A LEARNING MODULE TO DESIGN PRINTED CIRCUIT BOARDS FOR IOT-BASED DEVICES

MÓDULO DE APRENDIZAJE PARA DISEÑAR PLACAS DE CIRCUITO IMPRESO PARA DISPOSITIVOS BASADOS EN EL IOT

Fábio AMARAL ⓘ, Eberton MATIA ⓘ, Alexandre OLIVEIRA JÚNIOR ⓘ,
Matheus ZORAWSKI ⓘ, Jonas QUEIROZ [1] ⓘ and Paulo LEITÃO ⓘ

Research Centre in Digitalization and Intelligent Robotics (CeDRI), Instituto Politécnico de Bragança, Campus de Santa Apolónia, 5300-253 Bragança, Portugal
{alexandrejunior, matheuszorawski, jpqueiroz, pleitao}@ipb.pt

ABSTRACT: Printed circuit boards (PCBs) are essential for developing Internet of Things (IoT) devices. The proper design of these boards is vital to ensure that they comply with industrial standards. Despite the high demand for qualified professionals to develop this kind of PCBs, there is still a considerable gap between the skills taught to students during their academic journeys and the industry's needs. In general, many skills in design rules and technical aspects necessary to assure the proper development and performance of such devices are acquired by engineers based on experience in practical projects. Having this in mind, this paper presents a learning module that focuses on practical aspects and concerns regarding the design rules of PCBs. In this module, students follow a hands-on activity to design an IoT-based device PCB, employing a microcontroller to read one accelerometer data and communicate via Bluetooth. During the project development, the instructors discuss and share practical experience in various electrical aspects of the PCB components layout that can affect the circuit's performance. The attendees' positive feedback and performance during the practical activities demonstrate the effectiveness of the learning module.

KEYWORDS: smart devices design; IoT devices; printed circuit board; education 4.0.

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication Technologies for Innovation and Digital Transformation, pp. 13-29

[13]

F. Amaral, E. Matia, A. Oliveira Júnior, M. Zorawski, J. Queiroz, P. Leitão
A Learning Module to Design Printed Circuit Boards for IoT-based Devices

RESUMEN: Las placas de circuito impreso (PCB) son esenciales para el desarrollo de dispositivos del Internet de las Cosas (IoT). El diseño adecuado de estas placas es vital para garantizar que cumplen con las normas industriales. A pesar de la gran demanda de profesionales cualificados para desarrollar este tipo de PCB, sigue existiendo una brecha considerable entre los conocimientos que se enseñan a los estudiantes durante su trayectoria académica y las necesidades de la industria. En general, muchos de los conocimientos sobre las normas de diseño y los aspectos técnicos necesarios para garantizar el correcto desarrollo y rendimiento de este tipo de dispositivos son adquiridos por los ingenieros a partir de la experiencia en proyectos prácticos. Teniendo esto en cuenta, este trabajo presenta un módulo de aprendizaje que se centra en los aspectos prácticos y en las preocupaciones relativas a las reglas de diseño de las placas de circuito impreso. En este módulo, los estudiantes siguen una actividad práctica para diseñar una PCB de un dispositivo basado en el IoT, empleando un microcontrolador para leer los datos de un acelerómetro y comunicarse vía Bluetooth. Durante el desarrollo del proyecto, los instructores discuten y comparten la experiencia práctica en varios aspectos eléctricos de la disposición de los componentes del PCB que pueden afectar al rendimiento del circuito. Los comentarios positivos de los asistentes y su rendimiento durante las actividades prácticas demuestran la eficacia del módulo de aprendizaje.

PALABRAS CLAVE: dispositivos electrónicos inteligentes; dispositivos IoT; circuito impreso; educación 4.0.

