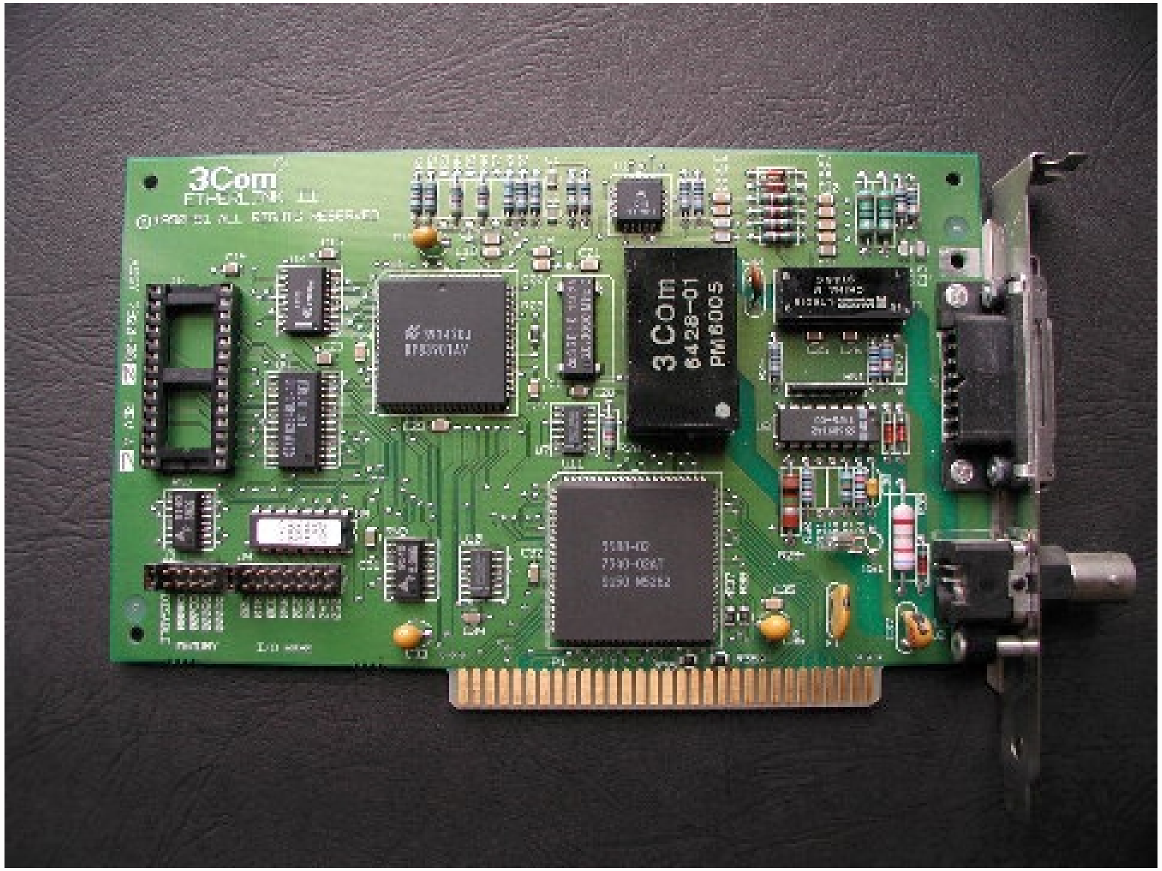
PCBs/Overview

– Printed Circuit Boards (PCB)
  - Introduction
  - Conductors. Supply Planes. Dielectric.
  - Vias
  - PCB Manufacturing Process
  - Electronic Assembly Manufacturing Process

PCBs/Overview

- For electronic assemblies PCBs are substrates providing mechanical support as well as electrical interconnect
- PCB: Rigid or flexible substrate with single or multiple layers of conductors separated by insulating layers



Note: PCBs are sometimes also referred to as PWBs (Printed Wiring Boards)

PCBs/General

- PCB Origin: United States
- Therefore: Non-SI units (oz, mil, in) have been universally adopted for specifying PCBs:
  - Board dimensions in inches
    (**1in = 25.4mm**)
  - Dielectric thicknesses and conductor widths/spacing in mil
    (**1mil = 0.001in** = 25.4μm)
  - Conductor (commonly copper) thicknesses in ounces (oz)
    The weight of conductor metal in a square foot of material. Typical copper thicknesses are:
    0.5oz (17.5μm), 1oz (35μm), 2oz (70μm), 3oz (105μm)

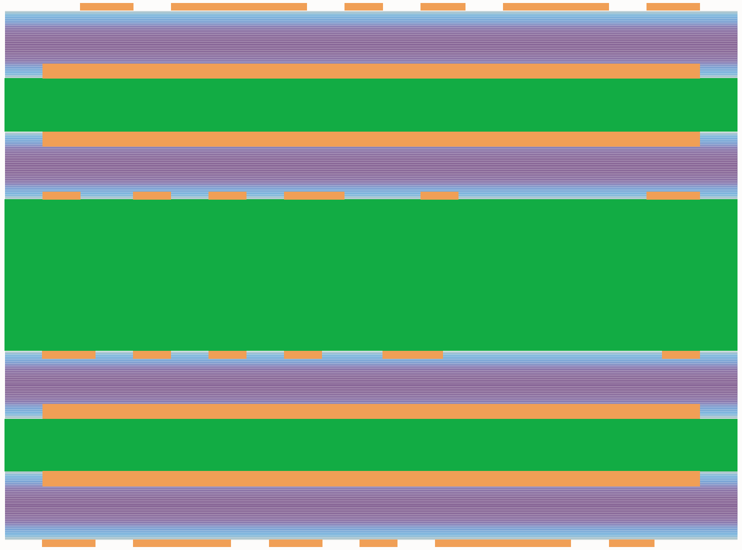
PCBs/General

– PCB General Dimensional Specifications:

