Professional Development Short-Courses in Electronics Engineering and Technology

- 2009 -

Courses

1. Design and Complex Characterization of High Performance PCB Structures
2. Schematic Design (OrCAD)
3. Circuit Simulation (PSpice)
4. Printed Circuit Board Layout Design (OrCAD)
5. Computer Aided Manufacturing (GerbTool) and interfacing the designer, manufacturer and assembler
6. Requirements and Characterization of Electrical and Electronics Assemblies - IPC-A-610 Certified IPC Specialist/ "Acceptability of Electronic Assemblies"
7. Requirements and Characterization of Electrical and Electronics Assemblies - IPC/EIA J-STD-001 Certified IPC Specialist/"Requirements for Soldered Electrical and Electronic Assemblies"
8. Requirements and Characterization of Electrical and Electronics Assemblies - IPC-A-600 Certified IPC Specialist/ "Acceptability of Printed Boards"
9. Requirements and Characterization of Electrical and Electronics Assemblies – IPC 7711&7721 Certified IPC Specialist/"Rework of Electronic Assemblies Repair and Modification of Printed Boards and Electronic Assemblies"

1. Design and Complex Characterization of High Performance PCB Structures

Course Contents

The course provides the participants with a systematic overview of high performance PCB design and complex characterization of PCB structures for high-speed/high-frequency/high-density applications and presents, based on numerous figures, formulas, case studies and examples, a practical approach on development, evaluation, and manufacturing of on-board interconnection structures. The eight chapters cover a large area of topics, from fundamentals of PCB design to signal integrity analysis, full-wave electromagnetic modelling and simulation, measurement techniques and CAE-CAD tools. Some Flash movies, which are destined to a better understanding of PCB design flow, are also presented.

Course Duration

3 days – 2.5 days for the course and 0.5 day for performance evaluation, technical remarks and final discussions.

Course Outline

DAY 1

1. Introduction in PCB

- Classification of PCB structures
- PCB materials – properties and influences
- Layer stack aspects
- Manufacturing ttechnologies
- Interfacing the designer and the fabricator

2. Fundamentals of high-speed/high-frequency/high-density PCB design

- Time, frequency and distance
- PCB Parasitic elements
- Lumped and distributed circuits
- Current capacity of traces

3. Transmission lines and controlled impedance

- Transmission lines and equivalent two-port networks
- Characteristic impedance control
- Topologies with controlled impedance; useful formulas
- Matching the impedance; termination technique

4. Signal Integrity Analysis

- Relative permittivity and effective permittivity
- Reflections
- Propagation delay time and length of traces
- Crosstalk and EMI
- Attenuation of signals on PCB

DAY 2

5. Electromagnetic characterization of PCB elements

- Trace segment
- Corner
- Via
- T-branch and step in width

6. Characterization of reference planes

- Power distribution
- Decoupling and bypass
- Modeling and simulation of reference planes

DAY 3

7. Measurement of PCB structures

- Introduction
- Techniques
- Practical aspects of on-board measurements

8. CAE-CAD tools for evaluation of PCB

- Tools for characteristic impedance calculation
- Tools for signal integrity analysis
- Tools for complex electromagnetic investigation

- Course summary/review
- Multiple-choice test for the theory
- Practice exam
- Instructor/student conference with technical remarks and final discussions

Course Notes

A set of course notes and various printed technical documents will be provided to each participant.

Fee

500 EUR/participant, minimum 2 participants or
1200 EUR/company, unlimited number of participants

- For independent specialists the fee can be lower, based on negotiations

Performance Evaluation

A final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one multiple-choice test for the theory and one practice exam focused on engineering aspects of the course.

Who should attend

The course is designed for electrical and electronics engineers involved in high-speed/high-frequency/high-density PCB design, professors in the field of electronic packaging, technical managers, specialists, and students who wish to get a comprehensive overview of PCB design and characterization, and to learn more about various applications. The course is focused on various practical aspects and, due to a large number of formulas and examples which can be directly applied in practice, is addressed also to R&D engineers and researchers from innovative companies, involved in development of high performance electronic products.

≈●≈

2. Schematic Design (OrCAD)

2.1 OrCAD Capture, basic level

Course Contents

The course provides the participants with an overview of schematic design and presents, based on numerous case studies and examples, a practical approach on development of diagrams from electronic products. It presents fundamentals of schematic design and parts libraries creation. Some Flash movies, which are destined to a better understanding of schematic design flow, are also presented.

Course Duration

2 days – 1.5 days for the course and 0.5 day for performance evaluation, technical remarks and final discussions.

Course Outline

DAY 1

- Introduction in CAE-CAD-CAM. Basics of computer aided design in development of electronic products. Block diagram of a high performance CAE-CAD-CAM environment.
- The Capture work environment; project manager, schematic page editor, part editor, session log, toolbar & tool palette.
- Starting a project; creating and opening projects, designs, and libraries, working with files in a project, saving projects, designs, and libraries.
- Placing, editing, and connecting parts and symbols; placing and editing parts, searching for parts, working with power and ground symbols, placing and connecting wires.
- Adding and editing graphics and text.
- Libraries and parts; categories of libraries and parts. Creating and editing parts.
- Creating simple & complex designs; flat designs, hierarchical designs, concatenated (multi-page) designs.

DAY 2

- Preparing a SCM design for post-processing; annotating, visual checking, manual checking of a design.
- Basics of SCM post-processing.
- Interfacing Capture with Layout, SCM - PCB transfer, procedure and files.
- Optimizing the transfer based on ECO technique.
- Preparing the technical documentation of an electronic project.

- Course summary/review
- Multiple-choice test for the theory
- Practice exam
- Instructor/student conference with technical remarks and final discussions

Course Notes

A set of course notes and various printed technical documents will be provided to each participant.

Fee

200 EUR/participant, minimum 2 participants or
500 EUR/company, unlimited number of participants

- For independent specialists the fee can be lower, based on negotiations.

Performance Evaluation

A final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one multiple-choice test for the theory and one practice exam focused on engineering aspects of the course.

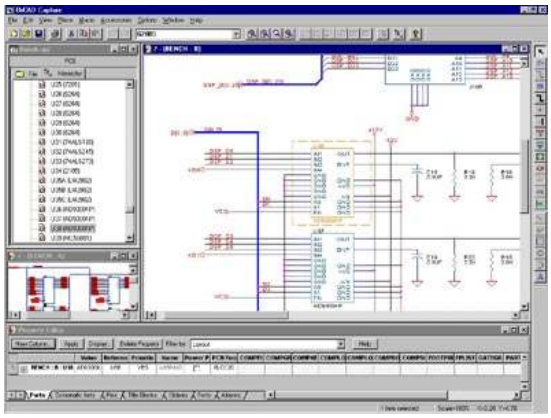
Who should attend

The course is designed for electrical and electronics managers and leaders involved in supervising CAD schematic and PCB design, engineers, professors in the field of electronic packaging, and students who wish to get an overview of schematic design and to learn more about the design flow. The course is focused on various practical aspects and, due to a large number of examples which can be directly applied in practice, is addressed to people who want to learn quickly the basics of CAD schematic design.

