motor controller, and provides pulse-width modulation (PWM) signals to a servo motor for its weapon. They also implemented a current sensor to protect against stall or over-current conditions. Their PCB is shown in Figure 10(a). To enclose their PCB, the team 3D-printed a frame to house their PCB, battery, and wire connections. Their final robot is shown in Figure 10(b).

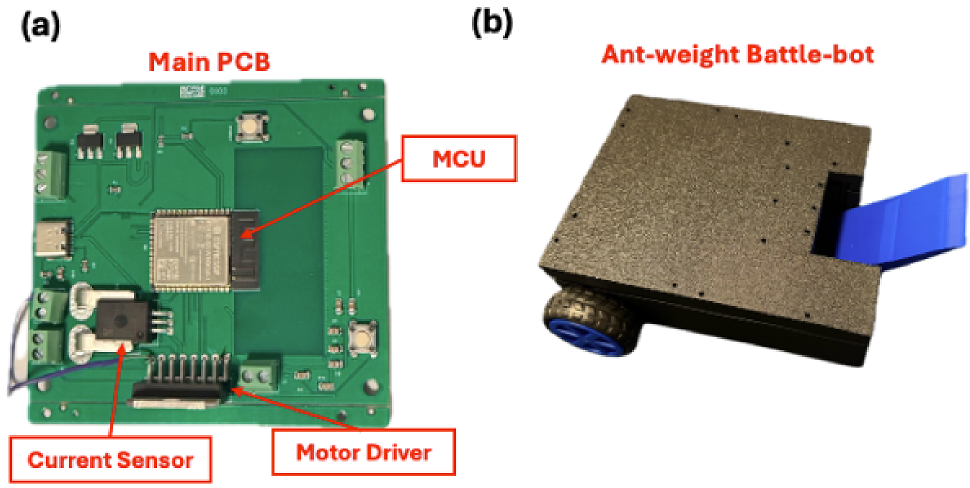


Figure 10: Example of a battlebot project (a) PCB for the student team's battle bot. Includes (b) Enclosure of the battle bot. The blue wedge is robot's wedge weapon that is controlled by a servo motor.

Colony Health Monitor

As an example of an IoT project, one senior design team has partnered with a research group in the environmental sciences department of the university to create a bee colony health monitor. Their main objective is to create a device that provides real time readings of humidity, temperature, and carbon-dioxide levels inside a colony. In this way, colony health can be assessed without physically disturbing it. Figure 11(a) shows the students deploying their device into a colony environment. Their PCB is shown in Figure 11(b). Here, the students use the ESP32-C3-WROOM to communicate with sensors placed inside the colony through a wire connection. Data is sent to an external data display to provide colony status and can also be transmitted via WiFi to a centralized data panel.

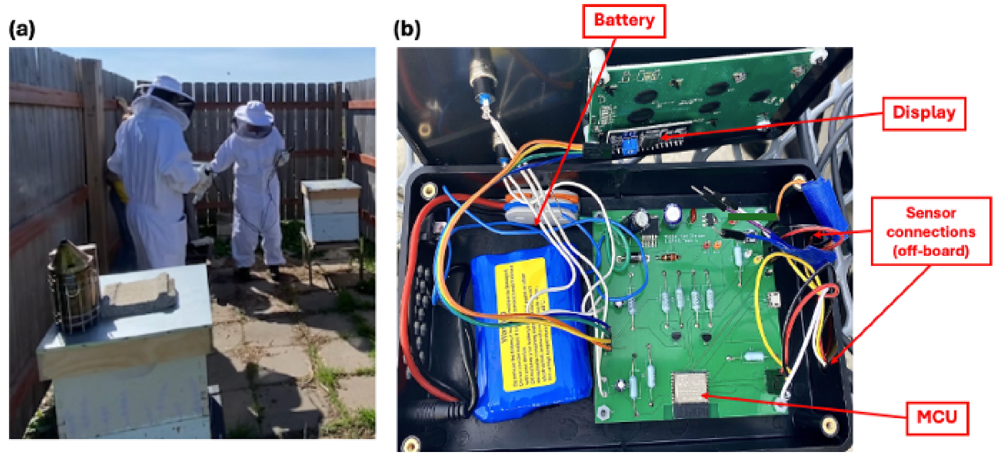


Figure 11: Bee colony health monitor project. (a) Students implementing their monitor into a bee colony. (b) Under the hood of their device. Includes their PCB design, power supply, monitor, and sensor feeds.

Conclusion

To conclude, we detail our efforts to improve an introductory PCB and soldering assignment for a capstone course. We designed two new assignments to replace the current assignment. The new assignments are tutorial in nature and provide students with a guide for PCB schematic capture, layout, assembly, programming, and test. One of the two new assignments was administered in conjunction with the current assignment. The PCB is intended to be used by students as a development board for prototyping their capstone projects.

Our findings show that students report confidence in the skills learned in the new assignments that are directly applicable to their capstone projects. Furthermore, the experience that the assignment provides is valuable for students in their industry endeavors. Students indicated that the skills learned were helpful for their capstone projects. However, our results suggest that beginner students' uncertainty in project success cannot be overcome by a guided assignment.

Future work could address this success uncertainty gap between beginner and experienced students. One possible option is to explore the development of a component selection exercise so that students retain the freedom to design while minimizing the risk of an unsuccessful project.

Ultimately, the success of projects across varied targeted applications validates the pilot assignment's role ability as pedagogical springboard. Introducing the STM32 and ESP32 early in the coursework provides students with a technical baseline to support their capstone projects.

References

[1] *Esp32-c3 series datasheet*, Version 2.1, Espressif Systems, 2024.
[2] *Stm32f103x8/b medium-density performance line arm-based 32-bit mcu*, Rev 17, STMicroelectronics, 2015.
[3] *Attiny25/45/85 datasheet*, DS40001904C, Microchip Technology, 2022.
[4] *Stm32cubeide user guide*, UM2609, STMicroelectronics, 2023.
[5] Arduino SA, *Arduino ide*, version 2.3, 2024.
[6] KiCad Developers Team, *Kicad eda*, version 8.0, 2024.
[7] *L293x quadruple half-h drivers*, Rev. D, Datasheet SLRS008, Texas Instruments, Dallas, TX, USA, 2016.