1   Introduction

Smart devices based on Internet of Things (IoT) technologies are found in a broad spectrum of applications, such as product supply chain management in industry, pest and disease monitoring in agriculture, energy consumption and transmission management in smart grids, wearable devices for healthcare, among others. However, despite the countless benefits to these fields, the wide diffusion of IoT devices also brings challenges that encompass energy efficiency, cybersecurity, interoperability between systems, and hardware design [1]. Regarding the challenges in hardware design, several development platforms mainly based on microcontrollers or embedded systems with wireless

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[14]

communication capabilities (e.g., development boards like Arduino, ESP32, Raspberry Pi, Adafruit, etc.) are being employed as enabling tools in the prototyping process of IoT devices, given its versatility, peripheral integration structure, among others features [2]. These platforms also act as essential educational tools for the practical development of applications by students, who take advantage of these tools' computational and hardware capabilities to prototype their solutions, that may integrate different sensors, actuators, and other peripherals [3].

Notwithstanding the advantages of development platforms, most IoT applications rely on the employment of multiple modules for data collection and transmission [4]. In many cases, it is more economically viable to design devices embedded with lower-cost off-the-shelf and IoT-enabled microcontrollers. In this context, printed circuit boards (PCBs) represent robust hardware solutions for device prototyping and manufacturing, providing the necessary electrical and mechanical connections for the device's circuitry electronic components [5].

Although PCB design is deeply related to fundamental concepts of electronic engineering, its studies are not adequately covered in the universities' curricula [6], which usually focus only on calculations for component dimensioning and forget to present some basic and more practical concepts of board design and industry standards. These aspects increase the lack of qualified professionals in a growing market that, according to reports, only North America will lose 55 % of its active professionals in the next ten years due to retirement [7]. A study published in November 2021 assessed the skills required for 400 open job positions at large technology companies such as Tesla, Lenovo, Intel, ABB, and others. About 52 of this job openings required PCB design skills [8].

There is still a significant shortage in literature for teaching and training strategies for PCB design, with few related papers. In [9], a learning module is implemented for the design of two PCBs (a continuity tester and a 4-bit DAC circuit), where the concepts worked in the approach are adapted for electrical engineering students and also for non-engineering students. Meanwhile in [10], a course is presented to students of electronics-related fields to introduce the main concepts of PCB production and design with the ECAD and MCAD tools.

Given the urgency for teaching and training strategies that meet the growing demand for highly skilled professionals, this paper presents the

Ediciones Universidad de Salamanca / cc by-nc-nd   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[15]

implementation of a learning module that addresses PCB design. In this module, essential concepts related to designing rules, electromagnetism effects on PCBs, and the best practices to mitigate these problems are presented to the attendees with the help of instructors with extensive practical experience in the field. Thus, this learning module aims to cover industrial practices that are not commonly covered by regular courses. The aspects discussed during the course are illustrated through a hands-on design project of an IoT-based device for vibration monitoring. The KiCad open-source computer-aided design (CAD) tool is used to develop the PCB project. The attendees' feedback indicates the module's effectiveness as a learning tool, supported by a strong foundation of practical concepts relevant to professional qualification.

The remaining of this paper is organized as follows. Section 2 presents the structure of the course modules and the learning goals, while Section 3 describes the implementation of the course. Section 4 discusses the main results and participant feedback, and Section 5 presents the conclusions and future work.

2   Course Modules and Learning Goals

The course was directed at students, professionals, and those interested in electronics. Since it covered some advanced topics in PCB design techniques, the learning action had as pre-requisites theoretical knowledge in electronic systems and components, practical experience in designing simple circuits, and CAD tool usage skills. The learning goals were to allow participants to understand better some of the potential problems that can be faced during the PCB development process and the best practices to be adopted. Choosing and purchasing suitable components, datasheets interpretation, PCB design, soldering principles, and electromagnetic effects on circuit boards were some of the covered topics. Fig. 1 illustrates the organization of the course that was divided into four modules.

2.1 Module 1 Introduction to PCB Design

The first module aims to provide an overview of aspects related to PCB design. An economic contextualization is done, approaching topics such as the

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[16]

silicon crisis that has been impacting the chip production and all the electronic devices production chain, and the difficulties found mainly by small and medium companies to purchase small quantities of components. It also discusses ways of searching for electronic components in the market, evaluating their availability, and considering the use of short lead-time components for orders to meet production demands. This last point is of utmost importance since replacing components after the project's end can cause the PCB's redesign, delaying the device manufacturing process.

