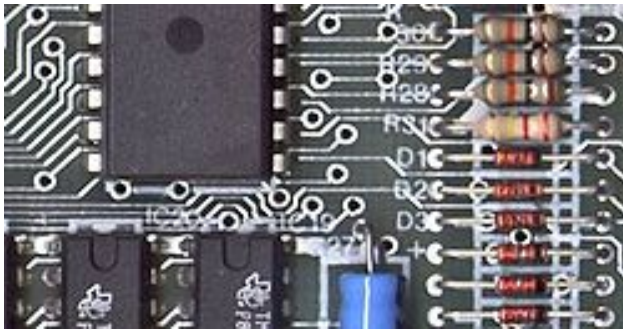




Printed circuit board



Part of a 1983 *Sinclair ZX Spectrum* computer board; a populated PCB, showing the conductive traces, vias (the through-hole paths to the other surface), and some mounted electrical components

A **printed circuit board**, or **PCB**, is used to mechanically support and electrically connect **electronic components** using **conductive** pathways, tracks, or **traces**, etched from copper sheets laminated onto a non-conductive **substrate**. It is also referred to as **printed wiring board** (**PWB**) or **etched wiring board**. A PCB populated with electronic components is a **printed circuit assembly** (**PCA**), also known as a **printed circuit board assembly** (**PCBA**).

PCBs are inexpensive, and can be highly reliable. They require much more layout effort and higher initial cost than either **wire-wrapped** or **point-to-point constructed** circuits, but are much cheaper and faster for high-volume production. Much of the electronics industry's PCB design, assembly, and quality control needs are set by standards that are published by the **IPC** organization.

History

The inventor of the printed circuit was the **Austrian** engineer **Paul Eisler** (1907–1995) who, while working in England, made one circa 1936 as part of a radio set. Around 1943 the USA began to use the technology on a large scale to make rugged radios for use in **World War II**. After the war, in 1948, the USA released the invention for commercial use. Printed circuits did not become commonplace in consumer electronics until the mid-1950s, after the *Auto-Semby* process was developed by the **United States Army**.

Before printed circuits (and for a while after their invention), **point-to-point construction** was used. For prototypes, or small production runs, **wire wrap** or **turret board** can be more efficient.

Originally, every electronic component had wire leads, and the PCB had holes drilled for each wire of each

component. The components' leads were then passed through the holes and **soldered** to the PCB trace. This method of assembly is called **through-hole construction**. In 1949, Moe Abramson and Stanislaus F. Danko of the **United States Army Signal Corps** developed the **Auto-Semby** process in which component leads were inserted into a copper foil interconnection pattern and dip soldered. With the development of board lamination and etching techniques, this concept evolved into the standard printed circuit board fabrication process in use today. Soldering could be done automatically by passing the board over a ripple, or wave, of molten solder in a **wave-soldering** machine. However, the wires and holes are wasteful since drilling holes is expensive and the protruding wires are merely cut off.

In recent years, the use of **surface mount** parts has gained popularity as the demand for smaller electronics packaging and greater functionality has grown.

Manufacturing

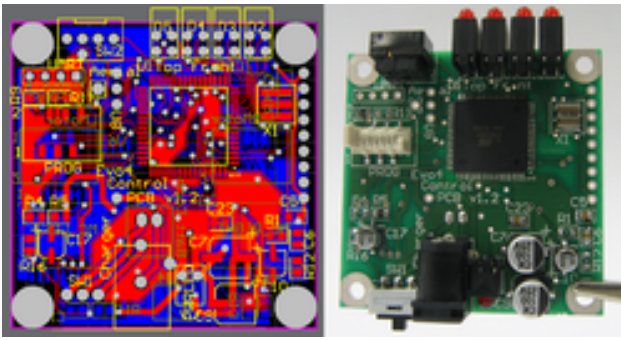
Materials

Conducting layers are typically made of thin copper foil. Insulating layers **dielectric** are typically laminated together with **epoxy resin prepreg**. The board is typically coated with a solder mask that is green in color. Other colors that are normally available are blue and red. There are quite a few different dielectrics that can be chosen to provide different insulating values depending on the requirements of the circuit. Some of these dielectrics are **polytetrafluoroethylene** (Teflon), FR-4, FR-1, CEM-1 or CEM-3. Well known prepreg materials used in the PCB industry are **FR-2** (Phenolic cotton paper), FR-3 (Cotton paper and epoxy), **FR-4** (Woven glass and epoxy), FR-5 (Woven glass and epoxy), FR-6 (Matte glass and polyester), G-10 (Woven glass and epoxy), CEM-1 (Cotton paper and epoxy), CEM-2 (Cotton paper and epoxy), CEM-3 (Woven glass and epoxy), CEM-4 (Woven glass and epoxy), CEM-5 (Woven glass and polyester). Thermal expansion is an important consideration especially with BGA and naked die technologies, and glass fiber offers the best dimensional stability.