≈●≈

2.2 OrCAD

OrCAD Capture, standard level



Course Contents

The course provides the participants with a systematic overview of schematic design and presents, based on numerous figures, case studies and examples, a practical approach on development of diagrams/schematics from electronic products. The chapters cover a large area of topics, from fundamentals of schematic design to complex hierarchical and multi-page projects. Some Flash movies, which are destined to a better understanding of schematic design flow, are also presented.

Course Duration

3 days – 2.5 days for the course and 0.5 day for performance evaluation, technical remarks and final discussions.

Course Outline

DAY 1

- Introduction in CAE-CAD-CAM. Basics of computer aided design in development of electronic products. Block diagram of high performance CAE-CAD-CAM environments.
- The Capture work environment; project manager, schematic page editor, part editor, session log, toolbar & tool palette.
- Starting a project; creating and opening projects, designs, and libraries, working with files in a project, saving projects, designs, and libraries.
- Setting up a project; defining preferences, setting up project template, defining title block information.
- Placing, editing, and connecting parts and symbols; placing and editing parts, searching for parts, working with power and ground symbols, placing and connecting wires and buses. Power & Ground (PWR & GND) of electronic circuits. Working with net alias and signals (underground/invisible connections).
- Adding and editing graphics and text; drawing lines, rectangles and squares, circles and ellipses, working with pictures, management of texts.

DAY 2

- Libraries and parts; categories of libraries and parts, part instances and occurrences, design cache. Creating and editing parts; parts and packages: homogeneous or

heterogeneous, creating a new part, defining a part, power and ground terminals, invisible power pins, checking and saving a part, editing an existing part.
- Creating simple & complex designs; flat designs, hierarchical designs, concatenated (multi-page) designs, hierarchical blocks, ports, and pins, off-page connectors.
- Preparing a SCM design for post-processing; annotating, visual checking, manual checking of a design.
- SCM post-processing activities.
- Checking for design rules violations – DRC & ERC.
- Creating netlists for various purposes.
- Creating reports: bill of materials, cross reference, wirelist; printing a design.

DAY 3

- Interfacing Capture with Layout, SCM - PCB transfer; preparing a Capture design for use with Layout, creating a netlist for interfacing with Layout, general rules for a proper matching between SCM and PCB environments, working with PCB footprints for SCM parts, technological viewpoints.
- Using the Component Information System (CIS); basics of CIS, CIS work environment, using the CIS interface.

- Course summary/review
- Multiple-choice test for the theory
- Practice exam
- Instructor/student conference with technical remarks and final discussions

Course Notes

A set of course notes and various printed technical documents will be provided to each participant.

Fee

350 EUR/participant, minimum 2 participants or
800 EUR/company, unlimited number of participants

- For independent specialists the fee can be lower, based on negotiations.

Performance Evaluation

A final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one multiple-choice test for the theory and one practice exam focused on engineering aspects of the course.

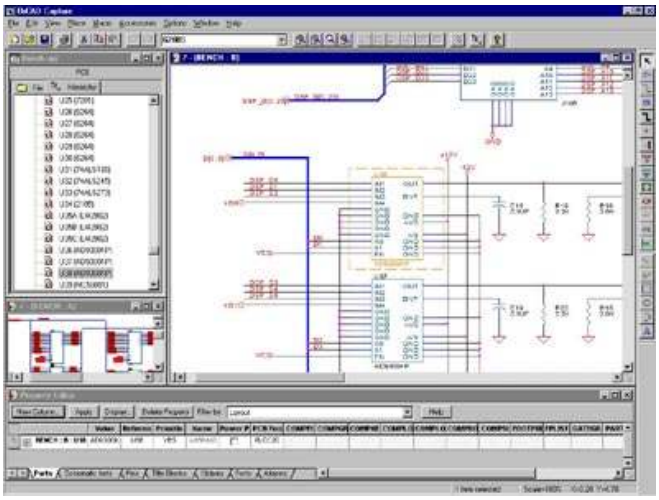
Who should attend

The course is designed for electrical and electronics engineers involved in schematic and PCB design, professors in the field of electronic packaging, technical managers, and students who wish to get a comprehensive overview of schematic design of electronic projects and to learn more about various applications. The course is focused on various practical aspects and, due to a large number of examples and case studies which can be directly applied in practice, is addressed also to R&D engineers and researchers from innovative companies involved in development of high performance electronic products.

≈●≈

2.3 OrCAD

OrCAD Capture, advanced level



Course Contents

The course provides the participants with a systematic overview of schematic design and presents, based on numerous figures, case studies and examples, a practical approach on development of diagrams from electronic products. The chapters cover a large area of topics, from fundamentals of schematic design to complex hierarchical and multi-page projects. Some Flash movies, which are destined to a better understanding of schematic design flow, are also presented. Additionally, high level topics, as aspects of technology, CIS, ECO, ITC, etc., are discussed.

Course Duration

4 days – 3 days for the course and 1 day for performance evaluation, technical remarks and final discussions.

Course Outline

DAY 1

- Introduction in CAE-CAD-CAM. Basics of computer aided design in development of electronic products. Block diagram of a high performance CAE-CAD-CAM environment. Integrated solutions for electronics industry.
- The Capture work environment; project manager, schematic page editor, part editor, session log, toolbar & tool palette.
- Starting a project; creating and opening projects, designs, and libraries, working with files in a project, saving projects, designs, and libraries, archiving a project.
- Setting up a project; defining preferences, setting up project template, defining title block information, creating a custom title block.
- Placing, editing, and connecting parts and symbols; placing and editing parts, searching for parts, working with power and ground symbols, placing and connecting wires and buses. Power & Ground (PWR & GND) of electronic circuits. Working with net alias and signals (underground/invisible connections). Using macros; Recording, playing, configuring a macro.

- Adding and editing graphics and text; drawing lines, rectangles and squares, circles and ellipses, working with pictures, management of texts.

