METHODS, RESOURCES AND RESULTS OF THE COURSE OF "DESIGN OF PRINTED CIRCUIT BOARDS" IN THE DEGREE IN INDUSTRIAL ELECTRONIC ENGINEERING

M. Fernández-Ros, M. Soler-Ortiz, R.M. García, N.N. Castellano, F. Portillo, J.A. Gázquez

CIAMBITAL, University of Almeria (SPAIN)

Abstract

The Teaching Innovation Project "Design of Printed Circuit Boards (PCB)" was created as a tool to solve the gap that had been detected in the Degree in Industrial Electronic Engineering at the University of Almería, between the skills that students must acquire and the mechanisms available to achieve those skills.

An analysis of the design cycle of an electronic circuit leads us to the selection of the stages on which the Innovation Project must focus, and the type and number of resources to be generated. These resources are in an initial design phase and have been partially applied, for the first time, in the 2019-2020 academic year, in the subject of Electronic Systems Design, in the second semester.

Digital resources (manuals and video tutorials) have been developed for a course on PCB Design for the Degree in Industrial Electronic Engineering at the University of Almería. These resources are a manual and a set of videos on *National Instruments® Circuit Design Suit* that includes the *Multisim®* program (for editing electrical circuit schematics and their simulation) and *Ultiboard®* program (for creating printed circuit boards). However, the simulation characteristics are outside the objectives of this work.

Parallel to the mentioned resources, an application methodology has been designed and the results were measured at the end of the academic year. The results are very positive since the degree of student satisfaction is very high, however, in contrast to this appreciation, some of the resources that have been developed are not valued as positively.

It is necessary to improve some of the resources and create self-assessment tools, which are considered essential if we want to promote the autonomous work of the student.

Keywords: SPOC, printed circuit board design, Electronics Engineering, Transversal competences, On-line teaching.

1 INTRODUCTION

In the 2019-2020 academic year, the Electronic Technology Area of the Engineering Department of the University of Almería launched the Teaching Innovation Project called "Design of printed circuit boards (PCB)". This project is initially aimed at students of the Degree in Industrial Electronic Engineering, with the goal of developing a set of resources that help students achieve the competencies [1] of the degree in relation to Printed Circuit Board design, in line with other works relying on additional resources to ensure students acquire the appropriate skills [2]. This competence is considered transversal and trains students in the ability to design and build electronic equipment.

In the last edition of ICERI2019, the bases of this Project were laid out [3]. In that paper, the need to have additional resources for the training of students in the design and manufacturing competencies mentioned in the previous paragraph was exposed, in addition to relating different software tools for PCB design. It was also mentioned that the chosen software was the Circuit Design Suite made up of *Multisim®* and *Ultiboard®* from National Instruments®. and from there, several resources have been developed, among them the book "Printed Circuit Design Manual with National Instruments *Circuit Design Suite*" [4]. Given the advantages of video resources, [5] several videos are in the preparation and revision phase and it is expected that at the beginning of the 2020/2021 academic year they will be finalized and can be used in that course.

2 METHODOLOGY

2.1 Design Method

Electronic design is a competence for students of the Degree in Industrial Electronic Engineering. Electronic design must be understood in its broadest content, encompassing everything from the idea to the prototype and / or final product. On this path there are several stages:

1. Definition of the specifications: What you want to do.
2. Analysis of the idea: Encompasses the mathematical part of the design. Tools such as spreadsheets, numerical calculation software, etc. can be used. In this stage, the circuit that must meet the specifications of stage 1 is shaped.
3. Creating and editing the electrical schematic: the circuit is transferred from paper to electrical schematics' design program. This stage is called editing. The purpose of editing electrical schematics can be the simulation of the circuit, or its transfer to the design of the printed circuit.
4. Circuit simulation: In this stage the correct operation of the circuit is verified, always within the possibilities of the program being used.
5. Printed circuit design: Once the circuit is drawn, the necessary characteristics are transferred to a printed circuit design program. In the design program, the location of the components on the board is decided and the tracks that join their pins are traced. You can choose to print the result and manufacture the board using homemade methods, or generate specific files used in the industry and ask a specialized company to manufacture it.
6. Assembly: All the components are positioned or inserted on the printed circuit board, then soldered. Some companies that manufacture the printed circuits also offer the possibility of assembling the components.
7. Functionality test: The necessary tests and measures are carried out on the physical assembly to verify that it abides the specifications defined in stage 1 and, find any mistakes that could've been made during any of the previous stages.
8. Construction of a box for the printed circuit board: in this box the support of the printed circuit and the holes where the different input / output connectors will be inserted and machined. It is the final product.