The standards of PCB design are also discussed according to the IPC (ipc. org), which contains many recommendations and specifications of standardization that can be applied for developing electronic circuits. The KiCad tool is also introduced, discussing its main features, benefits, and the steps to design a PCB.
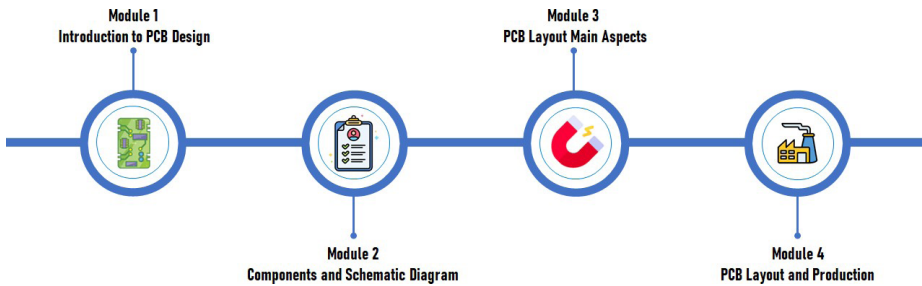

Fig. 1. Overview of the PCB course learning modules.

At the end of this module, the project to be developed throughout the course is introduced. It comprises the design of an IoT-based device PCB, its main components, and a basic model of the project's objective. Fig. 2 illustrates an overview of the PCB layout and the 3D model of the project developed in this learning module. Based on this project, their main components are searched to analyze their availability in the market, showing how to reduce the risks related to long lead-times and mitigate the need to re-designing the project. It also illustrates how to obtain and import the necessary files of each PCB component (libraries with component symbols and 3D model) to the KiCad platform.

The learning objectives of this module include: 1) be aware of the impacts of the silicon crisis on the purchase of electronic components; 2) understand the procedures and know some websites to search and evaluate the availability of

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[17]

electronic components that can be purchased in small or medium quantities; 3) know the existing framework and tools for the design of PCB and their features.

2.2 Module 2 Components and Schematic Diagram

This module starts with an overview of some IPC standards for PCB design to illustrate what a standard defines and how to interpret and use them, e.g., in the design of a PCB. Following this, the project to be developed throughout the learning module is discussed in more detail. Based on that, the aspects related to each one of the components are discussed and their proper organization in the circuit. The components were divided in sub-groups that can provide a better understanding of the whole circuit.

After this discussion, the KiCad platform is used to start the creation of the schematic sheet of the circuit. In this part, several best practices are discussed while the instructors exemplify the creation of the schematic diagram, as well as some features and configurations of the KiCad tool. It is also explained how to interpret the datasheet of the components, regarding their features, application recommendations and examples, and recommended soldering profiles. Since the instructors only create some parts of the schematic diagram to illustrate and exemplify some best practices during this phase, the creation of a different schematic, chosen by the participant, is set as a home exercise, in order to stimulate the practice of the aspects learned.
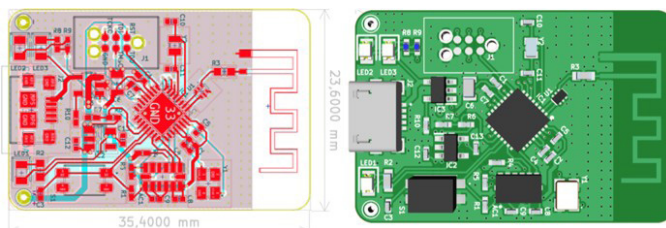

Fig. 2: IoT device project PCB layout and 3D Model.

The learning objectives of this module include: 1) learn how to organize the components of a circuit in functional groups; 2) understand and apply the design standards and best practices in the schematic design; 3) interpret the

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[18]

datasheets to understand the component's features and applications; 4) create parts of the schematic diagram on their own to practice their knowledge.