DAY 2

- Libraries and parts; categories of libraries and parts, part instances and occurrences, design cache, primitive and non-primitive parts. Creating and editing parts; parts and packages: homogeneous or heterogeneous, creating a new part, defining a part, power and ground terminals, invisible power pins, checking and saving a part, editing an existing part, editing a part in a library, editing a part on a schematic page.
- Creating simple & complex designs; flat designs, hierarchical designs, concatenated (multi-page) designs, hierarchical blocks, ports, and pins, off-page connectors.
- Preparing a SCM design for post-processing; annotating, visual checking, manual checking of a design.
- SCM post-processing activities;
- Checking for design rules violations – DRC & ERC.
- Back annotating, re-annotating.
- Creating netlists for various purposes.
- Creating reports; bill of materials, cross reference, wirelist.
- Printing a design.

DAY 3

- Interfacing Capture with Layout, SCM - PCB transfer; preparing a Capture design for use with Layout, creating a netlist for interfacing with Layout, enabling inter-tool communication between Capture and Layout, general rules for a proper matching between SCM and PCB environments, working with PCB footprints for SCM parts, THD and SMD, technological viewpoints.
- Using the Component Information System (CIS); basics of CIS, CIS work environment, CIS explorer window, working with database parts, using the CIS interface, placing a database part on a schematic page, browsing part properties, linking a placed part to a database part.

DAY 4

- Course summary/review
- Multiple-choice test for the theory
- Practice exam
- Instructor/student conference with technical remarks and final discussions

Course Notes

A set of course notes and various printed technical documents will be provided to each participant.

Fee

500 EUR/participant, minimum 2 participants or
1200 EUR/company, unlimited number of participants

- For independent specialists the fee can be lower, based on negotiations.

Performance Evaluation

A final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one multiple-choice test for the theory and one practice exam focused on engineering aspects of the course.

Who should attend

The course is designed for electrical and electronics engineers involved in schematic and PCB design, specialists, professors in the field of electronic packaging, technical managers, and students who wish to get a comprehensive overview of schematic design of electronic projects and to learn more about various applications. The course is focused on various practical aspects and, due to a large number of examples and case studies which can be directly applied in practice, is addressed also to R&D engineers and researchers from innovative companies involved in development of high performance electronic products.

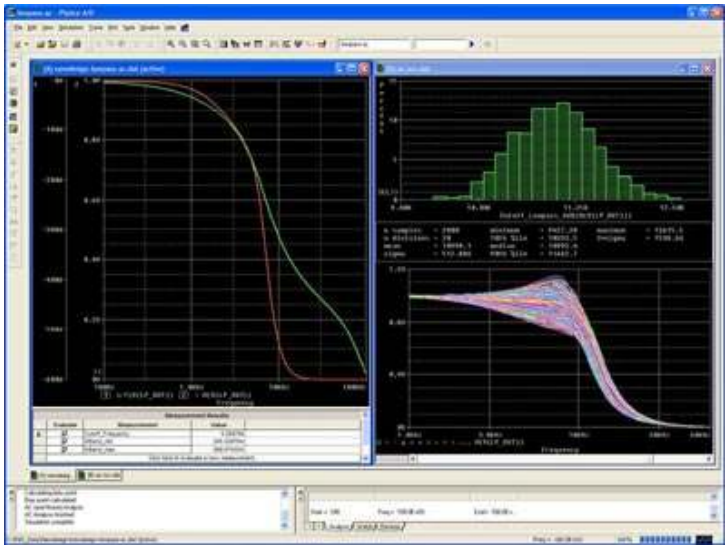
≈●≈

3. Circuit Simulation

(PSpice)

3.1 OrCAD

OrCAD PSpice, basic level



Course Contents

The course provides the participants with an overview of schematic design and circuit simulation and presents, based on numerous case studies and examples, a practical approach on development and optimizing of electronic projects based on circuit simulation. It presents fundamentals of schematic design and circuit simulation. Some Flash movies, which are destined to a better understanding of simulation flow, are also presented.

Course Duration

2 days – 1.5 days for the course and 0.5 day for performance evaluation, remarks and final discussions.

Course Outline

- Setup & management of OrCAD PSpice projects.
- Interfacing OrCAD Capture & OrCAD PSpice.
- Signal and power supply sources: types and fields of application.
- Introduction to Probe (traces, markers, zoom of signals, cursors, labelling & marking).
- DC analysis.
- Basics of time domain (transient) analysis.

- Basics of AC analysis.

DAY 2

- Parametric analysis & temperature analysis.
- Basics of digital simulation (sources, signals, digital devices and circuit analysis).

- Course summary/review
- Multiple-choice test for the theory
- Practice exam
- Instructor/student conference with technical remarks and final discussions

Course Notes

A set of course notes and various printed technical documents will be provided to each participant.

Fee

200 EUR/participant, minimum 2 participants or
500 EUR/company, unlimited number of participants

- For independent specialists the fee can be lower, based on negotiations.

Performance Evaluation

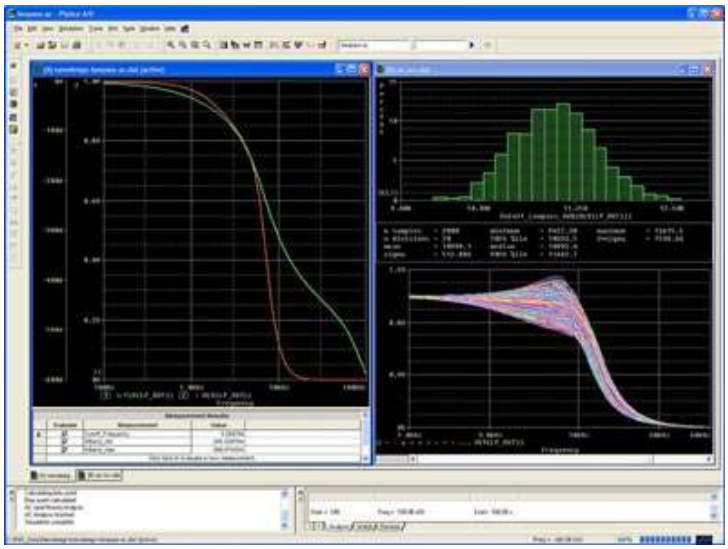
A final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one multiple-choice test for the theory and one practice exam focused on engineering aspects of the course.

Who should attend

The course is designed for electrical and electronics managers and leaders involved in supervising schematic and PCB design, engineers, professors in the field of electronic packaging, and students who wish to get an overview of schematic design and to learn more about the design flow. The course is focused on various practical aspects and, due to a large number of examples which can be directly applied in practice, is addressed to people who want to learn quickly the basics of PCB design.

