* CAD (Computer Aided Design) FOR PRINTED WIRING BOARDS
* DESIGN FOR MANUFACTURABILITY

Continued..
Electrical circuits
......made by interconnecting discrete
Passive and Active Components on a suitable substrate

Discrete Passives

Resistors ✓
Capacitors ✓
Potentiometers
Trimmers
Inductors
Connectors

Also Mechanical and electromechanical components
Relay, Transformers, display devices

Discrete Actives

Integrated Circuits
SCMs ✓
MCMs ✓
Transistors
Diodes
Connectors

Connectors play a very significant role

They are the weakest link on the board and therefore require a lot of care in selecting

Connector Types

- Power connectors
- Signal connectors
- PCB connectors

Selection

✓ Number of required contacts
✓ Operating current and voltage
✓ Voltage drop permitted
✓ How often make and break?
✓ Space limitations
✓ Contact pressure - decides plating finish

Au, Sn, Au, Sn-Pb, Alloy 42
Design Expertise

- Effects of a CAD layout on the final product quality and the cost of manufacturing
- A general understanding of the end-user’s assembling capabilities
- Overall view of the manufacturer’s capabilities
- PCB fabrication is now a complex process
  - Reasonable understanding of processes especially for HDI (high-density interconnects)
  - Design rules and manufacturing processes!

*technologies large nos.*
Schematic Capture

- Circuit diagram to be drawn, graphics program
- Generate circuit symbols on the screen
- Generate interconnection, verify
- Printer output, plotter output
- NETLIST generation; various formats-fundamental file for other operations
- Symbol library and Parts library
- Library manager; editing
Circuit simulators

- Simulation of circuits for determining performance
- Digital circuit simulation
- Logic verification, fault simulation
- Timing analysis, glitches and their effects
- Analog and power circuit simulation
- Power dissipation analysis
PCB Layout Design

- Component placement
  - Orientation of components
- Routing
  - manual and auto
- Design rule verification
  - Current density, line widths
  - line spacing, clearances etc.
- Multilayer capability
  - 4 - 64 layers
  - 4 - 12 layers
PCB Fabrication Tools

- Drilling details for a NC machine
- Solder mask pattern generation for two sides
- Legend printing masks for marking components during assembly, for two layers
- Outputs for bare board tester; electrical test
- Electrical layers; multilayer

(Technology Files)
Production documentation

- Component Lists
  - Lists, part numbers, vendor details
- Component layout
- PCB layout and test point markings
- Drilling details
- Quality procedures

Bill of Materials (BOM)

Test

Drilling via (plated) monitoring holes
Test and evaluation

- Thermal analysis
  - temperature profile of the board; hot spots
  - Component re-layout
  - Special heat sinks
- EMI
  - reflections, cross talk
Netlist example

- Net 1
  - OA3, 2 → C4, 1 → R6, 2 → R7, 1
- Net 2
  - D1, 1 → OA3, 3 → R7, 2 → C4, 2

- Net 21: GND
  - OA2, 3 → EC1, 5
  - U1, 7 → R9, 2
  - EC1, 2

- C3, 2 → Q2, 1
- Z1, 2 → EC1, 2

- Jumpers

- Multiple connections with different thickness Cu
Three placement procedures

- Manual / Interactive
  - You identify the components for auto placement
- Assisted
  - You allow the program to do initial random or constructive placement to get started
- Automated
  - The system attempts a placement with minimum connection length $(Cu)$
Conductor Routing

- Fast results
- Ordering; shortest to longest, pin & gate swapping
- Post-route processing
- Via minimization (optimization)
- Beveling of tracks (Mitring, 45°)
- Routers are grid-based

A right angle corner is necessarily wider than the rest of the trace. This results in the decrease in $Z_0$, the intrinsic impedance of the trace and therefore causes an impedance mismatch at the corner. This, in turn, causes reflections, signal distortions and noise along the trace.
Optimization Operations in CAD

♦ Creation of components and footprints (THT & SMT)
♦ Gate & pin swapping
♦ Manual, interactive & automatic routing
♦ Clean-up and optimizing the interconnection structure
♦ Final operations (segments modify, track width modify, miter & fillet corner)
♦ Copper maximizing
♦ Transfer between layers

Post processing Operations

♦ Creating Technology files for sending to manufacturer
♦ EXCELLON format for NC Drilling operation
♦ GERBER format for photoplotting operation
  * Photoplots are necessary for all manufacturing layers of PWB
  * Masks are from silver halide photo films
♦ Assembly Drawing to be printed/documentated
♦ Production planning document for manufacturing
♦ Edit software will be used by manufacturer for including tool holes, logos, batch nos. etc.
Wiring Board Configurations

- Single-sided
- Double-sided
- Double-sided with layer interconnections — (via or plated through hole)
- Multilayer
- Flexible and Rigid-flex
- High Density PWB
Wiring Board Configurations