Session 1526

Development of an Interdisciplinary Undergraduate Laboratory for a Course on Design and Manufacture of Surface Mount Printed Circuit Board Assemblies

Maher E. Rizkalla, Carol L. O'Loughlin, and Charles F. Yokomto,
Department of Electrical Engineering
Purdue School of Engineering and Technology at Indianapolis
Indiana University Purdue University Indianapolis

Abstract:

This paper describes a new interdisciplinary undergraduate laboratory experience that was developed for an existing electronic manufacturing course for senior electrical and mechanical engineering students. The project was partially funded by an NSF ILI grant to assist the Department of Electrical Engineering at IUPUI in upgrading the course by adding instructional laboratory materials, computer facilities, and student projects to the course. The course materials utilize UNIX software (Mentor Graphics) on Sparc and HP workstations and are detailed with the laboratory setup, including hardware and software. Student training in the use of Mentor's Board Station provides a unique experience which applies classroom manufacturing topics immediately to board design. Course and laboratory materials involving both hardware and software focus on epoxy-fiberglass boards utilizing mainly surface mount components. Student satisfaction with the reorganized course is presented. The new laboratory brings together theoretical study, laboratory design, and real product manufacturing of surface mount printed circuit assemblies as a whole. The paper also discusses the course as a model for cooperation between an education institution and manufacturing companies to provide state-of-the art technical training for senior students, and it emphasizes interdisciplinary group work for students from different disciplines.

1. Introduction:

With continued and rapid increase of usage of PCs and Workstations, electronic manufacturing has become a significant sector in manufacturing industry. Electronic production worldwide is undergoing a revolutionary change in both component manufacturing and the manufacturing techniques used in surface mount technology. With the new technologies, component size and the costs are reduced, and the reliability of electronic products is improved. Advancing the state-of-the-art in electronic manufacturing technology presents a challenge to U.S. electronic manufacturers in the highly competitive world of the 90s.

This project was partially funded by National Science Foundation, Grant # DUE-9551283

Design processes for electronic systems begin with an idea and end with a finished product. Traditionally, the designs are implemented from the bottom up, from the primitive component level, piece by piece, until the individual pieces make up the entire circuit. This process is capable of designing relatively small circuits, requiring relatively long time. With increasing design complexities and market competition, the bottom-up approach simply can not keep up with the engineer's needs to produce a working design. Modern design uses the newer top-down design methodology which offers a more effective method for meeting the demands of increasingly large and more complex environments. Top-down design is the process of capturing an idea at a high level of abstraction and implementing it from that high level, progressing down through increasing levels of detail. Electronic design automation (EDA) and computer aided design (CAD) describe software tools that aid the development of circuits and systems. EDA and CAD often refer to physical design tools for integrated circuits (IC) and printed circuit board (PCB) layouts, as well as conceptual design tools, such as schematic editors, circuit simulators, and programmable logic compilers. Mentor Graphics Corporation offers a complete suite of integrated EDA and CAD tools for the top-down design. Its software is widely used in industry, and the company has a higher education program in which IUPUI participates. The Mentor Graphics suite of software includes tools for schematic capture, functional, behavioral, and physical simulation. In addition, it provides computer aided manufacturing (CAM) files for PCB manufacture.

Course Description

The course "Electronic Manufacturing" consists of lecture material that includes ASIC design and PCB layout design emphasizing surface mount assemblies. The course taught by two instructors, gives students electronic manufacturing aspects for the design of integrated circuits as well as the printed circuit boards. Students in the class learn and actually manufacture boards and microchips. The details of the course is as follows:

Part I: ASIC Design

In this part of the course, students are introduced to the device aspects from the layout point of view. The device modeling when the channel shrinks including the secondary effects that are considerably important in this applications are covered with practical digital circuits. This material is built on introductory digital electronics classes which are prerequisites for this course. Students are required to design and generate the artwork file necessary for manufacture. Students spend five weeks on this part of the course. The materials covered in this period are:

- Historical background on the IC design that started from the invention of the point contact transistor until the most recent technology in 90s. The market share for digital, analog, BJT and MOSFET: NMOS and CMOS. Differences between both technologies in all aspects: integration, power, N.I., etc.
- The MOS transistor with considerations for short channel devices. Secondary effects such as mobility saturation with the electric field and degradation with the temperatures, hot electrons, electromigration, and aluminum spikes.
- Simple logic circuits with CMOS technology, with the use of PSpice simulation to study the circuit performance
- Introduction to IC fabrication, layout and design rules.
- Use of L-Edit on PCs

In this part of the course, some students who have been exposed to UNIX software have the option to use GDT, a layout editor in Mentor Graphics to do their project.

A typical project in this part of the course is eight bit shift register in a 40 pin package.

Part II: Printed Circuit Board Assemblies

The basic challenge in the remaining 10 weeks of the course is to teach students enough about manufacturing technology, circuit board layout design practices, and a design automation tool to provide them an overview of the electronic manufacturing process. And while the student completing the 10 weeks cannot be considered a fully functional designer, he is certainly in a position to become one much quicker than previously. The overview which the 10 weeks provides is just as helpful to students who engage in traditional electronic circuit design, component design, or some phase of manufacturing. It makes them far more aware of the influence of their work on the final product. For large boards, many companies employ concurrent design principles. That is, layout and manufacturing process design will begin before the circuit design is fully complete. The layout will proceed as simulations are run and changes are made to the circuit. This results in a significant reduction in time to market a new design. Modern software packages support such concurrent efforts, which may take place in different physical locations, and sometimes in different countries. An engineer operating in any portion of this process needs to be aware of how his work will effect the whole effort.

To provide an overview of such a complete design effort requires a first-hand application of the principles which the students need to learn. Students begin to learn how the software will use the physical characteristics of active and passive electronic component packages, even as they are studying various component styles. As they learn the fabrication processes which dictate design rules for component placement, they see how the software's autoplacement routines will lay out the circuit. The same side-by-side learning methods translates manual trace design practice into configuration and running of the autorouter. Since the software is so powerful, more time is devoted to learning how to operate it effectively than is devoted to learning the details of hand layout.

In 10 weeks it is not possible to provide insight into the details of many types of common board designs. The emphasis is placed on learning surface mount component package styles, and their placement on top and bottom sides of 2 to 6 layer fiberglass-epoxy boards. The common manufacturing processes for these boards are studied, and students will have a first hand opportunity to practice some of these processes, as described below.

3. Laboratory Components

3.1 Instructional Laboratories using Mentor Graphics Software:

Lab 1: Familiarization with the Mentor Graphics Design Architect

This lab introduces students to Mentor Graphics Design Manager and Design Architect. Students learn enough of the menu structure to enter schematic library symbols and make the connections for a simple digital circuit, and check the final layout. Many of the basic CAD operations for workplace management are learned in this exercise. These consist of selecting, moving, deleting, copying, zooming, etc. Mentor Graphics provides an extensive set of training exercises, and students use portions of these to shorten the time for learning to perform basic functions.

Lab 2: Component Placement

This lab teaches students manual component placement and repositioning. Additional CAD operations are learned along with the basic features of defining a board outline. Components can be grouped into functional blocks, e.g analog and digital subcircuits, to insure that they are physically grouped on the board by autoplacement. Some areas of the board may be designated as keep-out areas to prevent the placement of components in these areas. The graphical layer structure of Mentor Graphics software is studied to enable students to select only portions of the total design for viewing.

Lab 3: Automatic Component Placement

In this laboratory, students use auto layout to place all the components on the board. They learn how to improve the component placement to minimize the routing congestion by swapping components and pins. We previously defined a placement rule area for analog components and used the mapping technique to place the components in the area. In this exercise, students use auto placement to put the remaining components on the board. Students also learn how to evaluate the placement. Mentor Graphics generates two reports that help students determine if component placement improvement is needed to make routing easier by using the placement Histogram and the Routability Predictor.