The objectives of the Teaching Innovation Project enforce stages 3 and 5 above, which are considered essential to achieve the objectives set. The rest of the stages are developed by other competences, which are sufficiently covered in other subjects.

2.2 Organization of the Teaching Innovation Project

Once the need to generate additional resources for the training of students of the Degree in Industrial Electronic Engineering was detected, it was decided that they should be organized into a course with the Small Private Open Course (SPOC) format. This kind of format allows the teacher to address a small number of students while maintaining a certain degree of interactivity, necessary to achieve the programmed objectives. On the other hand, it encourages student's autonomous work, essential in other transversal competences.

Within the Teaching Innovation Project, the following lines of work were established:

1. Design software selection.
2. Selection of an electronic circuit for design.
3. Design division into 4 sections:
    a) Edition of electrical schematics.
    b) PCB layout.
    c) Electronic symbol editing.
    d) Footprint editing.

Item 1 was resolved when the Software Suite *Multisim* & *Ultiboard* was selected. Thus, University of Almería acquired software licenses, including licenses to give to students so they have access to the program for free.

For item 2, an Audio Power Amplifier was selected. This type of circuit is studied in the 3rd year of the Degree, in the subject of Analog Electronics. It is a circuit which students are familiar with and it's made up of a series of electronic devices sufficient for the design of the printed circuit to have a number of diverse components, both electronic and mechanical, such as:

- Fixed resistors.
- Variable resistors.
- Fuse.
- Large capacity electrolytic capacitors.
- Ceramic capacitors.
- Integrated circuit.
- Mechanical connection elements.
- Mechanical fixing elements.

Items 3.a and 3.b are the core of the design, starting with the edition of the electrical schematic (*Multisim*) and finally making the PCB (*Ultiboard*).

Items 3.c and 3.d are a complement to the previous ones. If the symbol of the electronic component does not exist, it must be designed in *Multisim* (3.c). Additionally, if the component doesn't have the required footprint attached, it must be created as well using Ultiboard (3.d).

The 4 blocks of item 3 are covered in the developed manual [4]. Items 3.a and 3.b are the first two chapters and items 3.c and 3.d are the final annexes.

In addition to this manual, a set of videos are being made to give multimedia support to the course and increase the course's accessibility. The videos have the following structure:

1 Basic introduction to printed circuit boards design.

2 Basic structure of *Multisim*. Databases and editing tools.

3 Editing electrical schematics with *Multisim*.

4 Editing electronic device symbols with *Multisim*.

5 Preparation of an electronic component's footprint, considering its dimensions, terminals, and general shape.

6 *Ultiboard* basic structure. Databases.

7 Creation of printed circuit boards with *Ultiboard*.

8 Editing footprints with *Ultiboard*.

2.3 The SPOC course

In the soon to be implemented course, students are guided in the design of a printed circuit board. In this case, a power amplifier of the audio frequency band. Along the course, they must carry out the following steps:

- Find the circuit components at an electronic component dealer. There they choose the specific component, brand, model, etc. that meets the design specifications.
- Check if *Multisim* has a suitable symbol for that component. If not, the student will have to create it.
- Check if Ultiboard has the footprint of that component. If not, the student will have to create it.
- Draw the electrical schematic of the electronic circuit.
- Prepare the electrical schematic to transfer its information to the design program.
- Create the printed circuit board.

All the previous steps are collected in the flow chart of Figure 1.

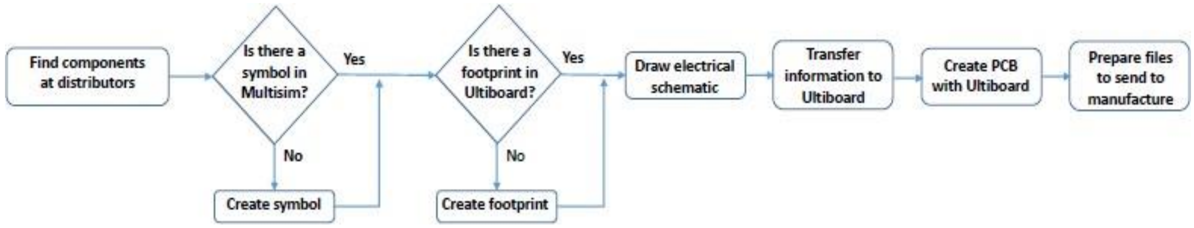

Figure 1. PCBs course methodology.