≈●≈

3.2 OrCAD PSpice, standard level



Course Contents

The course provides the participants with a systematic overview of schematic design and circuit simulation and presents, based on numerous figures, case studies and examples, a practical approach on development and optimizing of electronic projects based on PSpice circuit simulation. The chapters cover a large area of topics, from fundamentals of schematic design to simple or complex simulations, based on various practical projects. Some Flash movies, which are destined to a better understanding of schematic design flow and PSpice simulation, are also presented.

Course Duration

3 days – 2.5 days for the course and 0.5 day for performance evaluation, remarks and discussions.

Course Outline

DAY 1

- Setup & management of OrCAD PSpice Projects.
- Advanced interfacing between OrCAD Capture & OrCAD PSpice.
- Signal and power supply sources: types and fields of application.
- Introduction to Probe (traces, markers, zoom of signals, cursors, labelling & marking).
- DC analysis: ► bias point (passive & active structures) and ►DC sweep (I-V characteristic, DC transfer curves, temperature analysis).
- Time domain (transient) analysis (sources, maximum step size, convergence, circuits with BJTs, circuits with integrated circuits).

DAY 2

- AC sweep analysis (magnitude and phase, Bode plots, amplifier gain analysis).
- Parametric analysis (variable devices, maximum power transfer, parametric sweep).
- Temperature analysis (variable working temperatures, temperature sweep).
- Working with the Stimulus Editor.

DAY 3

- Editing models (changing the reference, modifying existing models, downloading models from manufacturers).
- Digital simulations (sources, signals, digital devices and circuits analysis).

- Course summary/review
- Multiple-choice test for the theory
- Practice exam
- Instructor/student conference with technical remarks and final discussions

Course Notes

A set of course notes and various printed technical documents will be provided to each participant.

Fee

350 EUR/participant, minimum 2 participants or
800 EUR/company, unlimited number of participants

- For independent specialists the fee can be lower, based on negotiations.

Performance Evaluation

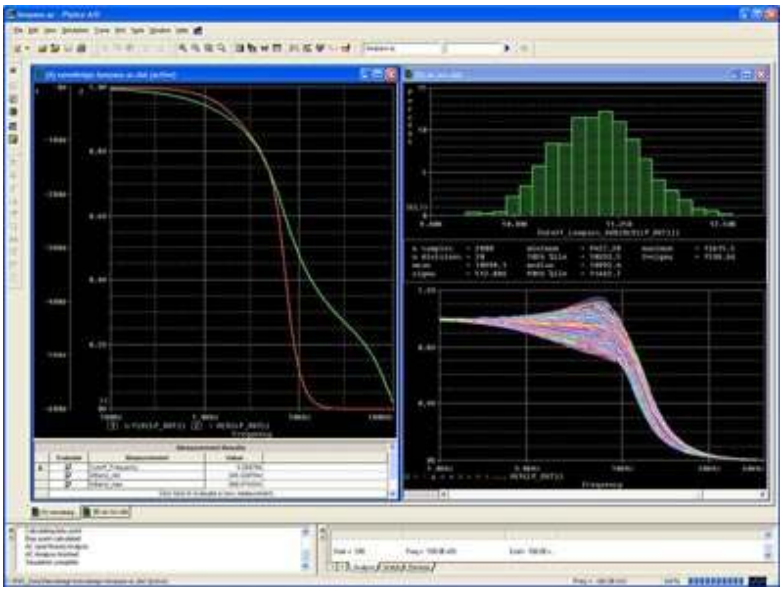
A final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one multiple-choice test for the theory and one practice exam focused on engineering aspects of the course.

Who should attend

The course is designed for electrical and electronics engineers involved in schematic and PCB design, professors in the field of electronic packaging, technical managers, and students who wish to get a comprehensive overview of schematic design of electronic projects and to learn more about various applications. The course is focused on various practical aspects and, due to a large number of examples and case studies which can be directly applied in practice, is addressed also to R&D engineers and researchers from innovative companies involved in development of high performance electronic products.

≈●≈

3.3 OrCAD PSpice, advanced level



Course Contents

The course provides the participants with a systematic overview of schematic design and circuit simulation and presents, based on numerous figures, case studies and examples, a practical approach on development and optimizing of electronic projects based on PSpice circuit simulation. The chapters cover a large area of topics, from fundamentals of schematic design to simple or complex simulations, based on various practical projects. Some Flash movies, which are destined to a better understanding of schematic design flow and PSpice simulation, are also presented. Additionally, deeper investigations on mixed A/D projects, convergence problem, analogue behavioural modelling of electronic and non-electronic devices, and models creation, are highlighted.

Course Duration

4 days - 3 days for the course and 1 day for performance evaluation, remarks and discussions.

Course Outline

DAY 1

- Setup & management of OrCAD PSpice Projects.
- Advanced interfacing between OrCAD Capture & OrCAD PSpice.
- Signal and power supply sources: types and fields of application.
- Introduction to Probe (traces, markers, zoom of signals, cursors, labelling & marking).
- DC analysis: ► bias point (passive & active structures) and ►DC sweep (I-V characteristic, DC transfer curves, temperature analysis).
- AC sweep analysis (magnitude and phase, Bode plots, amplifier gain analysis, amplifier bandwidth).
- Monte Carlo analysis.

DAY 2

- Time domain (transient) analysis (sources, maximum step size, convergence, circuits with BJTs, circuits with integrated circuits, advanced TD simulations).
- Parametric analysis (variable devices, maximum power transfer, parametric sweep).
- Temperature analysis (variable working temperatures, temperature sweep).
- Working with the Stimulus Editor.
- Editing models (changing the reference, modifying existing models, downloading models from manufacturers).
- Digital simulations (sources, signals, digital devices and circuits analysis).

DAY 3

- Simulation of mixed analogue-digital circuits; specific aspects for investigation of a mixed schematic.
- High performance modelling - ABM (analogue behavioural modelling) of electronic or non-electronic devices/projects.

DAY 4

- Course summary/review
- Multiple-choice test for the theory
- Practice exam
- Instructor/student conference with technical remarks and final discussions

Course Notes

A set of course notes and various printed technical documents will be provided to each participant.

Fee

500 EUR/participant, minimum 2 participants or
1200 EUR/company, unlimited number of participants

- For independent specialists the fee can be lower, based on negotiations.

Performance Evaluation

One final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one multiple-choice test for the theory and one practice exam focused on engineering aspects of the course.

Who should attend

The course is designed for electrical and electronics engineers involved in schematic and PCB design, professors in the field of electronic packaging, technical managers, and students who wish to get a comprehensive overview of schematic design of electronic projects and to learn more about various applications. The course is focused on various practical aspects and, due to a large number of examples and case studies which can be directly applied in practice, is addressed also to R&D engineers and researchers from innovative companies involved in development of high performance electronic products.