- Finished thicknesses
  - Standard: 31mil, 39mil and 62mil (0.8mm, 1.0mm and 1.6mm)
  - Non-standard:
    - » Readily available for high-volume orders
    - » Board thicknesses: 10mil-125mil
      (many PCB manufacturers stop at 20mil - depending on plating finish)
- Maximum dimensions typically 16in x 20in
- Irregular shapes/slots etc readily available (routing)

PCBs/Stackup

Shown: Cross-section of a typical 8-layer PCB Stackup



- Typical PCB Stackup:
  - Alternating layers of core and prepreg
  - Core: Thin piece of dielectric with copper foil bonded to both sides. Core dielectric is *cured* fiberglass-epoxy resin
  - Prepreg: *Uncured* fiberglass-epoxy resin. Prepreg will cure (i.e. harden) when heated and pressed
  - Outermost layers are prepreg with copper foil bonded to the outside (surface foils)
  - To avoid crosstalk: Wires on adjacent signal layers are routed mostly orthogonally
  - Stackup is symmetric about the center of the board in the vertical axis to avoid mechanical stress in the board under thermal cycling

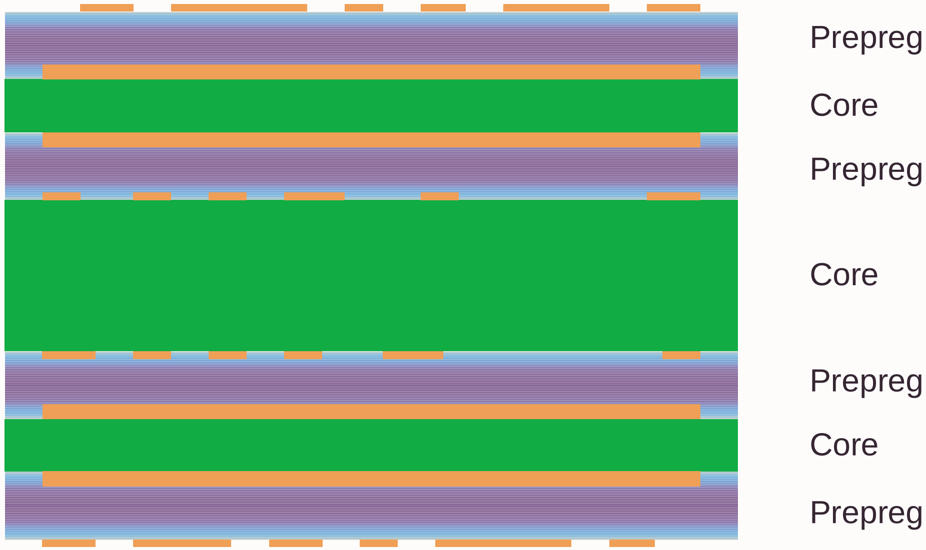
PCBs/Conductors

– Conductor:

- Material: Typically Cu
- Number of layers:
  - Single or multilayer (up to 20 layers, and more)
  - Dedicated supply layers (also called “ground layers”, “ground planes”)
  - Most popular: 4-8 signal layers plus 4-8 ground layers
- Material dimensions:
  - Thicknesses: 0.5oz-3oz typically. 0.5oz/1oz standard for inner layers.
  - Trend: towards 0.25oz (particularly for laminated IC packages)
  - Width and spacing: $\geq$ 5mil

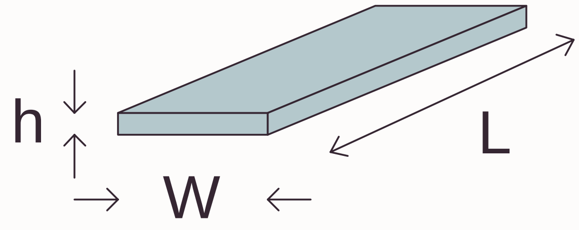
PCBs/Stackup

Shown: Cross-section of a typical 8-layer PCB Stackup



- Power Planes:
  - Power planes are typically built on thinnest core available from a fabrication vendor to maximize the capacitance between the planes
  - Power planes often use thicker copper layers than signal layers to reduce resistance
- Why power planes?
  - Provide stable reference voltages for signals
  - Distribute power to all devices
  - Control cross-talk between signals