3 RESULTS

The methodology set out in the previous section has been applied in the subject of Electronic Systems Design of the 4th year of the Degree in Industrial Electronic Engineering, subsequently the students answered a survey in which the following results were obtained:

- Figure 2 and Figure 3 show the results of the *Multisim* and Ultiboard survey. This software has been widely accepted by students, mainly *Multisim*, with ratings above 80%, in contrast to Ultiboard, where the rating is slightly below 50%. This may be because the students use the simulation tools that *Multisim* has available to test their circuits, however, this characteristic is not the object of study in the Innovation Project.
- The last two columns in Figure 2 and Figure 3 provide results on other programs on the market that students have used at some point. The ratings given by students of *Multisim* and *Ultiboard* improve their experience by 70% over other programs.
- It is noteworthy that the edition of symbols and the creation of footprints are the items that have been rated the lowest, as well as the PCB design, which is slightly below 50% acceptance. In contrast to these evaluations, the general satisfaction with both tools exceeds 80% acceptance.

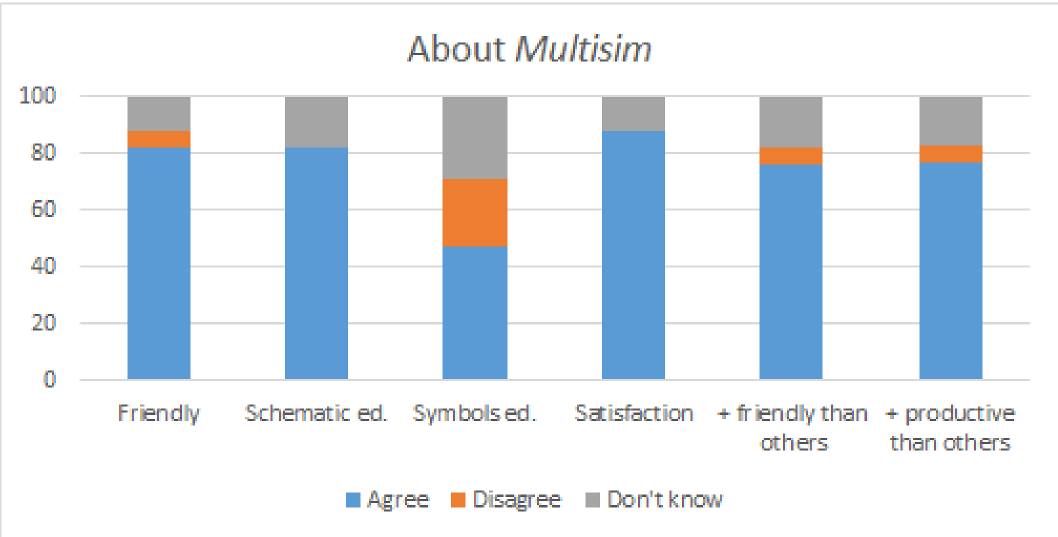

Figure 2. Multisim surveyed properties.

