A Multimedia Course for the Design of Microsystem Packages

Zsolt Illyefalvi-Vitéz, Miklós Ruszinkó, Norocel-Dragos Codreanu*, Lóránt Hertel and Péter Bojta
Department of Electronics Technology, Budapest University of Technology and Economics
Goldman t. 3, Budapest, H-1111, Hungary
E-mail: illye@ett.bme.hu, *Phone:* +36-1-463-2740, *Fax:* +36-1-463-4118
*CETTI, Politehnica University of Bucharest

Abstract

Teaching the design of microsystem packages is one of the most important topics in the BSc and MSc level university education of electronic engineering. Special emphases should be laid on teaching design, according to the general opinion that industrial companies are suffering from the shortage of design engineering staff. In addition to provide courses on CAD design in regular university curricula, courses and training should be developed and provided for industrial engineers as well. The aim of such courses is to bridge over the gap between the knowledge that is provided by the different undergraduate modules and that is necessary and relevant for the industrial labor market and/or requested by the graduate educational modules for further qualification. Internet accessible, interactive, multimedia equipped, distant learning virtual laboratory experiments have to be offered for exploratory training to improve the knowledge of industrial engineers. Present paper reviews a part of a course that tries to meet all the above mentioned requirements.

1. Introduction

The issues of Computer Aided Design (CAD) of electronic circuits and subsystems are taught at different levels for the students of Electrical Engineering at the Budapest University of Technology and Economics. For the promotion of the lecture courses and the laboratory experiments, the CAD systems of Cadence, in particular the the OrCAD design system are applied.

1. Four-hour lecture course and four-hour laboratory exercise are given for all students of a year (ca. 400 students) in the middle of the curriculum (in the 5^{th} semester), as the part of the core subject 'Electronics Technology' [1-4].

2. Detailed knowledge about CAD is lectured for the students of a master degree program 'Circuit Modules' (ca. 30 students/year in the 7. semester) [5]. The duration of the lecture course is about 56 hours, but it includes general considerations of circuit design as well. An additional 28-hour laboratory training is dedicated for CAD based on the OrCAD design system {6].

3. Students of the degree programs of 'Circuit Modules' and other ones use the OrCAD design system and other Cadence tools for the design of their circuit modules in the frame of 'Project Laboratory' and 'Diploma Project'.

2. Teaching CAD Basics in 'Electronics Technology'

The core subject 'Electronics Technology' contains four hour lectures and a four-hour laboratory experiment dedicated to CAD (Computer Aided Design) in the 5^{th} and 6^{th} semesters. This subject is obligatory for all students of Electrical Engineering, and it is the only subject to teach CAD basics.

The topics of the lectures are as follows [3]:

1. CAD process flow (block diagram).
2. Circuit diagram (schematic) editing, using Component Information System (CIS)
3. Basic structures of Printed Wiring Board (PWBs), most important design rules.
4. Layout editing, documentation and post-processing.

On the basis of the knowledge provided by these lectures, in the complementary laboratory experiment all students have to design a simple, typical circuit module using OrCAD.