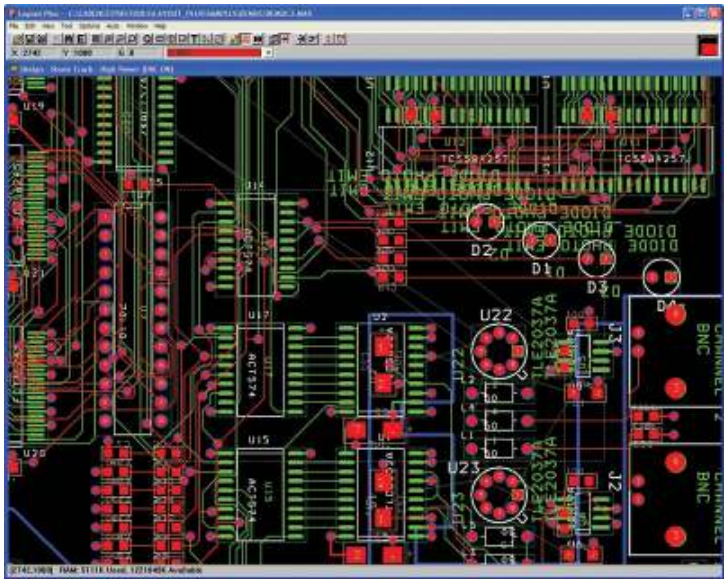
≈●≈

4. Printed Circuit Board

Layout Design (OrCAD)

4.1 OrCAD

OrCAD Layout, basic level



Course Contents

The course provides the participants with an overview of PCB design and presents, based on numerous case studies and examples, a practical approach on development of on-board interconnection structures. It presents fundamentals of PCB design and footprint libraries creation. A Flash movie, which is destined to a better understanding of PCB design flow, is also presented.

Course Duration

2 days – 1.5 days for the course and 0.5 day for performance evaluation, remarks and discussions.

Course Outline

DAY 1

- Introduction in OrCAD PCB Layout. The Layout design flow and design environment. PCB projects.
- Import of SCM netlists; ECO procedure.
- Fundamentals of CAD printed circuit board design: setting up the board; creating of board outline; placing of components; routing the board.

- Basics of PCB footprints creation.
- Methods for components placement: manual, interactive and automatic.
- Methods for conductive tracks routing: manual, interactive and automatic.

DAY 2

- Optimizing the printed circuit structure.
- Final operations and checking of PCB layout.

- Course summary/review
- Multiple-choice test for the theory
- Practice exam
- Instructor/student conference with technical remarks and final discussions

Course Notes

A set of course notes and various printed technical documents will be provided to each participant.

Fee

200 EUR/participant, minimum 2 participants or
500 EUR/company, unlimited number of participants

- For independent specialists the fee can be lower, based on negotiations.

Performance Evaluation

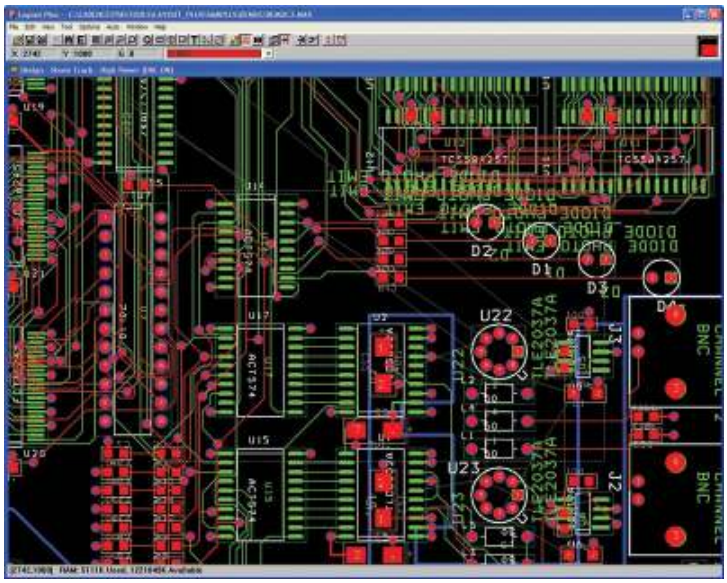
A final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one multiple-choice test for the theory and one practice exam focused on engineering aspects of the course.

Who should attend

The course is designed for electrical and electronics managers and leaders involved in supervising PCB design, engineers, professors in the field of electronic packaging, and students who wish to get an overview of PCB design and to learn more about the design flow. The course is focused on various practical aspects and, due to a large number of examples which can be directly applied in practice, is addressed to people who want to learn quickly the basics of PCB design.

≈●≈

4.2 OrCAD Layout, standard level



Course Contents

The course provides the participants with a systematic overview of PCB design and presents, based on numerous figures, formulas, case studies and examples, a practical approach on development and manufacturing of on-board interconnection structures. The chapters cover a large area of topics, from fundamentals of PCB design to signal integrity analysis and CAE-CAD tools. Some Flash movies, which are destined to a better understanding of PCB design flow, are also presented.

Course Duration

3 days – 2.5 days for the course and 0.5 day for performance evaluation, remarks and discussions.

Course Outline

DAY 1

- The OrCAD Layout design flow and design environment.
- Setup & management of OrCAD PCB projects.
- Import of SCM netlists; ECO procedure. Problems during SCM-PCB transfer process; solving the interface problems.
- Fundamentals of printed circuit board design: creating and setting up the board; creating and editing various types of obstacles; placing of components; routing the board.
- Using spreadsheets to manage design data and rules; the importance of „spreadsheet" menu in high performance/high quality PCB design.

DAY 2

- PCB footprint libraries. Types of libraries. Managing footprint libraries.
- Creating and editing PCB footprints; THD & SMD footprints; specific design aspects.
- Placement of components: manual, interactive and automatic.

- Inter-tool communication – a method of high performance CAD design.
- Optimization of PCB connections – „pin and gate swap" procedure. Back-annotation to Capture.
- Introduction in signal integrity analysis.
- Routing of PCB interconnection structure and generation of conductive tracks: manual, interactive and automatic. Grid-based and grid-less (shape-based) auto-router engines.

DAY 3

- Optimizing the printed circuit structure. Using thermal relief items and copper pour zones.
- Finishing and checking of PCB projects. Ensuring manufacturability. DFM methods.
- PCB post-processing for documentation and manufacturing.