PCBs/Conductors



– PCB sheet resistances

- Cu resistivity $\rho=1.7*10^{-8}\Omega m$
- Remember: Sheet resistance…

$$R = \frac{\rho \cdot L}{A} = \frac{\rho \cdot L}{h \cdot W} = Rs\frac{L}{W}$$

PCB Sheet Resistances at T=300K (TC Copper: +3930ppm)	
Conductor	**Rs in μΩ**
PCB Copper Track (0.5oz Cu)	971
PCB Copper Track (1oz Cu)	486
PCB Copper Track (2oz Cu)	243
PCB Copper Track (3oz Cu)	162

Compare to Semiconductor Rs figures:

Sheet Resistances	
Material	**Rs in Ω**
Metal (Aluminium) (top layer)	0.05
Metal (Aluminium) (lower layers)	0.1
Polysilicon (silicided)	6
Diffusion (n+, p+, silicided)	10
Polysilicon (doped)	30
Diffusion (n+, p+)	100
n-well	5k
Nichrome Mixed signal IC resistor material. Stable and laser-trimmable	several k
Polysilicon (undoped)	several Meg

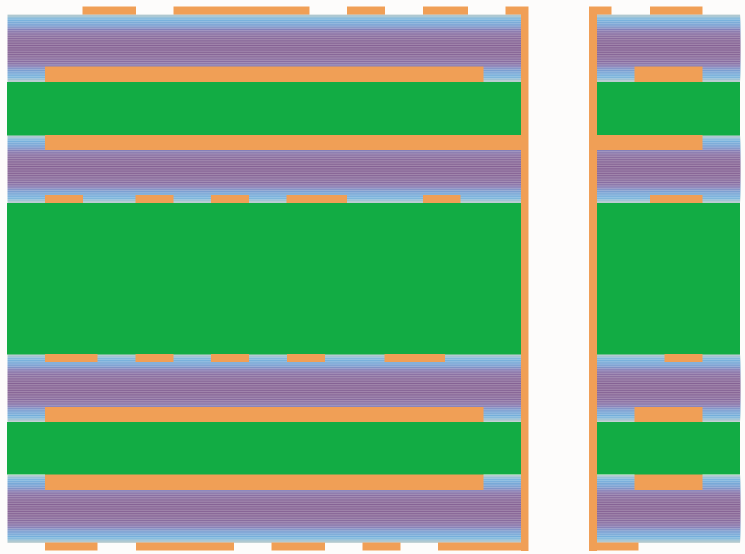
PCBs/Insulators

- Dielectric Materials:
  - Typically Fiberglass Epoxy-resin (FR4)
    - most common, widely available, relatively low cost
    - rigid structure
    - temperature range up to 130°C
  - CEM: Extremely low-cost. Punchable holes.
  - Also available:
    - Polyimide: high temperature, rigid or flexible
    - Teflon: high temperature
  - Thicknesses
    - Standard core thicknesses for ML PCBs: 5, 8, 10, 14, 20, 40 mil
    - Prepreg thicknesses: 4mil typical
- Most PCB materials support a (relatively) controlled dielectric/impedance
  - Suitable for transmission lines

PCBs/Vias

– Vias

- Interconnect layers through vias (plated holes)
- Via dimensions:
  - – Standard minimum finished hole sizes: ≥ 8mil
  - – Aspect ratio restrictions apply



Aspect ratio of a via: Ratio of board thickness to via diameter. Allows judgement of manufacturability. The larger the aspect ratio, the more difficult it is to achieve reliable plating. Premium charge for aspect ratios > 8.

PCBs/Vias

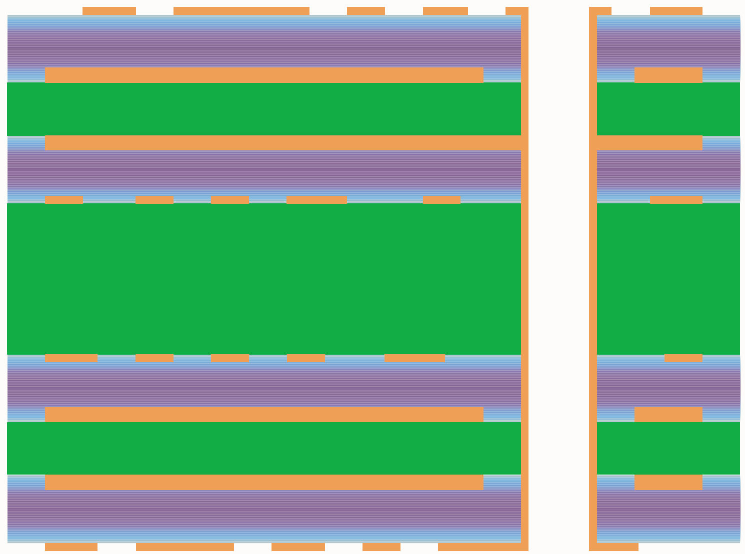
– Vias

- Vias require pads on each layer to which they connect. Because the holes are not guaranteed to be perfectly aligned with the copper traces there will need to be an annulus of copper around the plated hole. This is to ensure that the copper won't be broken by the drilling operation
- Pads on inner vias are larger than outer pads to allow for greater dimensional tolerances
- Where a via passes through a plane (i.e. not connect to the plane) a clearance hole is required
- Where a via is supposed to connect to a plane, a thermal relief structure is required (usually four small metal bridges between via and plane). Thermal relief is required to facilitate soldering operations.