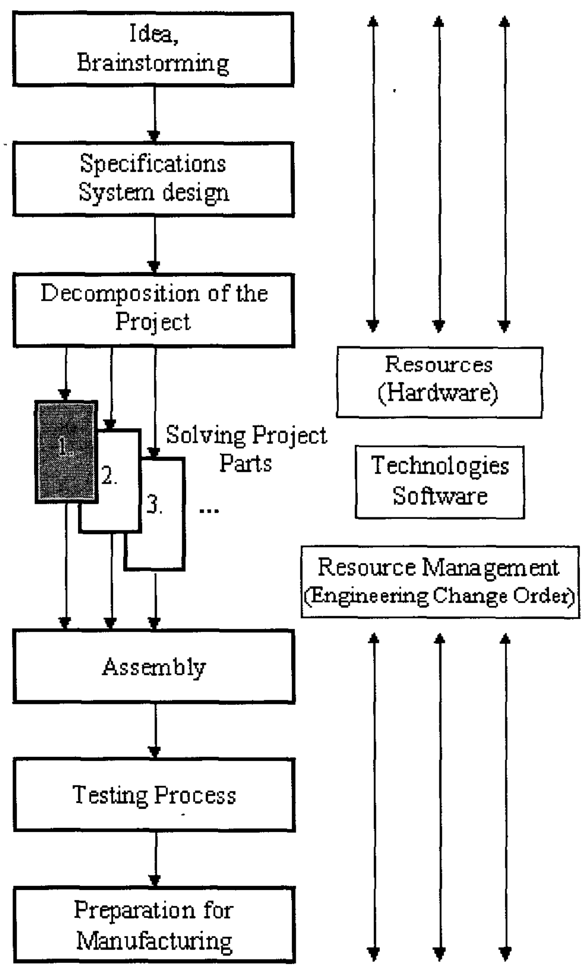


Fig. 1. The flowchart of CAD system design

2.1 Flowcharts of the Computer Aided Design Process

The design steps are discussed using two flowcharts. Figure 1. shows the steps of system (equipment) design.

System design begins with the summary of the objectives (e.g. market demand) and the ideas why to design and create a new product, and flows until the product is completely prepared for manufacturing. This process applies many hardware and software tools, and takes into consideration all information of the environment where the product will be used. The lectures of system design should define and clarify many important concepts including the topics, as follows:

- Circuit modules: the hierarchy of electronic systems, subsystems and components;
- Construction: physical arrangement, case and cover design, taking into consideration thermal conditions, electromagnetic compatibility, noise protection, etc.);
- Ergonomics: the design of interfaces to other systems and to the user(s);
- Logistics: acquisition and stocking materials.

Circuit module design (Figure 2.) is in the focus.

The flowchart of the CAD software of electronic circuit module design appears as a rather complex graph, because almost all elements are in relation to each other. However, there is an important distinction between the more general 'Engineering', which includes circuit specification, technology selection, modeling and similar design steps, and the 'Editing' or simply 'Design', which includes the real CAD of the module.

The main parts of the 'Design' are the schematic editing and the layout editing. There are different tasks for the user i.e. getting information about the performance of the circuit, and for the computer that includes the exact editing, processing, and verifying steps.

For better understanding, the flowchart is divided into three different columns:

- most important design steps;
- additional processes;
- databases, required for processes.

The design is completed by the documentation.

Since the diagrams are too large for a single Power Point slide, they are divided into parts. The first slide shows both diagrams completely, with no words, giving an overview for the students. The first chart is divided into two slides, with an overlapping in the middle. The second chart with all words and animations requires eight slides, overlapping by one block on each part, in the interest of continuity.

2.2 The Schematic Editor and the Component Information System (CIS)

The schematic editor has a Windows-like Graphical User Interface (GUI) with the usual functions. There are some specialties of CAD to teach:

- structure of the symbol library, Part Search command;
- schematic hierarchy and page structure (Figure 3.);

schematic editing.