2.3 Module 3 PCB Layout Main Aspects

The third module of the course comprises the development of the PCB layout, i.e., the physical placement of components and routing of traces. At this point, several good practices were addressed regarding the mitigation of electromagnetic interference on the PCB that certain types of components can cause (e.g., radio frequency antennas), communication traces, and according to the frequency of the components and their position. In this context, some conceptual aspects of PCBs are initially discussed regarding their layers, dielectric material, and the behavior of the electric current in the traces. Followed by the most common causes of the electromagnetic interference (EMI) phenomenon (e.g., rip current and crosstalk) and the strategies commonly used to mitigate them (e.g., the use of decoupling capacitors).

The main features and tools of the KiCad platform and how to use them for PCB layout development were discussed. This includes how to import the components from the schematic diagram, as well as how to set some parameters (e.g., the minimum trace width and spacing, drill or roller size, the distance between components, and other design rules), which should be compliant with the constraints of the company chosen for PCB manufacturing. In this context, some PCB manufacturers are presented, discussing how to evaluate their manufacturing process capabilities.

Since this module covers only the initial PCB layout development, the complete layout is left as an exercise for the attendees so that the most appropriate placement of the components on the board could be accomplished in the last module of the course, when the instructors present their project and explain some other important PCB layout aspects and concerns.

In this context, as the learning objectives of this module, the participants must: 1) understand the electromagnetism behavior in PCBs; 2) know the concerns and best practices to avoid or mitigate the EMI in the design of PCB (regarding the placement of components and trace routing); 3) use the KiCad tool to develop the PCB layout and configure the tool to meet the constraints of the PCB manufacturer.

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[19]

2.4 Module 4 PCB Layout and Production

The forth module continues the development of the PCB layout, discussing the best practices regarding the placement of the components and traces routing in order to mitigate the EMI effects. In this part, a special attention is given to the RF antenna used in the project that requires some special attention to avoid that the noise produced by other components affect it, and vice-versa. Besides that, considering the different layers of the PCB, it is also discussed the importance of the ground planes and their interconnection, specially to reduce the EMI (such as rip current and crosstalk), but also the heat dissipation properties (e.g., during the the components soldering phases).

After complete the PCB layout, it is presented how to use the KiCad tools to create the 3D model of the PCB, based on the component libraries. It is also discussed how to create the production files, including those that should be used to send to a PCB manufacturer or as the input of a PCB CNC machine. Additionally, it is discussed the BOM (Bill of Materials) file that is also important in order to prepare the purchase order, or even send to the PCB manufacturer, if it provides this kind of service (besides print the PCB it also buy and solder the components).

Since in small projects, it is common to buy the PCB and the components separately, this learning module discusses several aspects regarding the soldering process of the PCB. These aspects include some theoretical concepts, that are important to understand the electromechanical connection between the PCB and the terminals of the components, and also illustrates the importance of follow the manufacturer recommendations for soldering, such as component's soldering profile (in order to avoid a bad electromechanical connection that can cause issues), and cold solder joint (that can cause and accelerate the degradation of the PCB). Additionally, it is presented some of the soldering techniques and materials (tin, solder flux) and their applications, as well as some best practices and issues in the manual soldering.

As the learning objectives of this module, the participants must: 1) understand the best practices for components placement and trace routing that can mitigate EMI; 2) use and configure the KiCad tools to generate the PCB production files; 3) understand the soldering concepts, techniques and related issues.

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[20]

3 PCB Course Implementation

The course is presented in the form of a project that aims to design an IoT device for monitoring movements based on an accelerometer to acquire data, and send it over a Bluetooth BLE interface. It is important to note that the goal of this project is to illustrate a practical implementation of a PCB of an IoT-based device. In this sense, rather than concern about achieve high-performance and compact dimensions and space-saving design solution, as in commercial sector devices, the focus is on a prototype solution with a level of complexity that can be used to illustrate some important aspects that must be considered for the design of a PCB. For instance, the use of digital communication between components and an RF antenna pose some common concerns in the development of such kind of devices, regarding EMI.