PCBs/Vias

– Vias

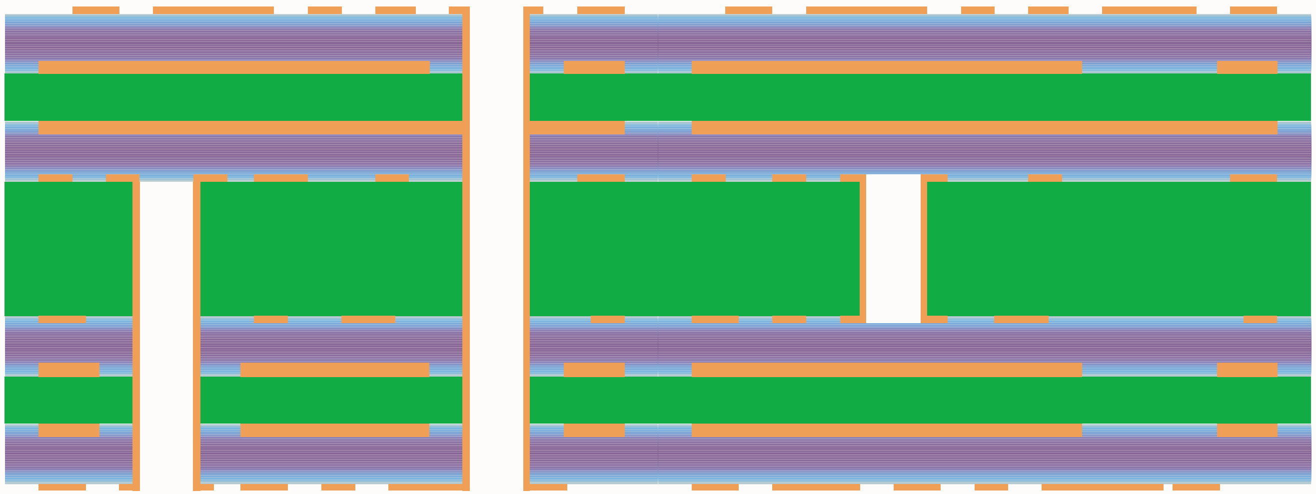
- Vias are much larger than signal wires
- Vias occupy all layers (with the exception of blind and buried vias)
- Consequence:
  Vias reduce wiring density
  and are therefore **expensive!**



PCBs/Special Vias

– Vias

- Special vias available for high-volume PCBs:
  - **Blind vias** (connection of outer layer to inner layer)
  - **Buried vias** (connection of inner layers - preferably on same core!)
- Advantages
  - Increased wiring density (vias don't occupy all layers)
  - Product safety (creepage and clearance distances for electrical insulation)
- Penalties:
  - Restricted choice of suppliers
  - More complex process:
    - » Cost
    - » Reliability



PCBs/Manufacturing Process

– Manufacturing process steps
(for a typical rigid multilayer PCB representing about 70% of all PCBs manufactured)
  - PCB data acquisition
  - Preparation of PCB laminate (core)
  - Inner layer image transfer
  - Laminate layers
  - Drilling and cleaning holes
  - Make holes conductive
  - Outer layer image transfer
  - Surface finish
  - Final fabrication

PCBs/Manufacturing Process/Step 1

– Step 1: PCB data acquisition
  - Files transferred from PCB design house to PCB manufacturing facility:
    - Gerber files, drill files, fabrication drawings
  - File review by PCB manufacturer
  - Creation of PCB tooling
    - Photo-tool for image transfer
      Image created by PCB software is reproduced on film using laser photoplotters
    - Drill files
    - Profile routing files
      CNC route file
    - All tooling is stepped and repeated for optimum utilisation of standard panels (24in x 18in)

PCBs/Manufacturing Process/Step 2

– Step 2: Preparation of PCB laminate (core)

- Dielectric material: Woven glass fiber or paper
  Material depends on the function of the PCB. Some materials perform better in some environments than others (heat, humidity). Some materials are more suitable for particular manufacturing processes (e.g. hole punching). Others again are chosen for electric properties (permittivity). Most widely used: FR4 / CEM
- Coat/impregnate dielectric material with resin & harden
- Copper foil is rolled or electrolytically deposited on the base laminate
- Core material is sheared to panel size
- Core material is cleaned mechanically and/or chemically
  Removal of surface contamination required to promote subsequent adhesion of photoresist (PR)

PCBs/Manufacturing Process/Step 3

– Step 3: Inner layer image transfer (photo-lithography)

Purpose: Transfer circuit image to core through “print-and-etch” process

- Coat copper foils with photoresist (PR)

  Negative PR: Light-sensitive organic PR polymerises (“hardens”) when exposed to light. Polymerised PR will resist etching.

- Place phototool and expose to light

  After expose, PR layer is developed. Polymerised areas remain, unexposed areas are washed away.