Lab 4: Routing

In this exercise, students learn about the three types of routers: Autorouter, Dynamic Editor, and Shaperouter. The first two are used to lay out the board. The techniques for setting up the Autorouter to conform to specific layout rules for traces, pads, connections, grid, vias, etc. are learned. Typically the circuits assigned to the students are small enough to be completed routed. Again the Mentor tutorials can be used to demonstrate the software's performance for automatic routing of the large tutorial circuit. Students use the dynamic editor’s ability to complete the routing process when the autorouter does not route all the paths. Students check for design rule violations before finalizing their board layout.

Lab 5: CAM Files for the Routed Board

This lab explains the various reports needed by a board fabricator to manufacture a bare board. Here the student learns to present manufacturing information needed for component placement and board documentation. Some features will be in paint, some in copper. The artwork files for silkscreen, copper, solder paste, soldermask, etc. are generated. The drill file artwork is needed, and also output files detailing component type and placement coordinates may be needed for some pick and place component handlers. Even though these files are now commonly submitted electronically, the students will generate the paper versions of them for inclusion in their course documentation.

This laboratory takes the student through different pieces of Mentor Graphics software. It starts with the creation of a schematics using the Design Architect. This is followed by the packaging of the design using Mentor Package Tools, layout and routing, and finally creating the artwork data for manufacturing.

Design Project

In the last four weeks in the semester, students work in groups and are given a problem statement where they design an electronic system and generate the PCB layout. In this project, students use ideas from the instructional lab material used during the semester.

Sample Projects:

Students usually choose one of the projects they designed in EE266 (Digital Logic Circuits) such as a burglar alarm, traffic control circuit, or a model for an elevator control circuit Knowing the design of the circuit, will give them enough time to focus on the printed circuit board design part of the project.

3.2. Hardware

The electronic manufacturing lab consists of four Sparcstations, one HP, and two X-term HP workstations. Mentor Graphics software is resident in both Sparcstations and the HPs. The laboratory is networked and connected to Computer Network Center (CNC), the school's computer network. Students can access their files from anywhere on campus. A color HP printer is attached to these machines, and a 150 MHz Oscilloscope is available for testing.

3.3 Mobile Electronic Manufacturing Laboratory (MEML)

The MEML is housed in a 36 foot trailer; and has its home base outside the Engineering and Technology building on the IUPUI campus. Until recently, this facility was managed by the Electronic Manufacturing Productivity Facility (EMPF). The MEML was not part of the original electronic manufacturing course structure, but it was always our intention to use EMPF facilities and personnel to demonstrate basic manufacturing, testing and cleaning operations. These aspects of electronic manufacturing are available from cooperative fabricators in nearly every urban setting. We have generous cooperation from a local high-quality board fabricator which can supplement our efforts to introduce students to industrial techniques. Nevertheless, the MEML represents a nearby resource which is convenient to introduce students to actual hands-on processes.

The MEML is a mini-factory capable of manufacturing small volumes of 2 layer fiberglass-epoxy boards with surface mount components placed only on the top side. It's purpose is educational, but its capability is sophisticated enough to make small production runs. There is a solder paste stencil printer, a programmable pick and place component handler, a convection reflow oven, and an aqueous ultrasonic cleaner. There is also a soldering station for special components which cannot be reflowed effectively in the oven. This station also serves as a rework area. The MEML has existing bare boards, components, and screen printing stencils for several small gadgets, suitable for either direct fabrication by small student groups, or for demonstrations to larger groups. The MEML accommodates 5 or 6 working adults at a time.

Because of the time limit imposed by a one semester course, as well as substantial tooling costs for bare boards and a solder paste stencil, students will not be able to manufacture one of their own designs. In our view, this will only slightly diminish student satisfaction with using the MEML to gain first hand experience with board manufacturing techniques.

4. Classroom Support

Because of the exceptional laboratory experiences, much of the formal course lecture material will be chosen specifically to support the laboratory. That is, to support the intelligent use of the Mentor Graphics software, students must be instructed in component package types, board substrate materials, layout design rules, common electronic manufacturing processes, use of solder mask, clean and no-clean technologies, and testing procedures. Because of the pervasive use of tin-lead solder, some attention is given to its metallurgy and to its effect on reliable board manufacturing. A guest lecturer normally gives a comprehensive overview of solder paste technology along with a brief discussion of the developing solder-alternative technology.