Based on that, this course covered aspects of all the phases of a PCB project design, from project idea, components evaluation, creation of schematic diagrams and PCB layout, to the PCB production (generating the production files, selecting and evaluating the PCB manufacturers, and the aspects for soldering the components).
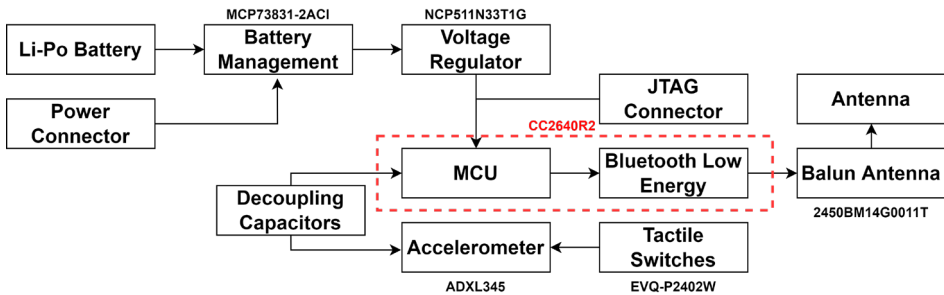

Fig. 3. Block diagram of the PCB project components.

3.1 IoT-based Device Project

The idea of an IoT-based device project is based on a Texas Instrument example project of a Bluetooth-enabled skin temperature measurement device, that use the Texas CC2640R2F wireless MCU microcontroller (ti.com/tool/TIDA01624). Instead of a temperature sensor, this project considers an

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[21]

accelerometer and some other changes, e.g., some components to manage the battery charge. Fig. 3 presents the project block diagram where the components are grouped according with their functions. The Bluetooth controller is integrated with the MCU, while the antenna is represented by another component that also requires a Balun antenna in order to balance its impedance. Another component is the JTAG connector used to provide a serial communication interface and program the microcontroller (update the firmware). The power supply circuit comprises several components, including a battery and its respective controller (to manage its charging/discharging), and a voltage regulator to provide 3.3V to the MCU. The accelerometer, the decoupling capacitors and a button are represented by other blocks.

Based on the project idea, the first step comprises a search and evaluation of the components that could be used, in terms of their features, requirements and availability in the market and delivery times, which is essential for the production of the project (for short and long term). In this context, besides illustrating how to find and interpret the technical information in the manufacturers datasheet, the instructors briefly discuss the impact of the silicon crisis in the production of chips, pointing out some concerns that the engineers should have in mind for selecting the electronic components for a project. They also provide some examples of search engines, suppliers and manufacturers that provide these kind of information. The application notes, usually present in the datasheet of some components, are also discussed, highlighting their importance as practical examples of usage of a given component in a circuit.
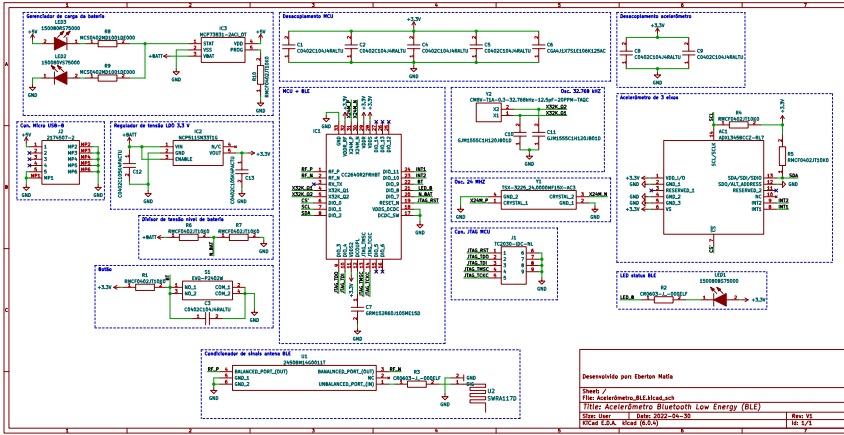
Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[22]

Fig. 4. Schematic of the IoT-based device project.

3.2 PCB Schematic and Layout Design

Along the course, the development of the project was supported by the KiCad framework . It provides tools for schematic design, PCB layout and 3D viewer, but also has several add-ons that can support the design of PCBs. The KiCad enables to configure several specification of the PCB project, in order to create production files that are compliant with the capabilities of the companies that will produce the PCBs.