Typical density of a raw PCB (an average amount of traces, holes, and vias, with no components) is 2.15g / cm³.

Patterning (etching)

The vast majority of printed circuit boards are made by bonding a layer of copper over the entire substrate,



A PCB as a design on a computer (left) and realized as a board assembly with populated components (right). The board is double sided, with through-hole plating, green solder resist, and white silkscreen printing. Both surface mount and through-hole components have been used.

sometimes on both sides, (creating a "blank PCB") then removing unwanted copper after applying a temporary mask (eg. by etching), leaving only the desired copper traces. A few PCBs are made by *adding* traces to the bare substrate (or a substrate with a very thin layer of copper) usually by a complex process of multiple **electroplating** steps.

There are three common "subtractive" methods (methods that remove copper) used for the production of printed circuit boards:

1. uses etch-resistant inks to protect the copper foil. Subsequent etching removes the unwanted copper. Alternatively, the ink may be conductive, printed on a blank (non-conductive) board. The latter technique is also used in the manufacture of **hybrid circuits**.
2. uses a photomask and chemical etching to remove the copper foil from the substrate. The photomask is usually prepared with a **photoplotter** from data produced by a technician using CAM, or **computer-aided manufacturing** software. Laser-printed transparencies are typically employed for *phototools*; however, direct laser imaging techniques are being employed to replace phototools for high-resolution requirements.
3. uses a two or three-axis mechanical milling system to mill away the copper foil from the substrate. A PCB milling machine (referred to as a 'PCB Prototyper') operates in a similar way to a **plotter**, receiving commands from the host software that control the position of the milling head in the x, y, and (if relevant) z axis. Data to drive the Prototyper is extracted from files generated in PCB design software and stored in HPGL or **Gerber** file format.

"Additive" processes also exist. The most common is the "semi-additive" process. In this version, the unpatterned board has a thin layer of copper already on it. A reverse mask is then applied. (Unlike a subtractive process mask, this mask exposes those parts of the substrate that will eventually become the traces.) Additional copper is then plated onto the board in the unmasked areas; copper may

be plated to any desired weight. Tin-lead or other surface platings are then applied. The mask is stripped away and a brief etching step removes the now-exposed original copper laminate from the board, isolating the individual traces. Some boards with plated thru holes but still single sided were made with a process like this. **General Electric** made consumer radio sets in the late 1960s using boards like these.

The additive process is commonly used for multi-layer boards as it facilitates the **plating-through** of the holes (to produce conductive **vias**) in the circuit board.

Lamination

Some PCBs have trace layers inside the PCB and are called *multi-layer* PCBs. These are formed by bonding together separately etched thin boards.

Drilling

Holes through a PCB are typically drilled with tiny drill bits made of solid **tungsten carbide**. The drilling is performed by **automated drilling machines** with placement controlled by a *drill tape* or *drill file*. These computer-generated files are also called *numerically controlled drill* (NCD) files or "**Excellon files**". The drill file describes the location and size of each drilled hole. These holes are often filled with annular rings (hollow rivets) to create **vias**. Vias allow the electrical and thermal connection of conductors on opposite sides of the PCB.

Most common laminate is epoxy filled fiberglass. Drill bit wear is in part due to the fact that glass, being harder than steel on the **Mohs scale**, can scratch steel. High drill speed necessary for cost effective drilling of hundreds of holes per board causes very high temperatures at the drill bit tip, and high temperatures (400-700 degrees) soften steel and decompose (oxidize) laminate filler. Copper is softer than epoxy and interior conductors may suffer damage during drilling.

When very small vias are required, drilling with mechanical bits is costly because of high rates of wear and breakage. In this case, the vias may be evaporated by **lasers**. Laser-drilled vias typically have an inferior surface finish inside the hole. These holes are called *micro vias*.

It is also possible with *controlled-depth* drilling, laser drilling, or by pre-drilling the individual sheets of the PCB before lamination, to produce holes that connect only some of the copper layers, rather than passing through the entire board. These holes are called *blind vias* when they connect an internal copper layer to an outer layer, or *buried vias* when they connect two or more internal copper layers and no outer layers.

The walls of the holes, for boards with 2 or more layers, are made conductive then plated with copper to form *plated-through holes* that electrically connect the conducting layers of the PCB. For multilayer boards, those with 4

layers or more, drilling typically produces a *smear* comprised of the high temperature decomposition products of bonding agent in the laminate system. Before the holes can be plated through, this *smear* must be removed by a chemical *de-smear* process, or by *plasma-etch*. Removing (etching back) the smear also reveals the interior conductors as well.