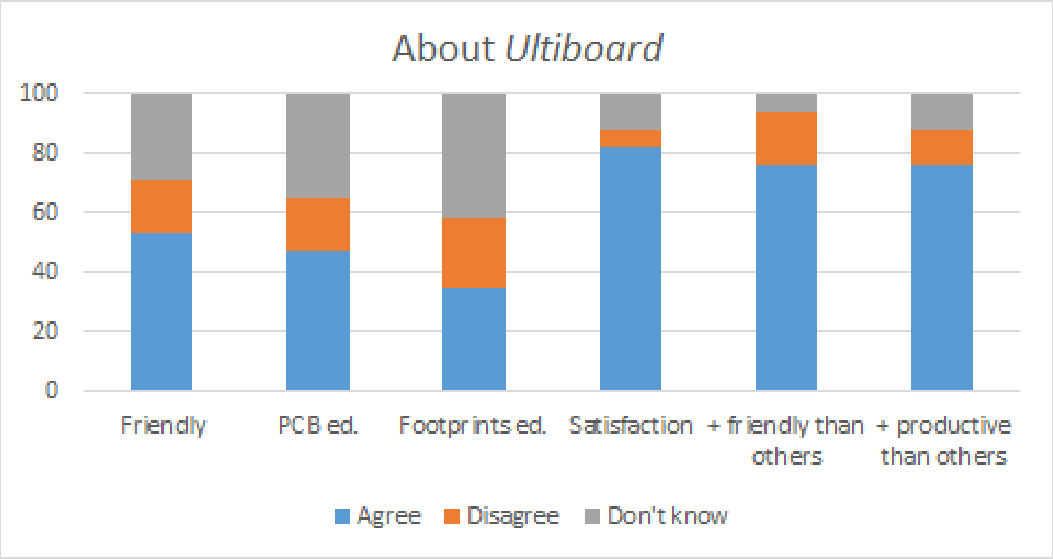

Figure 3. Ultiboard surveyed properties.

In another section of the survey, students state that they are satisfied with the tools made available to them to achieve the Degree's skills, however, they do not consider the resources made available to them (manuals, video tutorials, etc.) to be sufficient. Figure 4 shows that there are the same number of satisfied people, those who are not and those who do not have criteria.

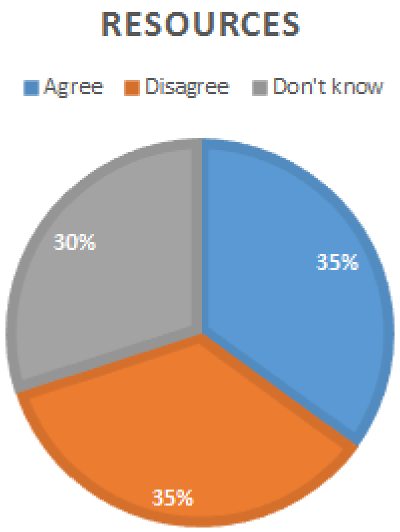

Figure 4. Assessment of the resources made available to students.

4 CONCLUSIONS

The development and application of this Teaching Innovation Project resources has not been without difficulties (we assume that, like the rest of professionals) due to the restrictions imposed by COVID-19. However, this situation has also served as an incentive since it has shown the need to have digital resources that complement and ultimately can replace ordinary teaching. Already, from the beginning, these resources were conceived as an autonomous tool that would allow students to properly acquire the necessary transversal competencies during their Degree studies.

There is still a long way to go, though. The results show that, although our students are satisfied with the tools that they have been taught to handle, with the edition of electrical schematics and creation of PCBs, some aspects have to be worked more, specifically those that we could call peripheral like the edition of symbols and footprints. Therefore, we are committed to improving the development of these resources. The creation of self-assessment tools is also pending.

ACKNOWLEDGEMENTS

The authors of this paper would like to thank University of Almería for funding this work through the Teaching Innovation Project "Printed Circuit Board (PCB) Design". Call 2019-2020.

Thanks to the help of the CIAMBITAL group, in the project "Advances in sensorization and processing of natural signals in the ELF band" (TEC2014-60132-P).

REFERENCES

[1] University-of-Almería, "Competencias específicas de título - Grado en Ingeniería Electrónica Industrial," *University of Almería*, 2020. .

[2] C. Ma, K. Srishti, W. Shi, Y. Okada, and R. Bose, "Educational Material Development Framework Based on Linked Data for Iot Security," *ICERI2017 Proc.*, vol. 1, no. November, pp. 8048–8057, 2017.

[3] M. Fernández-Ros, M. Soler-Ortiz, R. M. García, N. N. Castellano, and J. A. Gazquez, "INTEGRATION OF PRINTED CIRCUIT BOARD DESIGN SPOC IN ELECTRONIC ENGINEERING DEGREE," *Int. Conf. Educ. Res. Innov.*, p. 7, 2019.

[4] M. Fernández-Ros, M. J. Soler-Ortiz, N. Novas-Castellano, R. M. García-Salvador, and J. A. Gázquez-Parra, *Manual de diseño de circuitos impresos con Circuit Design Suite de National Instruments*, 1ª. Almería, 2020.

[5] V. Pisarenko, "Foreign Language Training Based on Audio-Visual Technologies," *ICERI2017 Proc.*, vol. 1, no. November, pp. 7108–7116, 2017.