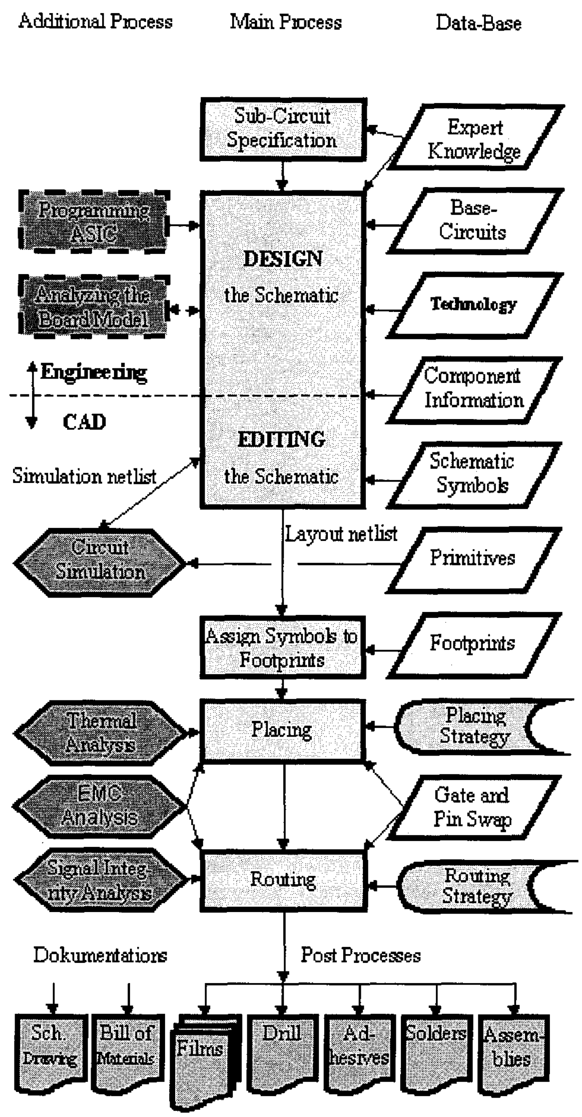


Fig. 2. The flowchart of circuit module design

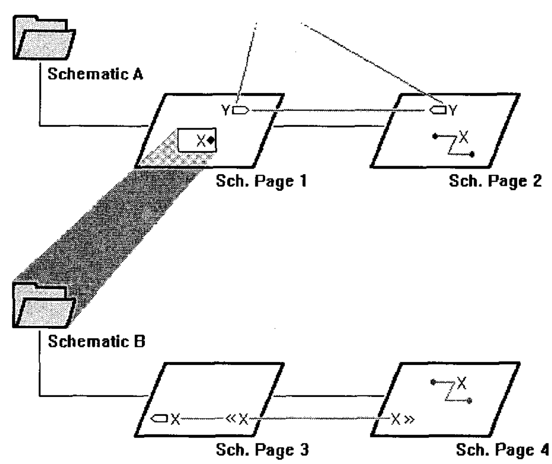


Fig. 3. The illustration of hierarchical design [6]

The steps of schematic editing are as follows:

- placing and editing symbols (if they are not available in the required form);
- generating connections (not only wires are required, but also special buses, ports and labels);
- editing part properties and attributes;
- miscellaneous (date, document number, title, name, revision, etc.);

Possibilities of Locale Part Database and Internet-based (online Component Assistance) use of CIS for:

- prices;
- shipping possibilities (e.g. from stock);
- datasheet acquisition.

2.3 Structure of the Layout. Design Rules.

In accordance with the usual structure of Printed Wiring Boards (PWBs), CAD softwares use layer structures, too. Fig. 4. shows a complete circuit module with a four-layer PWB structure including the following layers:

- outer copper layers for the component-side and the solder-side patterns;
- oxidized (black) copper layers with the power- and ground-plane patterns;
- isolation (pre-preg) layers.

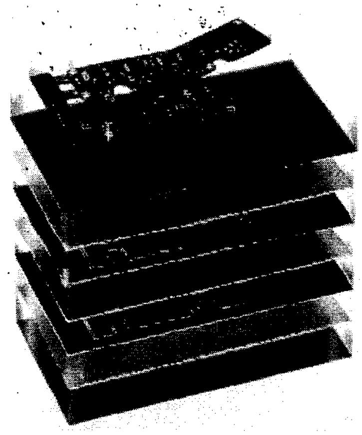


Fig. 4. Illustration of multilayer PWB

In addition to the physical 'functional' layers of PWBs, the CAD software uses documentation layers, too. Table 1. presents with animation the complete layer structure of a CAD system. Each row appears on the lecture in the order of the numbers. The following comments have to be added to each layer:

1. This is the only layer in a single-sided PWB.
2. "Virtual layer", the bases of drilling.
3. This is the component side of a double-sided through-hole mounted PWB.
4. These are the power and ground layers of a multilayer PWB, using thermal pads.
5. Additional inner layers, appearing always by pairs.
6. SM (Solder Mask): Protective layer for wave soldering and against corrosion.
7. Component position drawing for manual purpose.
8. Mask for solder paste printing.
9. Component coordinates for CNC placing tools.