- Course summary/review
- Multiple-choice test for the theory
- Practice exam
- Instructor/student conference with technical remarks and final discussions

Course Notes

A set of course notes and various printed technical documents will be provided to each participant.

Fee

350 EUR/participant, minimum 2 participants or
800 EUR/company, unlimited number of participants

- For independent specialists the fee can be lower, based on negotiations.

Performance Evaluation

A final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one multiple-choice test for the theory and one practice exam focused on engineering aspects of the course.

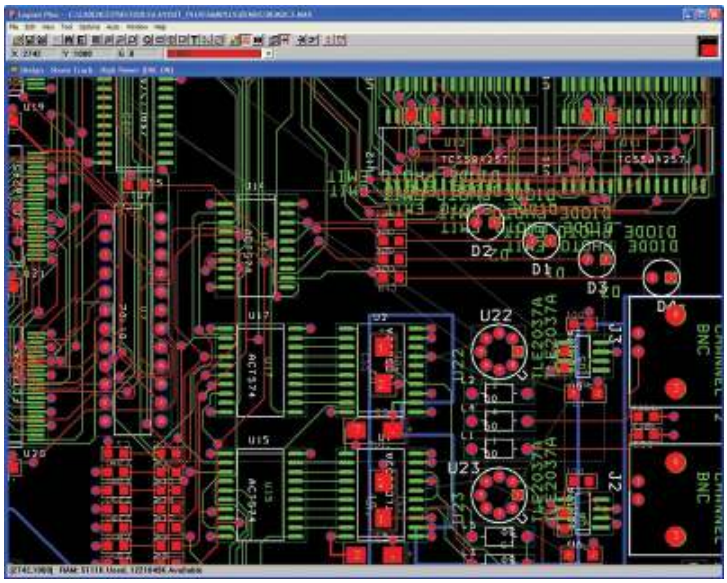
Who should attend

The course is designed for electrical and electronics engineers involved in PCB design, professors in the field of electronic packaging, technical managers, and students who wish to get a comprehensive overview of PCB design and characterization, and to learn more about various applications. The course is focused on various practical aspects and, due to a large number of formulas and examples which can be directly applied in practice, is addressed also to R&D engineers and researchers from innovative companies involved in development of high performance electronic products.

≈●≈

4.3 OrCAD

OrCAD Layout, advanced level



Course Contents

The course provides the participants with a systematic presentation of PCB design aspects and offers, based on numerous figures, formulas, case studies and examples, a practical approach on development and manufacturing of on-board interconnection structures. The chapters cover a large area of topics, from fundamentals of PCB design to technology, signal integrity analysis and CAE-CAD tools. Some Flash movies, which are destined to a better understanding of PCB design flow, are also presented. Additionally, various errors and mistakes found in practical projects, which often appear due to “time to market” pressure are explained and discussed.

Course Duration

4 days - 3 days for the course and 1 day for performance evaluation, remarks and discussions.

Course Outline

DAY 1

- The OrCAD Layout design flow and design environment. Interfacing OrCAD Layout with other blocks and CAD-CAM software systems.
- Design and manufacturing standards. Importance of standardization in CAD design. Management of OrCAD PCB projects based on standards.
- Import of SCM netlists; ECO procedure. Problems during SCM-PCB transfer process; solving the interface problems. Case studies based on practical projects.
- Fundamentals of printed circuit board design: creating and setting up the board; creating and editing various types of obstacles; placing of components; routing the board. Case study: low complexity PCB layout design without netlist import.
- Using spreadsheets to manage design data and rules; the importance of „spreadsheet” menu in high performance/high quality PCB design. Case studies based on practical projects.

DAY 2

- PCB footprint libraries. Types of libraries. Managing footprint libraries.
- Creating and editing PCB footprints; THD & SMD footprints; specific design aspects based on standards. Advanced topics in libraries/footprints management.
- Placement of components: manual, interactive and automatic. Introducing in auto-placement. Using and optimizing auto-placement. Advantages and disadvantages.
- Optimization of PCB connections – „pin and gate swap" procedure. Advanced issues: manual vs. auto swapping. Back-annotation to Capture. Cross-probing in OrCAD environment.
- Introduction in signal integrity analysis and electromagnetic compatibility at board level.

DAY 3

- Routing of PCB interconnection structure and generation of conductive tracks: manual, interactive and automatic. Theory of auto-routing. Auto-routing vs. interactive routing. Using routing strategy files. Solving routing problems.
- Grid-based and grid-less (shape-based) auto-router engines. Smart Route and Specctra stand-alone routers. The work environment. Opening and saving designs. Setting up the board for routing. Auto-routing and batch routing of PCB. Editing the board.
- Optimizing the printed circuit structure.
- Reference planes - total planes, partial planes. Using thermal relief items and copper pour areas.
- Finishing and checking of PCB projects. Ensuring manufacturability. DFM methods. Case studies based on practical projects.
- Advanced PCB post-processing for documentation and manufacturing. Interfacing the designer with the manufacturer.
- Introduction in CAM systems. Gerbtool environment. Configuration, basic operations, performance tips, menus and commands. Aperture Conversion Rule files. Standard Gerber vs. extended Gerber formats. Excellon vs. Sieb & Mayer drilling formats.

DAY 4

- Course summary/review
- Multiple-choice test for the theory
- Practice exam
- Instructor/student conference with technical remarks and final discussions

Course Notes

A set of course notes and various printed technical documents will be provided to each participant.

Fee

500 EUR/participant, minimum 2 participants or
1200 EUR/company, unlimited number of participants

- For independent specialists the fee can be lower, based on negotiations.

Performance Evaluation

A final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one multiple-choice test for the theory and one practice exam focused on engineering aspects of the course.

Who should attend

The course is designed for electrical and electronics engineers involved in PCB design, professors in the field of electronic packaging, technical managers, and students who wish to get a comprehensive overview of PCB design and characterization, and to learn more about various applications. The course is focused on various practical aspects and, due to a large number of formulas and examples which can be directly applied in practice, is addressed also to R&D engineers and researchers from innovative companies involved in development of high performance electronic products.

≈●≈

5. Computer Aided Manufacturing (GerbTool) and interfacing the designer, manufacturer and assembler

Course Contents

The course provides the participants with a systematic overview of CAM processing and presents, based on numerous figures, case studies and examples, a practical approach on CAM activities and manufacturing of PCBs. The chapters cover a large area of topics, from fundamentals of CAM and CAM tools to standards and formats for manufacturing of PCB layouts. A few Flash movies, which are destined to a better understanding of CAM activities and related topics, are also presented.

Course Duration

3 days – 2.5 days for the course and 0.5 day for performance evaluation, remarks and discussions.

