ELEC-2005
Electronics in High Energy Physics
Winter Term: Introduction to Electronics in HEP

Printed Circuit Boards (PCB) and Hybrids: Part 1
Design and Assembly

Betty Magnin / TS-DEM
3 February 2005
From schematic to assembly

A 3-step process

Design → Manufacturing → Assembly
Design of PCB layout

1. Design steps
2. Tools
3. Good practice
Design of PCB layout

1. Design steps
   - Select and create components
   - Capture schematics
   - Define mechanical layout
   - Place components
   - Define PCB stackup
   - Routing
   - Autorouting
   - Post routing issues
   - Create manufacturing and assembly documents
Design of PCB layout

Select and create components

• Search into libraries
• Select according to electrical requirements
• Select component package (PTH/SMD, QFP/BGA, …)
• Create new components (standardize)
Design of PCB layout

Capture schematics
- Use hierarchical structure (pages, blocks)
- Insert all components (decoupling caps, safety components, ..)
- Trace bus and interconnections
- Arrange the presentation
- Verify (NC, all 0V, grounds and power supplies, …)
Design of PCB layout

Define mechanical layout

• Specific or standard formats (3U, 6U, 9U…)
• Mechanical requirements (fixing holes, cooling, …)
• Position of components on the front panel
Design of PCB layout

Place components

• Electrical constraints (Impedance, EMC, RF, …)
• Mechanical constraints (limited height, cooling…)
• Assembly process (PTH/SMD, top, bottom, clearance for machines, …)
Design of PCB layout

Determine the PCB layer stackup

• Even number of copper layers
• Symmetrical stackup
• Voltage planes next to ground planes to form embedded capacitors
• Impedance requirements
Design of PCB layout

PCB Routing

- Electrical requirements (line spacing and width, impedance, …)
- Iterative process between placing and routing
- Consider manufacturing capabilities
- Add test pads (for automatic test fixture)
Design of PCB layout

Autorouting

- Saves time for simple designs or simple parts of complex designs
- Routes according to specified parameters
- Not always optimum in terms of PCB manufacturing cost and efficiency (increases number of vias and density of lines)
- Suited for small quantities
Design of PCB layout

Post routing issues

• Add identification (part name, number, revision, …)
• Analyze the layout for potential signal integrity problems
• Add copper surfaces to equalize copper density
• Use CAD control tools to verify (minimum line spacing, shorts or not connected, …)
Design of PCB layout

Create specification for manufacturing and assembly

- Generate Gerber files for layers, use Extended Gerber (embedded apertures) format
- Generate drilling files
- Specify stackup and impedance controlled layers, laminate, marking, solder mask, finish required, hole plating requirements
- Create BOM and placing files for assembly
- Create a panel drawing (several PCB’s on one panel)
Design of PCB layout

2. Tools

- Design
- Documentation for assembly
- Archiving
Design of PCB layout

Design

- PCAD, CADENCE: maintained at CERN (training, support by IT and design office, centralized CERN library, …)
- MENTOR, PROTEL, Express PCB, Eagle, WinqCAD, Target 3AD, PCB123, CAD Design,…
Design of PCB layout

Documentation for assembly

Fabmaster

Import CAD and BOM
Import design data from more than 65 CAD tools; the most processors in the industry today.

Define Programs for Test Machines
Generate programs for more than 75 multi-vendor test machines; the most available in the industry today.

Shop Floor Documentation
Automatically create electronics assembly process instructions and shop floor documentation, with automatic ECO updates.

Define Assembly Sequences and Program SMT Lines
Generate programs for more than 150 SMT machines; the most outputs in the industry today.

Repair Process
Intelligent graphics and easy-to-use interfaces for logging defect data, and Hot Spot functionality allowing users to concentrate on areas where defects are more likely.

eMServer — Supporting manufacturing applications and information throughout the process
Design of PCB layout

Archiving
EDMS

3 February 2005  ELEC-2005  17
Design of PCB layout

3. Good practice

• Clearly specify all requirements before starting a layout: do it right the first time, rather than adding continuous changes will save time and problems (use check-lists and design reviews)

• Comply with manufacturing and assembly design rules when designing libraries and layout: fiducials, locating holes, minimum panel size, pads for wave soldering, distance around a component for inspection and repair, …
3. Good practice

- Keep in touch with progress in technologies: Pb-free will have an influence on a layout!
- Comply with known standards: IPC working groups have already done the job
- Use all possible tools to verify each step (Cadence: Checkplus, ..)
- Provide a clear and complete job to manufacturing and assembly: they cannot guess what you expect if you do not tell them (write instead of say)
From schematic to assembly

A 3-step process

Design → Manufacturing → Assembly
Assembly

- Components
- Variants of layouts
- Assembly steps
- Assembly process: automatic or manual
- Conditions for quality
Assembly

Components

• Through hole (PTH, Trad, …)
• Surface mount (SMD, CMS, …)
• Packaging : row, tape and reels, sticks, trays
Assembly

Layout

• Components may be placed on several manners
• Each will have an influence on the process to be used
Assembly

Through hole
Assembly

SMD, single side or both sides
Assembly

Trad + SMD-bottom
Assembly

Trad + SMD-top + SMD-bottom
Assembly steps

- Prepare components
- Mount
- Solder
- Clean
- Inspect
- Repair
Assembly processes

- 100% machine
- 100% manual
- Mixed (used at CERN)
Assembly

100% machine

- Process for series (>30 pieces)
- Manufacturing price is important for large quantities
- Components on tapes and reels or trays
- Board design must respect design rules (solder pads, fiducials, board sizes, spacing around components, …)
Assembly

Screen printing of solder paste
Assembly

Placement of SMD components
Assembly

Reflow soldering: convection, IR, vapor phase
Assembly

Insertion of through hole components, automatic or manual
Assembly

Wave soldering
Assembly

Inspection
AOI
Assembly

Test: In-Circuit, flying probe, dedicated functional
Manual or semi-manual process

- Process for prototypes or small series
- Handling of bulk components or parts of tapes is possible, but not preferable (risk in handling and placement of polarized components, ...)
- May use parts of automatic process (i.e.: placement of 100 coupling caps on 3 boards, reflow soldering, ...)
- Board design must respect design rules (SMD pads, free space around components, fiducial for solder paste dispensing or machine placement, ...)
- Expensive process
Assembly

Solder paste dispensing
Assembly

Placement
Assembly

Hand soldering
Assembly

Visual inspection
Assembly

Repair
Assembly

Conditions for quality

- Quality of assembly process
- Quality of components
- Quality of design
Assembly

Quality of assembly process

• Machines and processes understood and under control
• Trained operators
• Repetitive results
• Cleanliness is mandatory for reliability
Assembly

Quality of components

• Protect from
  • Humidity
  • Static discharge
  • Mechanical damage (bent pins)
  • Oxidation

• Allow automatic handling
  • Even for manual process
Assembly

Quality of components
• PCB is the most important component
• Adapted to controlled process

1206 : 12 * 0.001" X 6 * 0.001"
Assembly

Quality of design

• Designer’s functional requirements
• Manufacturing requirements
• Assembly requirements
• Compliance with tested standards (IPC)
• In phase with current technologies (lead-free)
ELEC-2005

Electronics in High Energy Physics

Winter Term: Introduction to Electronics in HEP

Printed Circuit Boards (PCB) and Hybrids: Part 1
Design and Assembly

Betty Magnin / TS-DEM
3 February 2005