9	Assembly Top
8	Solder Paste Top
7	Silk SCreen Top
6	Solder Mask Top
3	TOP (Component side)
5	INNers...
5	INNer3
5	INNer1
4	PoWeR
4	GrouND
5	INNer2
5	INNer4
5	INNers...
1	BOTtom (Solder side)
2	DRiLl (and DRill Drawing)
6	Solder Mask Bottom
7	Silk Screen Bottom
8	Solder Paste Bottom
9	Assembly Bottom

Table 1. The illustration of layer structure

The most important design rules are as follows:

- defining power and ground system (estimation rule for current density and track resistance: a 35 μm thick, 1 mm wide track of 1 m length has approx. 0.5 Ohm);
- defining pattern resolution (general spacing and track width);
- defining the required number of layers (theoretically 2 layers may be enough).

2.4 Layout editing and documentation

The most important steps of the design flow are demonstrated here, using the parts of OrCAD:

- Padstack contains the parameters of all pads used in the symbols (they can be different on each layer);
- Footprint: viewing and editing symbols used in the layout, containing pads, contour, reference designator;
- Footprint assignment: annotation in the Schematic Capture;
- Placing: strategy and assistant tool;
- Routing: strategies (nets, layers, sweep, options, and costs);
- Documentation (Gerber, Excellon formats).

2.5 Laboratory exercise

The four-hour laboratory exercise is sufficient for the complete design of a simple circuit module, however it is not suitable to teach the students for the individual use of the OrCAD system. The students use a guide, and can complete the design with no help of the instructor (who is present just to provide assistance if for any reason it is necessary). The students' guide contains the followings:

- general knowledge about editing methods (group selection for moving components and editing attributes);

• a sequence of tasks and procedures with their software command description (e.g. generating connections with „Tab" menu Place->Wire, or „Hotkey" Shift+W, or „Toolbar"  icon).

The guide is separated into 4 pages including topics:

• OrCAD Capture CIS;
• Pspice AD;
• Layout Plus;
• Helpful figures about them.

This structure allows access to the information without turning over pages.

An analogue amplifier (Figure 5.) is given to the students to exercise the design process. The circuit diagram contains part symbols of the base component libraries.

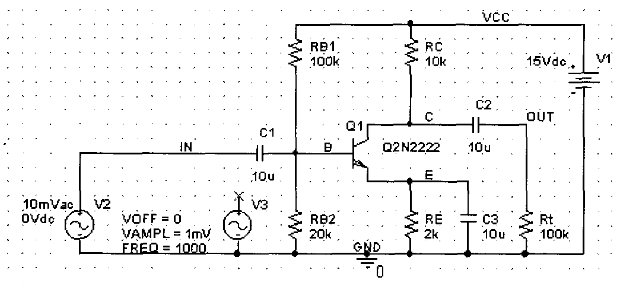


Fig. 5. The circuit diagram

The simulation results are bias point values, graphical presentation of the time domain response (Figure 6.) and the Bode-plot (Figure 7.).