Although the time limit restricts us to use one style of board, the glass-epoxy FR-4 substrate, other board styles are introduced to place this in context. The Mentor Graphics training circuit uses multiple layers to demonstrate the major features of the software, and classroom material presents the use of ground and power planes, signal planes, and the design rules for standard, blind, and buried vias in multi-layer boards. These features are handled differently in double-sided boards, with varying trace widths, and restricted power and ground trace placement.

Trace placement design guidelines are explained as background for configuring autorouter(s) and hand tracing unfinished routes. When time permits, an introduction to high frequency layout is given. Background support structure available to the students consists of the comprehensive electronic manufacturing library at the EMPF, which is located 4 blocks from campus, opportunities to hear and question guest lecturers on solder technology, reflow and wave soldering equipment, cleaning methods, component handling and testing. There are some graduate students with board layout experience who are informally available to the seniors. Taped lectures on high speed design layout and some advanced packaging techniques, such as multi-chip modules, are also available. A tour toward the end of the semester of Diversified Systems, Inc., a local board house specializing in glass-epoxy bare board manufacturing, is a very unifying experience for the students. Diversified Systems also does some original electronic design and some final board manufacture and testing.

5. Discussions

The three year project started August 1, 1995. This is the second year on which the laboratory is fully developed and ready to use. The course in its new format will be offered for the first time on campus in Fall 1997. Student assessment data will be collected, and their feedback will be used to update the course for the second offering in Fall 1998. The course was offered in experimental form, and student satisfaction data were presented in references [1, 2]. Various students have used the newly developed materials to do related projects. Some have done senior projects for the fall and summer 1996 semesters and were excited and motivated by their experiences with the equipment and software. They learned not only PCB layout, but also gained experience in computer networks and the UNIX operating system. In summary, students have reported that the course with this new format was exciting and motivating.

5. References:

[1] Maher E. Rizkalla, Carol L. O'Loughlin, Charles F. Yokomoto, and Gary Burkart, "A New Electronic Manufacturing Course for the Electrical Engineering Curriculum," IEEE Transaction on Education, November 1996.

[2] Maher E. Rizkalla, Carol L. O'Loughlin, Charles F. Yokomoto, and Gary Burkart, "An Innovative Model for Senior Level Undergraduate Engineering education in Electronic Manufacturing," Accepted for Publication in the International Journal of Applied Engineering Education.

6. Acknowledgment

We are grateful to Joseph Bouchez, a senior undergraduate electrical engineering student at IUPUI, for his assistance in the laboratory. We also thank Mr. Greg Smith, the director of the School Computer Network Center for maintaining the electronic manufacturing lab equipment in excellent condition and making it ready for students at any time.

Biography of Authors

Dr. Maher E. Rizkalla holds the rank of Professor of Electrical Engineering at IUPUI. He received his Ph.D. in electrical engineering from Case Western Reserve University, Cleveland, Ohio, in 1985. His research interests include electromagnetics, VLSI design, electronic manufacturing, and applied engineering applications. He has published over seventy papers in journals and conference proceedings in these areas. He is the recipient of two FIPSE Grants in 1995 and 1996 and many NSF educational Grants. He is also a participant in many of the industrial grants related to VLSI Signal Processing. Dr. Rizkalla received the Outstanding Teaching Award at IUPUI in 1988 and 1993, and the Professor of the Year Award at Purdue University Calumet in 1986. He is a Professional Engineer registered in the State of Indiana. He is a senior member of IEEE and a member of ASEE.

Dr. Carol L. O'Loughlin, Ph.D., P.E. She is an Associate Professor in Electrical Engineering at IUPUI. She is also the Undergraduate Coordinator for the Department. Her current interests are electronic manufacturing and solid state devices. She received the Outstanding Teaching Award several times over the years at IUPUI.