Course Outline

DAY 1

- Finishing and checking of PCB projects. Ensuring manufacturability. DFM methods.
- PCB post-processing for documentation and manufacturing.
- Basics of manufacturing formats and standards. Apertures and D-codes, G-codes, M-codes. Description of the photo-plotter equipment and its actions. Gerber file contents. Aperture lists. Example of manufacturing files based on RS 274-D.
- RS 274-X standard. Benefits of RS 274-X format. Embedding format info in the 274-X header. 274-X and Apertures. The basic aperture definitions. Custom apertures. Example of manufacturing files based on RS 274-X. Polarity. Step and Repeat. Mirror Image, Downside to RS274-X.
- Excellon format for drilling the PCB. Advantages and drawbacks. Tool (drill bit) lists: standard drill, extended drill lists. Sieb & Mayer vs. Excellon.
- Gerber files and Excellon files for manufacturing of PCBs. Specific topics at the CAD-CAM interface.

DAY 2

- Gerbtool software system for advanced CAM activities
- Gerbtool user interface. Gerbtool window. Title & menu bars. Toolbar button reference. Layer bar. The Navigator. Data & commands tabs. XY, aerial & colour bars.

- Selection Filter. Item Properties Display. Status Bar.
- Opening/Importing & Saving/Exporting of various CAD files.
- Working in workspace.
- Preparing a PCB project for manufacturing. Panelisation and advanced actions for smoothing the link to manufacture equipment.

DAY 3

- Creating a new CAM design. Development of PCB layouts directly in Gerbtool.
- Special RF and microwave structures created in Gerbtool
- Interfacing the designer, the manufacturer and the assembler, practical and important aspects for obtaining high-quality projects.

- Course summary/review
- Multiple-choice test for the theory
- Practice exam
- Instructor/student conference with technical remarks and final discussions

Course Notes

A set of course notes and various printed technical documents will be provided to each participant.

Fee

350 EUR/participant, minimum 2 participants or
800 EUR/company, unlimited number of participants

- For independent specialists the fee can be lower, based on negotiations.

Performance Evaluation

A final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one multiple-choice test for the theory and one practice exam focused on engineering aspects of the course.

Who should attend

The course is designed for electrical and electronics engineers involved in PCB design and CAM activities, manufacturing professionals and specialists from printed circuits industry, professors in the field of electronic packaging, technical managers, and students who wish to get a comprehensive overview of CAM activities and CAM tools. The course is focused on various practical aspects and, due to a large number of examples which can be directly applied in practice, is addressed also to R&D engineers and researchers from innovative companies involved in development of high performance electronic products, who would like to have a solid background in the field of CAD-CAM interface and preparing files for PCB manufacturing.

≈●≈

Requirements and Characterization of Electrical and Electronics Assemblies

6. Certified IPC Specialist in IPC-A-610 "Acceptability of Electronic Assemblies"

Course Contents

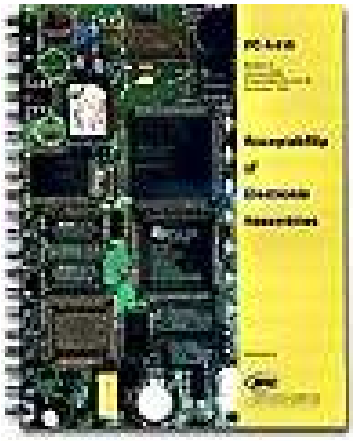
The courses are based on Association Connecting Electronics Industries (IPC) training programs (see http://www.ipc.org/). These programs started in 1995 when U.S. Department of Defense certification programs, which focused on relative small users groups, were canceled. Since then, IPC developed an industry traceable training and certification program for electronic assembly acceptance. These training programs provides with a systematic overview requirements, criteria for good solder joints, the goal been to produce electronics assemblies with best ratio performance/costs.

Course Duration

3 days

Course Outline

The IPC-A-610 is the most widely used specification for post assembly inspection. It contains visual accept/reject criteria examples for all three classes of production. Learn to utilize the IPC-A-610 document to ensure quality of your PCB assemblies.



Topics include:

- Handling electronic assemblies
- Inspecting soldered connections for terminal and plated through-holes with Pb and Pb-Free alloys
- Identifying fractured solder due to cutting
- Understanding ECO jumper wire practices
- Addressing mechanical assembly requirements

- Recognizing acceptable surface mounting for chip & cylindrical components, leadless chip carriers, & leaded components with Pb and Pb-Free alloys

DAY 1

- Introduction
- Terms and Definitions
- Handling Electronic Assemblies
- Mechanical Assemblies
- Component Installation

DAY 2

- Soldering
- Cleanliness
- Marking
- Coatings
- Laminate Conditions
- Discrete Wiring Assemblies
- Surface Mount Assemblies

DAY 3 (ONE HALF DAY)

- Course Summary/Review
- Open Book Examination
- Closed Book Examination
- Instructor/Student Conference

Course Notes

Everyone who successfully completes the Certification examination with at least 70% average grade will receive:

- IPC-A-610 Standard
- IPC-A-610 Student Handbook
- IPC Certificate of Training

Fee

400 EUR/participant (the fee includes all original documents from IPC).
Maximum number of participants is 15.

Performance Evaluation

One final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one Open Book Examination and one Closed Book examination.

Who should attend

This course is for anyone responsible for the quality and reliability of electronic assemblies. This includes engineers, quality supervisors, inspectors and manufacturing personnel responsible for quality assurance.

≈●≈

7. Certified IPC Specialist in IPC/EIA J-STD-001 "Requirements for Soldered Electrical and Electronic Assemblies"

Course Contents

The J-STD-001 is the authority for electronics assembly manufacturing. It has been adopted by the U.S. Department of Defense to replace Mil-Std-2000. The standard describes materials, methods and verification criteria for producing high quality soldered interconnections. It emphasizes process control and sets industry-consensus requirements for a broad range of electronic products. Learn assembly procedures for Wire and Terminals, Through-Hole and Surface Mount technology. Students will demonstrate soldering skills and deliver a brief presentation along with completing open and closed book exams. Customize this course by selecting only the days/modules that meet your training requirements.

Course Duration

5 days

Course Outline

Topics include:

- Soldering Wire/Terminal Connections
- Soldering Through-Hole Components
- Soldering SMT Components, including Fine Pitch
- Selecting and using the proper tools. Understanding industry terminology
- Recognizing acceptable criteria for all solder connections

- Selecting solder, flux and solder paste
- Applying accepted cleaning requirements

DAY 1 - Module 1

Overview of J-STD-001

Students will learn the requirements of J-STD-001 and related standards as they apply to operators and inspectors involved in the assembly of products to the requirements of J-STD-001. Module 1 is a prerequisite to all other modules.