Since the focus of the course was not in the KiCad usage, only its basic tools and features were presented, including how to find and load commercial components libraries (that includes schematic symbol, PCB footprint, 3D model, etc.). Note that, although the KiCad provide standard components that can be used to design a project, it is recommended to find and use the libraries of the components that will be used and are provided by the manufacturers.

After define the components that will be used in the project and include their libraries in the KiCad, the next step is the creation of the schematic diagram, defining the links between the terminals of the components. This task requires electronic engineer knowledge in digital/analog circuits, and can be assisted by the technical specification and the application examples that can be found in the datasheet of the components. Fig. 4 presents the complete

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[23]

schematic diagram of the projectwhere the components are grouped in some functional blocks as illustrated in Fig. 3.

The PCB is designed using the layout and routing tools of KiCad. In this part the instructors illustrate and explain several concerns and best practices for the PCB layout, especially considering strategies to avoid or mitigate EMI caused by rip current and crosstalk.

Initially, some general aspects regarding the placement of components were discussed. For instance, usually the PCB must attend the requirements of the device frame and/or ergonomic aspects, connectors should be in the borders, while antennas and oscillators crystals must be put isolated or far from the rest of the circuit, especially to avoid interference. Another practice is to position components in diagonal in order to facilitate the distribution of the traces when there is dimension constraints.

More advanced aspects were also discussed, such as the rip current and crosstalk that are mainly caused by the behavior of the electric current in the traces. After briefly explain the behavior and the electromagnetic characteristics of the electric current and their effects in the circuit, the layout of the PCB of the project was used as an example to illustrate and explain the most suitable places to put each component, as well as those that should be avoided.

Such aspects and the related best practices were further discussed during the routing process, since the position and path of the traces are the main causes of these phenomena. In this context, it was explained why it is not recommended to use the auto-routing tools to perform the routing for the whole system, since such tools usually do not consider all these factors. In order to avoid or mitigate these phenomena it is required to assure that the traces has an appropriate return path. This implies the proper use of ground planes, as well as decoupling capacitors. However, in commercial projects, that must attend size constraints, it is not always possible to completely avoid the routing issues, being necessary to adopt some strategies to reduce their impact in the circuit.

Fig. 2 illustrates the project complete PCB layout. After this phase, the production files can be generated using the tools provided by the KiCad. It also enables to set some configurations that may be necessary to make the project compliant with the PCB manufacturer production capabilities.

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[24]

3.3 Soldering Techniques

Once the PCB design has been completed and the production files created, a discussion regarding some manufacturers that provide services to produce PCBs and some soldering concepts were presented. In this context, the instructors discuss the characterization of soldering, intermetallic formation, tin alloy types, soldering flux types, and industrial soldering examples.

An important step in the electronic soldering process is the formation of the intermetallic, which is the electromechanical connection that allows the conduction of electric current between the terminals of electronic components and the PCB tracks. The intermetallic is created as the result of the union of the different metals during the soldering process. For instance, the used tin alloy, the copper tracks, and the component terminal material.
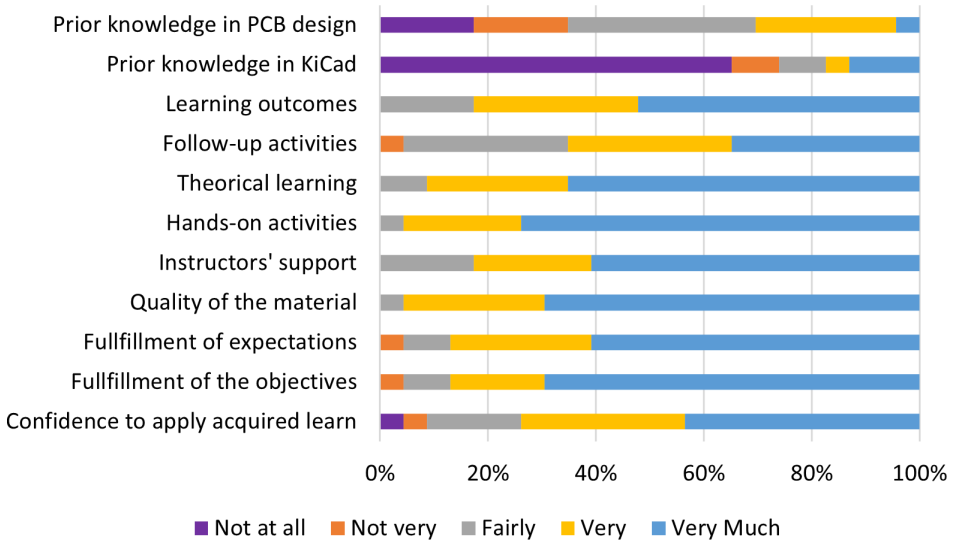