Dr. Charles F. Yokomoto holds the rank of Professor of Electrical Engineering at IUPUI. He received his Ph.D. from Purdue University in 1970. His current interests are in the area of assessment of learning outcomes, coaching, learning styles, problem solving, and personal heuristics. He has been using the Myers-Briggs Type Indicator (MBTI) in research and classroom applications. In the field of electrical engineering, his research interests are in the area of computer-aided network design, optimization, and design centering.

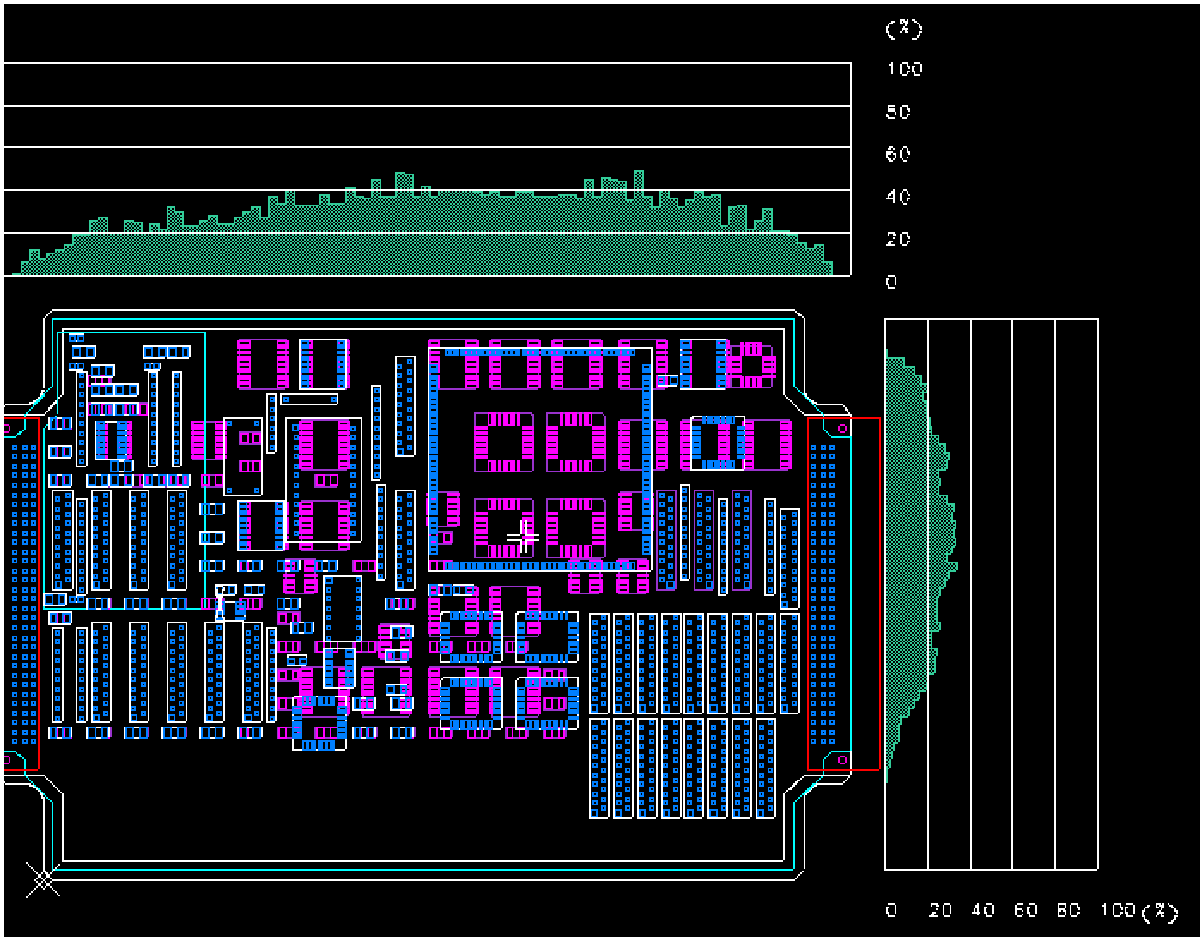

Figure 1: Component placement by auto placement

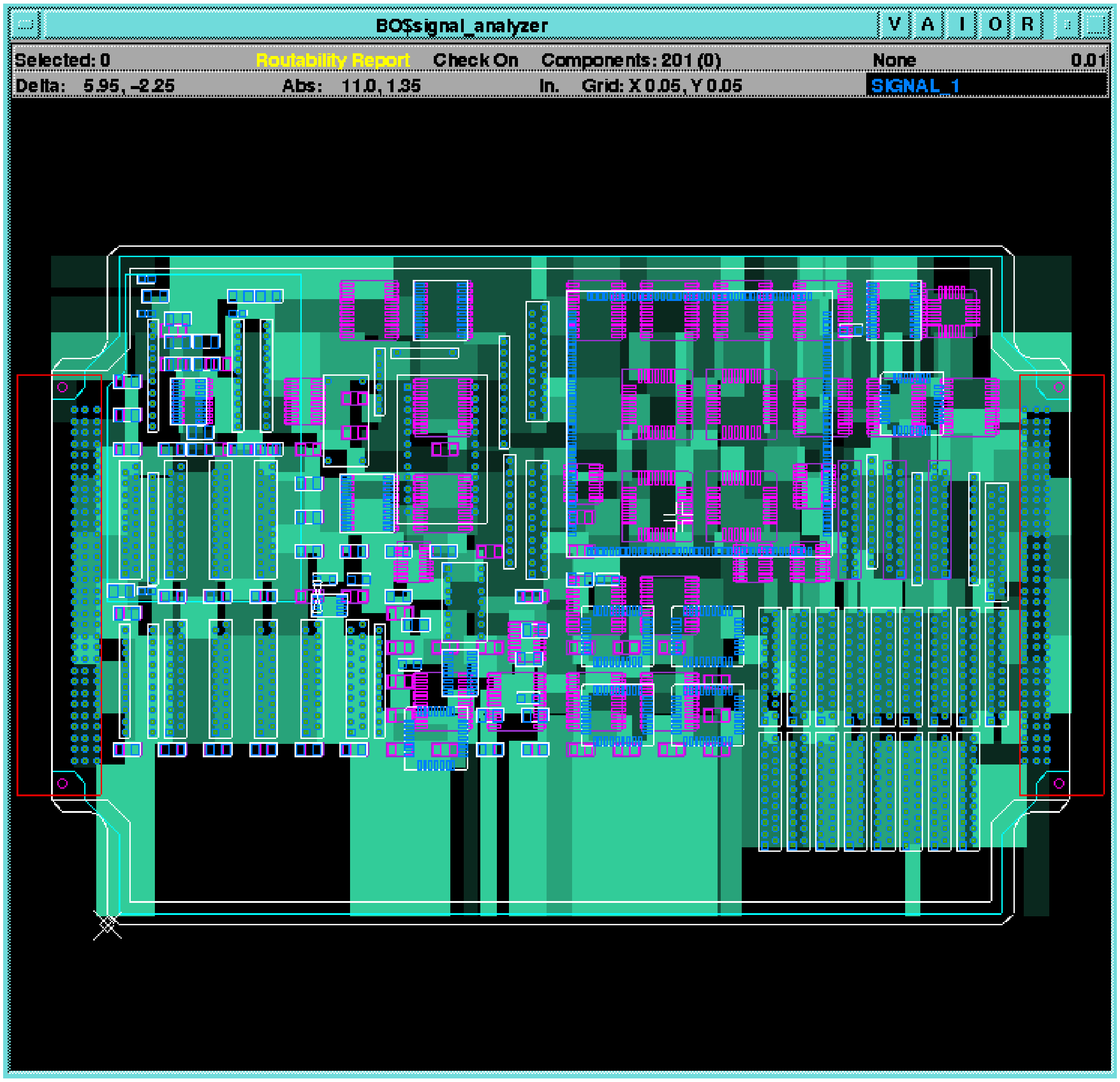

Figure 2: The PCB layout as laid by the autorouter