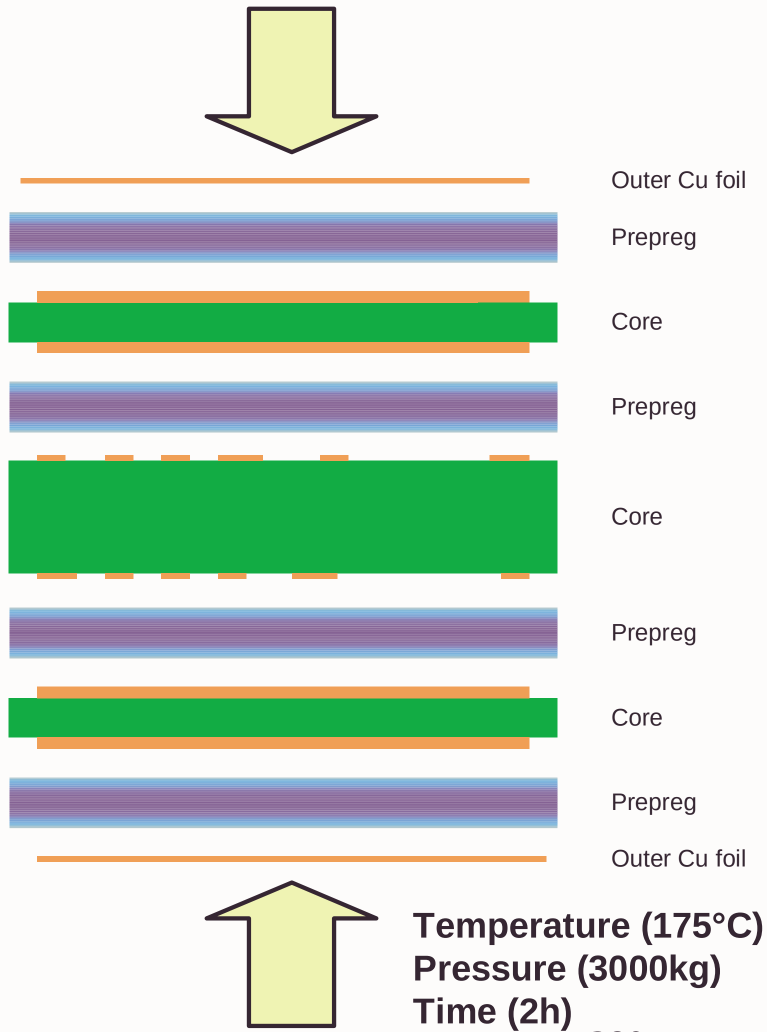
- Etching

  Selectively remove exposed copper areas. Etching is performed with conveyorised equipment (etchant flood rinse, several water rinses). Common etchants: Acidic cupric chloride and alkaline ammoniacal.

PCBs/Manufacturing Process/Step 4

– Step 4: Lamination

- Cores are pinned in a stack with sheets of prepreg (b-stage) separating the copper layers. Outer layers are made with a foil of copper
- Horizontal alignment critical!
- Stack is pinned between two heavy metal plates creating a “book”.
- Book is put in a heated hydraulic press for about 2h.



Prepreg is available in different styles with varying amounts of resin and glass fibers. This allows the manufacturer to control thickness between layers and thickness of the overall PCB.

PCBs/Manufacturing Process/Step 5

– Step 5: Drilling and cleaning

Purpose: Holes are drilled through PCB to interconnect layers (vias), and to allow the insertion of PTH components

- Drilling performed with CNC equipment

  Using drill files. Alternative methods to drilling exist (punching, laser).

- Multiple panels can be drilled together

  Drilling of complex boards can take several hours per load

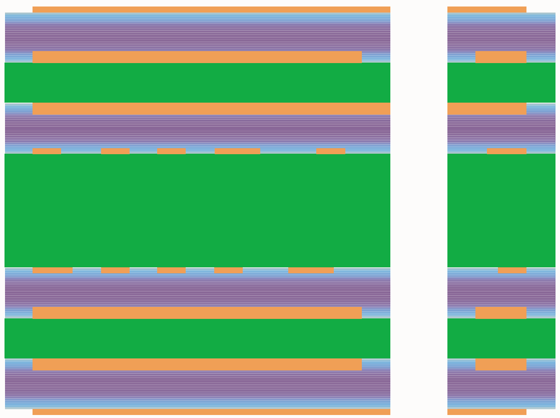
- Desmear

  Desmear removes the melted resin smear

- Etchback

  Etch glassfibers. Goal: Copper on the inner layers protrude out into the barrel of the hole

- Panels are deburred/scrubbed after drilling



PCBs/Manufacturing Process/Step 6

– Step 6: Make holes conductive

PCB substrate is not conductive. Therefore a non-electrolytic deposition method is required. In a later process step, electroplating to the required thickness can be performed

- Process: Electroless copper

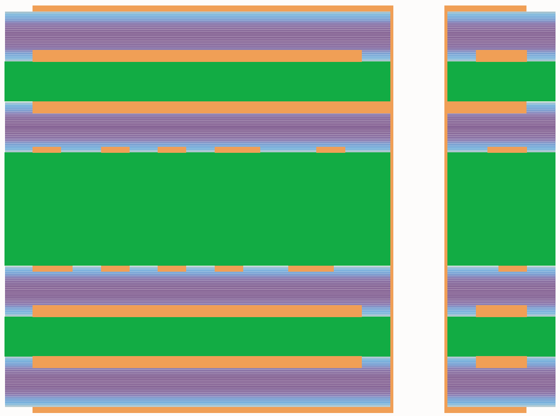
  Electroless copper is reliable but alternative methods exist. Electroless copper has some significant disadvantages (like exposure to formaldehyde, which is carcinogen).

