Unit 4 "Internet based education, distance learning"

The use of design automation tools in training young specialists in radio electronics

A. A. SEREDA, S. V. SHALUPAEV, Yu. V. NIKITJUK
Francisk Skorina Gomel State University, Gomel, Republic of Belarus

Abstract
The paper considers the stages of mastering the computer-aided design of printed circuit boards for students of radio engineering specialties. The scheme allows a recent graduate to more effectively master the phases of constructing electronic devices and acquire professional skills in the development of printed wiring elements.

Introduction
Using classical methods and means of "manual" design, it is currently impossible to carry out quality and timely production design engineering. The complexity of the designed objects makes it impossible to produce sufficiently high-quality design documentation within a reasonable period of time. The design automation system is intended to carry out the design process with an optimal distribution of functions between the developer and the PC and to ensure maximum automation of all design procedures.

Phases of mastering systems for designing printed wiring elements
To develop any electronic device, it is necessary to go through several phases: develop the concept of the future product, its input and output parameters, operating modes and then develop the electrical circuits of the device, implement them on printed circuit boards, design the frame product, and much more. Computer-aided design systems are widely used to optimize this process. They allow preliminary computer modeling of the operation of circuit schematics, the optimal layout of radioelements on a printed circuit board and tracing the conductors, and, if necessary, make design changes before manufacturing a prototype.
Currently, there are numerous software packages to automate the processes of circuit simulation. These are Sprint-Layout, Eagle, DipTrace, ExpressPCB, TARGET 3001, FreePCB, Kicad, DesignSpark PCB, SoloPCB Design, PCB123, TopoR, Pad2Pad, PCB-Investigator, EDWinXP, P-CAD, Mentor Graphics PADS, ZenitPCB, CADSTAR Express, ZofzPCB 3D Gerber Viewer, PCBWeb, CometCAD, Layo1 PCB, PCB Elegance, NI Ultiboard, CAM350, BoardMaker3, GerberLogix, PCB Artist, VUTRAX, CADintPCB, etc. A brief description on each of them is provided by [1] and the official websites of the manufacturers. Each of them has its own advantages and disadvantages. Some packages are free, but they have a limited feature set (for example, GerberLogix); some are focused on a narrow range of tasks (for example, only for the development of a drawing of conductors, as in Layo1 PCB and Sprint-Layout, or only for modeling electrical circuits); others have good functionality, but a rather complex interface, etc.
Specialists in radio electronics should have experience in various software products that allow to development of an electronic device of any complexity in the shortest possible time and as efficiently as possible. Laboratory workshops facilitate the process. Teachers and students majoring in radio engineering can use them not only during laboratory classes but also as practical reference guides for self-study and mastering, as well as for distance learning. While studying the principles of designing electronic devices, the future specialist needs to follow the "simple-to-complex" principle, consistently mastering the principles of designing printed circuit boards. For example, the first projects can be performed with Sprint-Layout software. It has a simple, intuitive interface but a limited feature set in automating the layout of elements and

tracing conductors, designing multilayer boards, importing electrical circuits, etc. So, it is ideal for a beginner radio amateur and allows mastering the steps and features of the manual and semi-automatic design of printed circuit boards without building 3D models of radioelements and the possibility of importing electrical circuits into the project. On receiving basic skills in such a software product, it is easier for a young specialist to master professional design packages that allow developing a multilayer board project using vias, signal and ground layers, the ability to import electrical circuits, etc. For example, the second software product to learn could be Ultiboard by National Instruments. Here you can import electrical circuits from the Multisim program, develop 3D models of radioelements, create vias, prohibition zones, and much more. After mastering this product, a future specialist can move on to mastering professional end-to-end design programs that allow all phases of the electronic device development to be implemented in a single project, i.e., to export a developed circuit diagram to a module for designing printed circuit boards, place the resulting three-dimensional models in a 3D modeling module and convert them to drawings for design documentation. Altium Designer can be such a software product. At the same time, some of the specialized interface elements, functions, menu items will be easier to understand and master for a specialist who has already become familiar with easier software packages.

Conclusion

The concept described above was implemented in laboratory workshops for students of the Department of Radiophysics and Electronics at Francisk Skorina Gomel State University. Starting with simpler design systems in the early years of study and mastering more professional software packages in senior years, graduates become specialists in radio electronics and can work effectively in the electronics industry. Fig. 1 shows the development results of projects in different design systems.
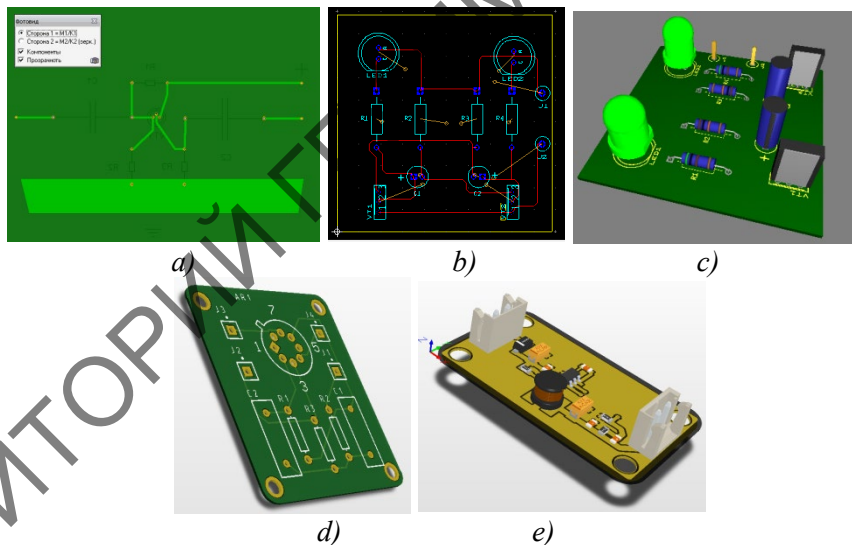

Fig. 1. The result of project development in different design systems:
a) – Sprint – Layout; b), c) – Ultiboard; d), e) – Altium Designer

References

[1] PCB design software [Electronic resource] / MMO "Electric soldering iron site" – URL: https://cxem.net/software/soft_PCB.php.