Fig. 5. Assessments of the participants' feedback.

Tin alloys for application in electronics are divided into two groups, lead alloys, and lead-free alloys, each one with specific features and applications. For instance, it is important to note that the lead-free alloy, preferred in the industry (Sn96.5 % Ag3 % *Cu*0.5 %), has a melting temperature of 217 °C,

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[25]

which is very close to the operating limit temperature of some electronic components. Equipment used in the lead-free welding process must be certified for this function, as there is little margin for error. This highlights the importance to evaluate the soldering profiles of the components that can also be used to define their soldering order and the methods used.

Tin alloys can be found in different forms according to their application, such as wire (with or without solder flux, used for manual soldering), ingots (used in soldering pots and wave soldering process), paste (used in SMT assembly lines), and balls (used in components with BGA encapsulation). The tin can be used with solder fluxes, which are acidic compounds that serve to clean the surface to be soldered, as well as protect it from oxidation during the soldering process. There are different types of fluxes, such as paste, gel, liquid and resinous liquids. The No-Clean flux has encapsulating agents, so it does not need to be removed at the end of the process. For other types of fluxes, cleaning with isopropyl alcohol is recommended, to avoid PCB corrosion and electrolysis damage over time.

4 Discussion and Participants Feedback

The learning module took place in May 2022, with a total duration of 8 hours, divided into four 2-hour online sessions. The dissemination of the course was done through e-mails and social medias of the DISRUPTIVE project partners, totaling 23 participants from four different higher educational institutions of Portugal, Brazil, and Spain. Fig. 5 presents a summary of the participants feedback survey. A Likert scale is used to indicate the level of agreement on different questions that evaluate the methodology and learning objectives of the module.

About 34.8 % and 73.9 % of the respondents reported having no or minimal experience with the design of PCBs and with the KiCad platform, respectively. After their participation, 82.6 % said to have significantly achieved the learning objectives, with 65.2 % being able to follow all of the introduced activities and concepts with some or considerable facility. The theoretical contents were very well evaluated by about 91.3 % of the participants, who considered topics such as design rules, technical standards, and

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[26]

electromagnetic behavior of PCBs fascinating and essential. An important point raised by the majority of the participants is that many of these concepts were not adequately introduced during their academic careers.

The hands-on activities in PCB design with the KiCad platform were approved by 95.7 % of the attendees, with 82.6 % being delighted with the monitoring provided by the instructors, having praised the didactics and knowledge presented in the theoretical lecturing and during the clarification of doubts. The support material provided had a 95.7 % approval rating. Nearly to 86.5 % of the participants met their expectations with the learning module. This same rate of 86.5 % was obtained when asked if the module's objectives were achieved. Despite the short duration of the learning module, 73.9 % of the participants feel confident that they can apply the learned concepts to the development of a PCB design with the KiCad platform.

Overall, the learning module has proved to be of significant value to the participants, who have had the opportunity to learn essential concepts for PCB design, leading them to reflect on essential electrical and design characteristics that are not commonly presented in the academic environment. The use of the KiCad platform was also highly pleasing to the participants and their training since many were not familiar with the open-source platform. Aiming at future learning modules that deepen the themes worked on or even covered other PCBs typologies, 56.5 % of the participants say they are very interested in participating, while the other 43.5 % would participate depending on what is covered.