- Electroless copper bath

  Deposits 75-125µIn of copper

- Constituents of electroless copper:

  Sodium hydroxide, formaldehyde, EDTA and copper salt. Complex reaction catalysed by palladium, formaldehyde reduces the copper ion to metallic copper.



PCBs/Manufacturing Process/Step 7

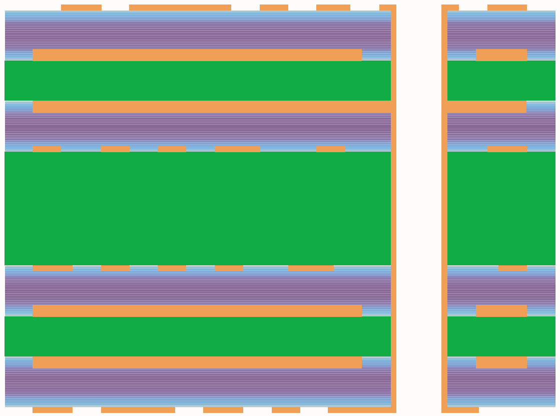
– Step 7: Outer layer image transfer

Most common process: Print, pattern plate, and etch

- Coat copper foils with photoresist (PR)
- Place phototool and expose to light

  Outer layer phototools are positive images of the circuit. Circuit image is developed away exposing the copper. PR remaining on the panel will act as plating resist
- Pattern plating (copper electroplating)

  Outer layers will be plated to a thickness of 1.5mil (to ensure a minimum thickness of 1mil in the holes). Copper electroplating takes place in a copper sulfate bath. Plating is performed at roughly 30A/ft^2. Plating duration is roughly 1h.
- Plate metallic etch resist
- Etching



PCBs/Manufacturing Process/Step 8

– Step 8: Surface finish

Purpose: Prevent copper oxidation. Facilitate solderability.

- Most popular surface finish process: SMOBC/HASL:
  SMOBC: Solder-mask-over-bare-copper. HASL: hot-air-solder-leveling
- Solder mask pre-clean (mechanical scrub)
- Application of solder mask
  Purpose of solder mask: Insulate those portions where no solder is required. Most popular mask type: LPI (liquid photoimageable).
- Application of flux
  Provides oxidation protection. Affects heat transfer during solder immersion.
- HASL
  Panels are dipped into molten solder (237°C). Panels are then rapidly carried past jets of hot air. Exposed copper is coated with solder. Masked areas remain solder-free. Panels are then cleaned in hot water.