Exposed conductor plating and coating

PCBs^[1] are plated with Solder, Tin, or Gold over Nickel as a resist for **etching** (removal) away the (unneeded after plating) underlying copper.^[2] Matte solder is usually fused to provide a better bonding surface or stripped to bare copper. Treatments, such as benzimidazolethiol, prevent surface oxidation of bare copper. The places to which components will be mounted are typically plated, because untreated bare copper oxidizes quickly, and therefore is not readily solderable. Traditionally, any exposed copper was coated with **solder** by hot air solder levelling (**HASL**). This solder was a **tin-lead** alloy, however new solder compounds are now used to achieve compliance with the **RoHS** directive in the **EU** and **US**, which restricts the use of lead. One of these lead-free compounds is SN100CL, made up of 99.3% tin, 0.7% copper, 0.05% nickel, and a nominal of 60ppm germanium.

It is important to use solder compatible with both the PCB and the parts used. An example is Ball Grid Array (BGA) using tin-lead solder balls for connections losing their balls on bare copper traces or using lead-free solder paste.

Other platings used are OSP (organic surface protectant), immersion silver (**IAG**), immersion tin, electroless nickel with immersion gold coating (**ENIG**), and direct gold (over nickel). **Edge connectors**, placed along one edge of some boards, are often nickel plated then **gold plated**. Another coating consideration is rapid diffusion of coating metal into Tin solder. Tin forms intermetallics such as Cu₅Sn₆ and Ag₃Cu that dissolve into the Tin liquidus or solidus(@50C), stripping surface coating and/or leaving voids.

Electrochemical migration (ECM) is the growth of conductive metal filaments on or in a printed circuit board (PCB) under the influence of a DC voltage bias.^{[3][4]} Silver, zinc, and aluminum are known to grow whiskers under the influence of an electric field. Silver also grows conducting surface paths in the presence of halide and other ions, making it a poor choice for electronics use. Tin will grow "whiskers" due to tension in the plated surface. Tin-Lead or Solder plating also grows whiskers, only reduced by the percentage Tin replaced. Reflow to melt solder or tin plate to relieve surface stress lowers whisker incidence. Another coating issue is **tin pest**.

Solder resist

Areas that should not be soldered to may be covered with a polymer *solder resist* (*solder mask*) coating. The solder resist prevents solder from bridging between conductors and thereby creating short circuits. Solder resist also provides some protection from the environment. Solder resist is typically 20-30 microns thick.

Screen printing

Line art and text may be printed onto the outer surfaces of a PCB by **screen printing**. When space permits, the screen print text can indicate **component designators**, switch setting requirements, test points, and other features helpful in assembling, testing, and servicing the circuit board.

Screen print is also known as the *silk screen*, or, in one sided PCBs, the *red print*.

Lately some digital printing solutions have been developed to substitute the traditional screen printing process. This technology allows printing variable data onto the PCB, including serialization and barcode information for traceability purposes. Also some manufacturers tend to coat their boards in a thin layer of micro-film used to keep electricity from escaping the conductivity of the wire-strips.

Test

Unpopulated boards may be subjected to a *bare-board test* where each circuit connection (as defined in a *netlist*) is verified as correct on the finished board. For high-volume production, a **Bed of nails tester**, a fixture or a **Rigid needle adapter** is used to make contact with copper lands or holes on one or both sides of the board to facilitate testing. A computer will *instruct* the electrical test unit to apply a small voltage to each contact point on the bed-of-nails as required, and verify that such voltage appears at other appropriate contact points. A "short" on a board would be a connection where there should not be one; an "open" is between two points that should be connected but are not. For small- or medium-volume boards, *flying-probe* and *flying-grid* testers use moving test heads to make contact with the copper/silver/gold/solder lands or holes to verify the electrical connectivity of the board under test.

Printed circuit assembly

After the printed circuit board (PCB) is completed, electronic components must be attached to form a functional *printed circuit assembly*^{[5][6]}, or PCA (sometimes called a "printed circuit board assembly" PCBA). In *through-hole* construction, component leads are inserted in holes. In **surface-mount** construction, the components are placed on *pads* or *lands* on the outer surfaces of the PCB. In both kinds of construction, component leads are electrically

and mechanically fixed to the board with a molten metal solder.

There are a variety of **soldering** techniques used to attach components to a PCB. High volume production is usually done with **machine placement** and bulk wave soldering or reflow ovens, but skilled technicians are able to solder very tiny parts (for instance 0201 packages which are 0.02" by 0.01") by hand under a **microscope**, using tweezers and a fine tip **soldering iron** for small volume prototypes. Some parts are impossible to solder by hand, such as **ball grid array** (BGA) packages.