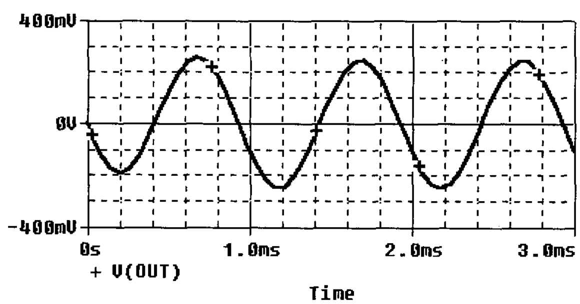


Fig. 6. The result of Transient analysis

The students can compare their results to the examples individually. Figure 8. shows the results of the manual layout design and the assembly drawing.

3. Teaching CAD in the Degree Program

Students take part in laboratory exercises in 2 semesters, 2 hours a week. The main goal is to learn the individual use of OrCAD. The topic of the 1st semester (Table 2.) is circuit design, schematic editing, simulation and acquisition of component data (cost, datasheet). The topic of 2nd semester (Table 3.) is layout design and realization, i.e. assembling and circuit adjusting.

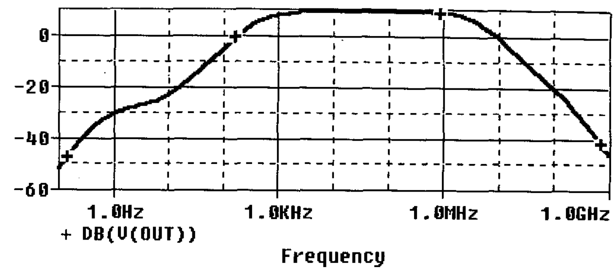
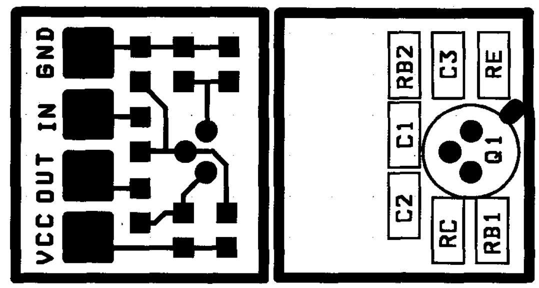


Fig. 7. The result of AC analysis



a.) Bottom layer b.) Assembly drawings

Fig. 8. The results of layout design

Week	Topic
1	Introductory meeting
2 - 3	Introduction into the use of OrCAD
4	Choosing a circuit
5 - 6	Editing the circuit schematic
7	Simulation with OrCAD Pspice
8	Correcting or redesigning the circuit
9	First report: schematic, and simulation results
10	Acquiring the datasheets of the applied components
11	Calculation circuit costs
12	Creating documentation
13	Oral report
14	Completion and corrections

Table 2. Time table of the 1st Semester

Week	Topic
1	Introductory meeting
2	Lecture about the main layout design rules
3	Acquisition of required components. Choosing cases and fixing type. Placing components
4 - 6	Layout routing
7	Post processing: Creating CAM files (aperture list, CNC drill file, layout films)
8	Manufacturing of PWB
9 - 10	Assembly and fixing the circuit in the case
11	Adjusting of the circuit
12	Creating documentation. Presenting the circuit.
13	Oral report
14	Completion and corrections

Table 3. Time table of the 2nd Semester

In the 3rd semester the students analyze their circuit modules in the course of the exercise of the Quality Control Laboratory, using complex quality management methods.

4. Project Laboratory and Diploma Project

There are many students of Electrical Engineering, whose tasks of Project Laboratory and/or Diploma Project include circuit design and realization. These students also have to use OrCAD system for the completion of their work.

Another interesting Project Laboratory task for a few students is to analyze the operation and performance of a selected part of OrCAD [7]. The better understanding of the operation of the system makes possible the improvement of the students' guide and also provides assistance to the users for successful design.

5. Hardware and Software Environment

For teaching CAD in the Electrical Engineering Branch, OrCAD 9.1 is available thanks to Cadence.

From the structural point of view, there is a License Server (LS), two packets of software and as many workstations as it is required. The installed programs and their functions are usable on the workstations when they are enabled at the moment of start by the LS. The number of active workstations at the same time is limited.