PCBs/Manufacturing Process/Step 9

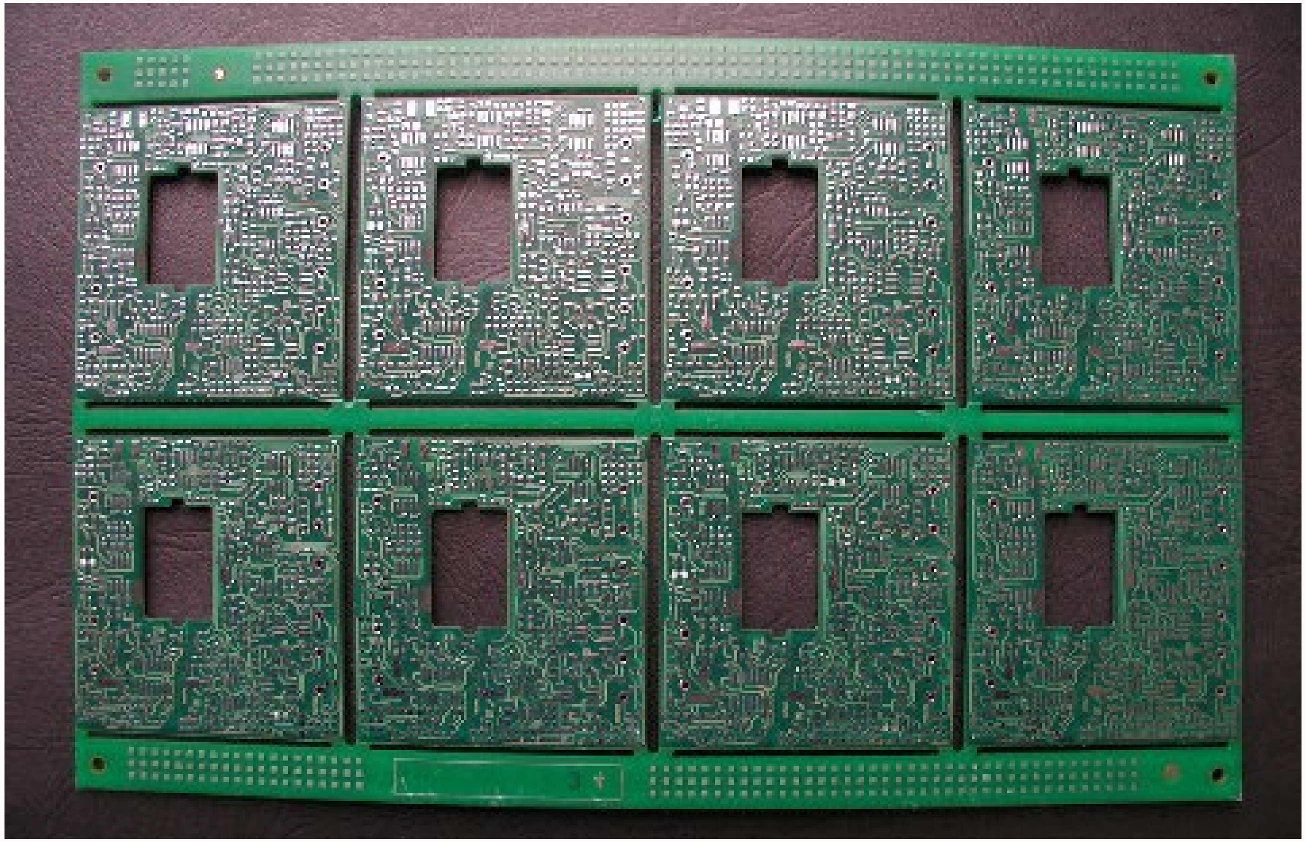
– Step 9: Final fabrication

Mechanical features are added to the board (mounting holes, cutouts, etc)

- Routing done through CNC machines
- De-panelisation
  - Partial de-panelisation. Most of the circuit is routed out of the panel, but tabs remain to hold the circuit in place. This allows the assembly machine to populate multiple boards at once. Afterwards, the circuit can be snapped or broken out of the panel. Such panels are called “breakaways”, “snaps”, or “arrays”.
  - The alternative to partial de-panelisation is to have the panel V-scored. V-scoring is done through a thin rotating scoring blade that will route across the top and the bottom of the panel with about 30% of the thickness of the panel. V-scoring allows more circuits per panel (no spacing is required for routing bits).

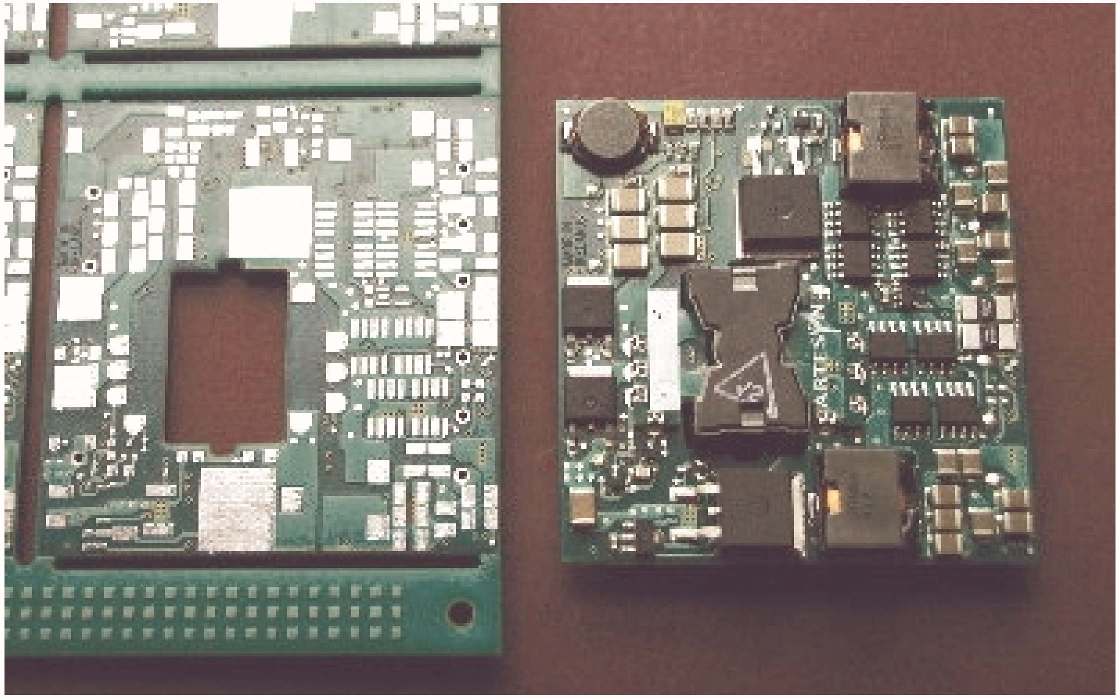
PCBs/Panels

- PCB Boards are fabricated in panels to minimise cost of PCB manufacturer and assembly manufacturer
- Typical panel dimensions are 18*24in (460x610mm)
- Overall board dimensions are kept much smaller than a full panel