5 Conclusions and Future Work

The IoT-based applications have shown several benefits in different sectors, contributing to the increase demand for IoT devices and consequently for qualified professionals able to develop them. In this sense, this paper presented a learning module that focuses on some practical issues and best practices during the development of PCBs for IoT devices. Through a hands-on activity methodology, the learning modules covered several aspects of the whole process of develop an IoT-based devices and design a PCB. The use of the

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication
Technologies for Innovation and Digital Transformation, pp. 13-29

[27]

KiCad tool allowed to approach in a more practical way the topics of the course, from the theoretical aspects to the best practices related to PCB. This also contributed to achieve learning outcomes, such as component sourcing, creation of the schematics and PCB layout, generation of the production files, and welding concepts.

The available time and the learning module being in a remote mode caused some limitations, both in interactions between tutors and participant and in the time to carry out the activities during the modules. Even so, homework challenges were made available, depending more on the individual dedication of the participants to have a better learning experience. Although the practical activities during the modules could be improved to increase the confidence of the participants in applying what was learned, the feedback from participants showed positive points regarding the theory, activities, learning material and support from the instructor. In spite of that the participants shown interest to subscribe in future editions. In this sense, the future work will be devoted to develop new editions of the course that cover other aspects and best practices in the development of PCBs for IoT-based devices.

Acknowledgments

References

1. Lohiya, R., Thakkar, A.: Application domains, evaluation data sets, and research challenges of iot: A systematic review. IEEE Internet of Things Journal **8**(11), 8774-8798 (2021). https://doi.org/10.1109/JIOT.2020.3048439
2. Singh, D., Sandhu, A., Sharma Thakur, A., Priyank, N.: An overview of iot hardware development platforms. International Journal on Emerging Technologies **11**(5), 155-163 (2020).

Ediciones Universidad de Salamanca / CC BY-NC-ND   Proceedings of the V Workshop on Disruptive Information and Communication Technologies for Innovation and Digital Transformation, pp. 13-29

[28]

3. Al-Masri, E., Kabu, S., Dixith, P.: Emerging hardware prototyping technologies as tools for learning. IEEE Access **8**, 80207-80217 (2020). https://doi.org/10.1109/ACCESS.2020.2991014

4. Aheleroff, S., Xu, X., Lu, Y., Aristizabal, M., Velásquez, J.P., Joa, B., Valencia, Y.: Iot-enabled smart appliances under industry 4.0: A case study. Advanced engineering informatics **43**, 101043 (2020).

5. Bhunia, S., Tehranipoor, M.: Chapter 4 printed circuit board (pcb): Design and test. In: Bhunia, S., Tehranipoor, M. (eds.) Hardware Security, pp. 81-105. Morgan Kaufmann (2019). https://doi.org/10.1016/B978-0-12-812477-2.00009-5

6. Aranguren, G., Etxaniz, J., López-Nozal, L.A.: Design of printed circuit boards in university? In: 2012 Technologies Applied to Electronics Teaching (TAEE). pp. 6-10 (2012). https://doi.org/10.1109/TAEE.2012.6235397

7. Mehendale, N., Koli, B., Mistry, K., Shah, S., Kelkar, V.: Analysis of skills required to get jobs in companies of electronics engineering domain. Available at SSRN 3965419 (2021).

8. Peterson, Z.: Why PCB Design Courses Are Essential in an Engineering Curriculum (March 2022), https://resources.altium.com/p/ why-pcb-design-courses-are-essential-engineering-curriculum

9. Liddicoat, A., Pan, J., Harris, J., Perks, G., Shepherd, L.: Development and assessment of a pcb layout and manufacturong laboratory module in introductory electric circuits for ee and non-ee majors. Industrial and Manufacturing Engineering (2010).

10. Danković, D., Vračar, L., Prijić, A., Prijić, Z.: An electromechanical approach to a printed circuit board design course. IEEE Transactions on Education **56**(4), 470-477 (2013). https://doi.org/10.1109/TE.2013.2257784

Ediciones Universidad de Salamanca / cc by-nc-nd   Proceedings of the V Workshop on Disruptive Information and Communication Technologies for Innovation and Digital Transformation, pp. 13-29

[29]