One of the packets is used for teaching CAD at the Department of Electronics Technology.

The other packet can be installed on any computers of the Faculty of Electrical Engineering and Informatics, to teach Computer Aided Design for the students of any other departments.

6. Conclusions, Development Strategies

Having recognized the industrial need for courses on microsystem packaging design, a self-financed project has been initiated to develop the multimedia content of our existing course at the Budapest University of Technology and Economics. This course of microsystem packaging design is the part of an MSc module for the education of microsystem and circuit packaging technology. The design course contains lectures and laboratory experiments. On the lectures the students get acquainted with the design rules and the use of the Cadence OrCAD software, while in the laboratory the students have to design, simulate, realize and document a functional circuit. In the class the lecturer uses PowerPoint presentations and other multimedia tools to present the capabilities of the CAD programs. The aim of the project is to develop an Internet based multimedia course, which can be easily accessed by industrial engineers and university students as well.

The course will focus on the engineering process flow and the core design tasks, as follows:

- schematic based design entry;
- digital, analog, and mixed-signal simulations;
- printed circuit board layout design;
- design for test and for electromagnetic compatibility.

The interactive features of the different modules of Cadence OrCAD, like Capture, PSpice, CIS (Component Information System), ICA (Internet Component Assistant), CDS (Component Data Server), and others, will be exploited in virtual experiments to enhance the efficiency of the course.

Internet accessible multimedia course and a systematic set of virtual laboratory experiments will be developed and offered for industrial engineers and university students to improve their knowledge in microsystem packaging design.

7. Acknowledgments

On behalf of the Budapest University of Technology and Economics, the Faculty of Electrical Engineering and Informatics, the Department of Electronics Technology, and all students and staff members, the authors would like to express their thanks for the educational support which was provided by PannonCAD Ltd. (the Hungarian representative of Cadence) and the Cadence Inc. by providing the license for the use these CAD systems.

8. References

1. Illyefalvi-Vitéz,Zs.; Németh,P.; Szikora,B.: "Problem-Oriented Education of Electronics Technology at the Technical University of Budapest". *Proceedings 47th Electronic Components & Technology Conference*, San Jose, California, May 18-21, 1997, ISBN 0-7803-3858-8, pp.942-950.
2. Illyefalvi-Vitéz,Zs.; Ruszinkó,M.; Pinkola,J.; Harsányi,G.: Lab Sessions and Prototyping in Interconnection and Packaging Education. *Proceedings of 23rd IEEE/CPMT International Electronics Manufacturing Technology Symposium,* Austin Texas (USA), October 19-21 1998. pp.362-372.
3. Illyefalvi-Vitéz,Zs.; Ripka,G.; Harsányi,G.: Elektronikai technológia. (A CD-ROM book.) Muegyetemi Kiado, Budapest, 2001. (Mainly in Hungarian, partly in English)
4. Varadarajan,M.; Illyefalvi-Vitéz,Zs.; Zimmermann,J.; Tummala,R.R.: Fundamentals of System-Level PWB Technologies. Chapter 16. in Tummala,R.R. (editor): Fundamentals of Microsystems Packaging, McGraw-Hill, USA, 2001. pp.612-657.
5. Illyefalvi-Vitéz,Zs.; Hajdu,I.; Ripka,G.: How to Teach Electronics Packaging Technology. *Proceedings of 23rd International Spring Seminar on Electronics Technology*, Calimanesti-Caciulata (Romania), May 5-9, 2001. pp.7-12.
6. OrCAD User's Guide. Published by Cadence Inc., 2001.
7. Drumea,A.; Svasta,P.: Fine Tuning of Autorouter Parameters in OrCAD Environment. *Proceeding of the 6th International Symposium for Design and Technology for Electronic Modules. SIITME 2000.* Bucharest (Romania), September 21-24. 2000, pp.184-188.