PCBs/Typical Assembly Process

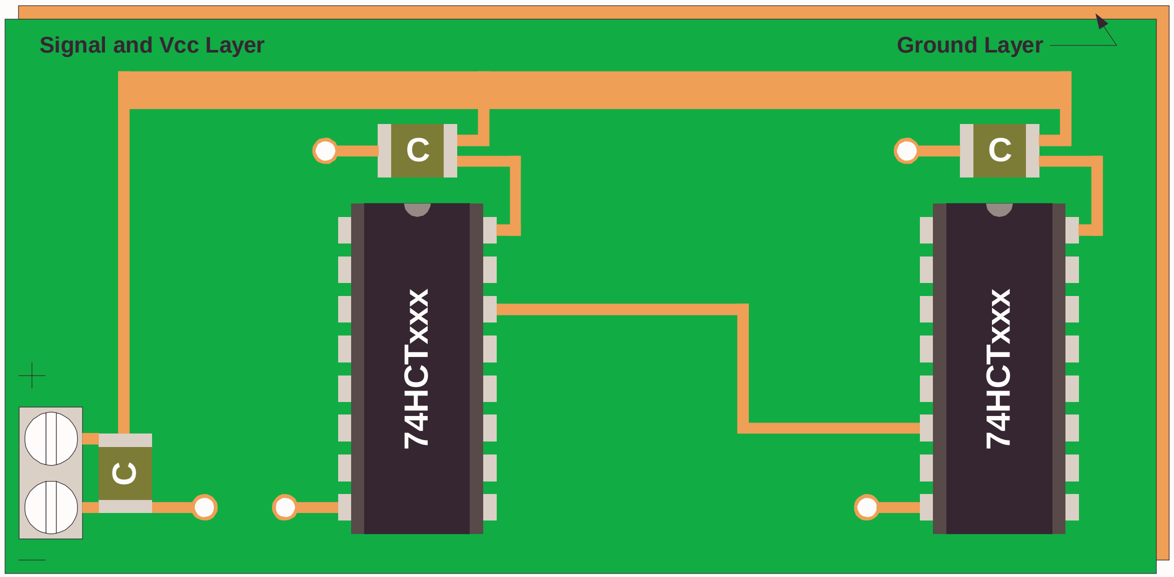
– Panel of an electronic unit (left)
– Electronic assembly after manufacturing process
– Typical manufacturing process:
    - Silk-screen solder paste onto the panel
    - Pick-and-place (P&P) components (P&P machine or manually)
    - Heavier components and components on bottom side of the board need to be glued down (epoxy-based glue)
    - Soldering in IR reflow oven (providing the required reflow temperature profile of gradually increasing, sustaining, and removing heat)
    - Visual inspection (manual or vision systems)
    - Electrical testing (ATE)



PCBs/Layout/Do's and Don'ts

– Do's:

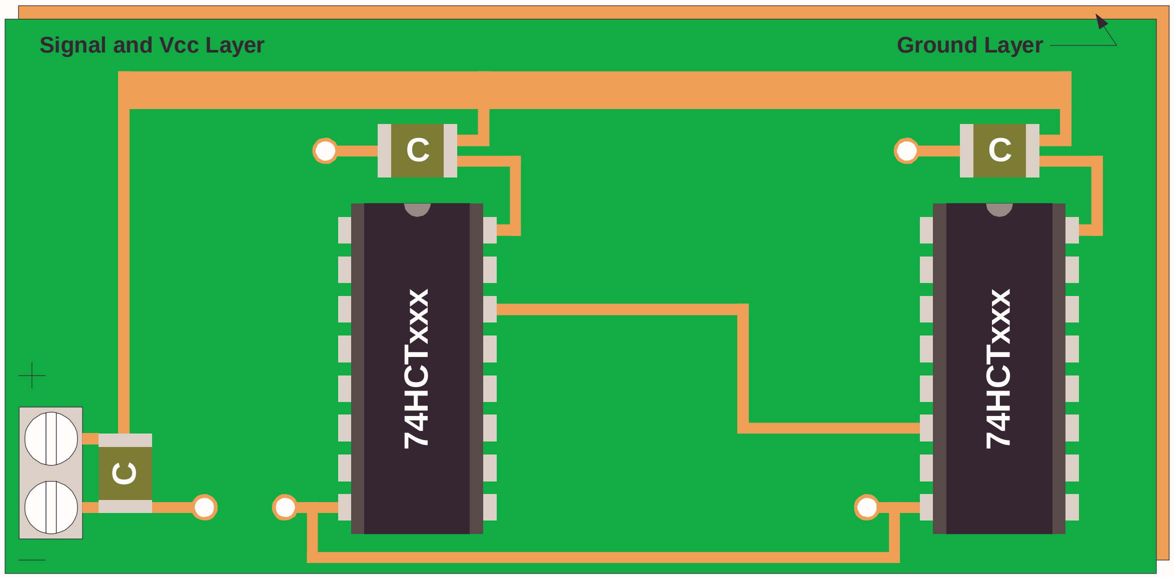
- Use (continuous) supply layers whenever possible. Keep connections to supply layers short
- Use SMT components wherever possible
- Use filter components where required and possible
- Place blocking capacitors as close as possible to supply pins of transient loads. Use star-point connections at blocking caps.



PCBs/Layout/Do's and Don'ts

– Do not's:

- **Do not create ground loops!** Mutual inductances couple ground loops to other currents loops. Ground loops typically have very small impedances. Currents coupled into ground loops can be very large. They cause problems like ground bounce, signal distortion, etc.



PCBs/Layout/Do's and Don'ts

– Do not's:

- **Avoid discontinuities in grounds layers!** High frequency return currents in ground layers follow the path with the least inductance. This path is usually directly underneath the signal trace. If there are discontinuities in the ground layer high frequency currents cannot flow underneath the signal trace. Large loop areas are created, and cause a variety of problems (ground bounce, cross-talk)