- Course Overview
- Safety
- EOS/ESD
- Classes of Equipment
- Solder Theory
- Solderability
- Solder Flux and Solder Alloys
- Facilities, Tools & Equipment
- Training
- PTH - Assembly/Solder
- Surface Mount Technology
- Cleaning
- Module 1 Review
- Module 1 Examination

DAY 2 - Module 2

Wires & Terminals

Students will learn the requirements of J-STD-001, and demonstrate the skills for stripping and tinning wire and hand soldering wires of different gauges to various types of commonly used solder terminals.

- Wire Preparation
- Solder to Terminals
- Terminal Inspection
- Wire & Terminal Demonstration
- Wire & Terminal Lab
- Module 2 Review
- Module 2 Examination

DAY 3 - Module 3

Through-Hole Technology

Students will learn the requirements of J-STD-001, and demonstrate the skills for preparing and mounting Through-Hole components to PWBs.

- Lead Preparation
- Component Mounting
- PTH Inspection Criteria
- PTH Soldering Demonstration
- PTH Lab
- Module 3 Review
- Module 3 Examination

DAY 4 - Module 4

Surface Mount Technology

Students will learn the requirements of J-STD-001, and demonstrate the skills for preparing and mounting Leaded and Leadless Surface Mount components to PWBs.

- SMT Criteria
- SMT Inspection Criteria
- SMT Demonstration
- SMT Lab
- Module 4 Review
- Module 4 Examination

DAY 5- Module 5

Inspection Methodology

Students will learn the quality and inspection requirements of J-STD-001.

- Theory of Inspection, SPC
- Defect Definition and Disposition
- Inspection Skills Demonstration
- Inspection Skills Lab
- Module 5 Review
- Module 5 Examination

Course Notes

Everyone who successfully completes the Certification examination with at least 70% average grade will receive:

- IPC-J/STD-001 Standard
- IPC-J/STD-001 Student Handbook
- IPC Certificate of Training

Fee

475 EUR/participant (the fee includes all original documents from IPC).
Maximum number of participants is 10.

Performance Evaluation

One final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one Open Book Examination and one Closed Book examination.

Who should attend

The course emphasis is on J-STD-001 document interpretation. Experienced assemblers who wish to gain in depth background on soldering practices should become certified to this program. As this course is designed to compliment existing soldering skills, prior soldering experience is recommended.

≈●≈

8. Certified IPC Specialist in IPC-A-600 "Acceptability of Printed Boards"

Course Contents

The IPC-A-600 describes preferred, acceptable, and nonconforming conditions–externally or internally observable on printed boards. Use the IPC-A-600 to ensure quality of your bare boards. This 3-day, lectured course utilizes the images in the IPC-A-600 document to discuss the internally and externally observable characteristics as well as flexible printed circuits, rigid, metal core and flush printed boards.

Course Duration

3 days

Course Outline

Topics include:

- Identifying laminate defects before they're a problem
- Identifying drill and hole plating defects
- Dealing with registration problems with solder masks
- Rejecting inferior incoming materials
- Shipping quality products
- Understanding and interpreting industry standards and specifications Improving SPC gathering skills

DAY 1

- Introduction
- General Overview
- Terms and Definitions
- Acceptance Criteria
- Externally Observable Characteristics
- Board Edges
- Base Material Surface and Subsurface
- Solder Coatings and Fused Tin Lead
- Holes Plated Through and Unsupported
- Printed Contacts
- Marking
- Solder Resist
- Dimensional Characteristics

DAY 2

- Internally Observable Characteristics
- Dielectric Materials
- Conductive Patterns
- Plated Through - Holes (General, Drilled, Punched)
- Flex Printed Circuit & Metal Core
- Flexible Printed Circuits
- Rigid - Flex Printed Boards
- Metal Core Printed Boards
- Flush Printed Boards
- Cleanliness Testing
- Solderability Testing
- Electrical Integrity

DAY 3 (ONE HALF DAY)

- Course Summary/Review
- Open Book Examination
- Closed Book Examination
- Instructor/Student Conference

Course Notes

Everyone who successfully completes the Certification examination with at least 70% average grade will receive:

- IPC-A-600 Standard
- IPC-A-600 Student Handbook
- IPC Certificate of Training

Fee

400 EUR/participant (the fee includes all original documents from IPC)
Maximum number of participants is 15.

Performance Evaluation

One final exam will be used at the end of the course to evaluate the performance. The exam is divided in two parts, one Open Book Examination and one Closed Book examination.

Who should attend

This being an advance course, anyone responsible for determining the quality and reliability of printed wiring board products should become certified. This includes quality supervisors, engineers, manufacturing supervisors and users of printed wiring boards.

≈●≈

9. Certified IPC Specialist in IPC7711 & 7721 "Rework of Electronic Assemblies Repair and Modification of Printed Boards and Electronic Assemblies"

Course Contents

The IPC-7711 is designed for soldered assembly rework–restoring PCB assemblies to their original drawings. It is comprised of 6 modules, which include general knowledge content, and specific Wires & Terminals, Through-Hole and Surface Mount rework procedures. IPC-7721 is designed for board repair– restoring a board's functional capability. It contains three modules–each emphasizing a different aspect of board repair.

Course Duration

3 days

Course Outline

IPC 7711&7721 Certified IPC Specialist Program is an advanced hands-on course that requires ample soldering skills. Candidates should have substantial soldering/electronics

assembly experience and should possess adequate soldering ability.

Day 1

- General knowledge (required)
- Common procedures
- Open Book exam
- Wire splicing Lab
- Through-Hole Lab
- Review & Assessment

Day 2

- Chip & Melf Procedures Removal/Installation Lab
- SOIC/SOT Procedures Removal/Installation Lab
- Review & Assessment

Day 3

- J-Lead & QFP Procedures Removal/Installation Lab
- PWB Circuit Repair
- Laminate Repair
- Conformal Coating Repair
- Review & Assessment

Course Notes

Everyone who successfully completes the Certification examination will receive:

- A student Handbook with excerpts from IPC-7711 and/or IPC-7721
- IPC Certificate of Training

Fee

350 EUR/participant (the fee includes all original documents from IPC)
Maximum number of participants is 15.

Performance Evaluation

The exam will be performed during the course to evaluate the performance. The exam is divided in two parts, one Open Book Examination and one very practical examination.

Who should attend

Anyone involved in the rework of components and/or repair of printed wiring boards should become certified.

≈●≈