Often, through-hole and surface-mount construction must be combined in a single assembly because some required components are available only in surface-mount packages, while others are available only in through-hole packages. Another reason to use both methods is that through-hole mounting can provide needed strength for components likely to endure physical stress, while components that are expected to go untouched will take up less space using surface-mount techniques.

After the board has been populated it may be tested in a variety of ways:

- While the power is off, **visual inspection**, **automated optical inspection**. JEDEC guidelines for PCB component placement, soldering, and inspection are commonly used to maintain **quality control** in this stage of PCB manufacturing.
- While the power is off, **analog signature analysis**, **power-off testing**.
- While the power is on, **in-circuit test**, where physical measurements (i.e. voltage, frequency) can be done.
- While the power is on, **functional test**, just checking if the PCB does what it had been designed for.

To facilitate these tests, PCBs may be designed with extra pads to make temporary connections. Sometimes these pads must be isolated with resistors. The in-circuit test may also exercise **boundary scan** test features of some components. In-circuit test systems may also be used to program nonvolatile memory components on the board.

In boundary scan testing, test circuits integrated into various ICs on the board form temporary connections between the PCB traces to test that the ICs are mounted correctly. Boundary scan testing requires that all the ICs to be tested use a standard test configuration procedure, the most common one being the Joint Test Action Group (JTAG) standard.

When boards fail the test, technicians may **desolder** and replace failed components, a task known as **rework**.

Protection and packaging

PCBs intended for extreme environments often have a **conformal coating**, which is applied by dipping or spraying after the components have been soldered. The coat prevents corrosion and leakage currents or shorting due to condensation. The earliest conformal coats were **wax**.

Modern conformal coats are usually dips of dilute solutions of silicone rubber, polyurethane, acrylic, or epoxy. Some are engineering plastics **sputtered** onto the PCB in a vacuum chamber.

Many assembled PCBs are **static** sensitive, and therefore must be placed in **antistatic bags** during transport. When handling these boards, the user must be **grounded (earthed)**. Improper handling techniques might transmit an accumulated static charge through the board, damaging or destroying components. Even bare boards are sometimes static sensitive. Traces have become so fine that it's quite possible to blow an etch off the board (or change its characteristics) with a static charge. This is especially true on non-traditional PCBs such as **MCMs** and **microwave** PCBs.

Design

- **Schematic capture** or schematic entry is done through an EDA tool.
- Card dimensions and template are decided based on required circuitry and case of the PCB. Determine the fixed components and **heat sinks** if required.
- Deciding stack layers of the PCB. 4 to 12 layers or more depending on design complexity. **Ground plane** and **Power plane** are decided. Signal planes where signals are routed are in top layer as well as internal layers.^[7]
- **Line impedance** determination using dielectric layer thickness, routing copper thickness and trace-width. Trace separation also taken into account in case of differential signals. **Microstrip**, **stripline** or dual stripline can be used to route signals.
- Placement of the components. Thermal considerations and geometry are taken into account. **Vias** and **lands** are marked.
- Routing the **signal trace**. For optimal **EMI** performance high frequency signals are routed in internal layers between power or ground planes as **power plane** behaves as ground for AC.
- **Gerber File** generation for manufacturing.

Safety certification (US)

Safety Standard UL 796 covers component safety requirements for printed wiring boards for use as components in devices or appliances. Testing analyzes characteristics such as flammability, maximum operating temperature, electrical tracking, heat deflection, and direct support of live electrical parts.

The boards may use organic or inorganic base materials in a single or multilayer, rigid or flexible form. Circuitry construction may include etched, die stamped, precut, flush press, additive, and plated conductor techniques. Printed-component parts may be used.

The suitability of the pattern parameters, temperature and maximum solder limits shall be determined in accordance with the applicable end-product construction and requirements.

"Cordwood" construction



A cordwood module.

Cordwood construction can give large space-saving advantages and was often used with **wire-ended components** in applications where space was at a premium (such as missile guidance and telemetry systems). In 'cordwood' construction, two leaded components are mounted axially between two parallel planes. Instead of soldering the components, they were connected to other components by thin nickel tapes welded at right angles onto the component leads. To avoid shorting together of different interconnection layers, thin insulating cards were placed between them. Perforations or holes in the cards would allow component leads to project through to the next interconnection layer. One disadvantage of this system was that special **nickel** leaded components had to be used to allow the interconnecting welds to be made. Some versions of cordwood construction used single sided PCBs as the interconnection method (as pictured). This meant that normal leaded components could be used.

Before the advent of **integrated circuits**, this method allowed the highest possible component packing density; because of this, it was used by a number of **computer vendors** including **Control Data Corporation**. The cordwood method of construction now appears to have fallen into disuse, probably because high packing densities can be more easily achieved using **surface mount** techniques and integrated circuits.

Multiwire boards

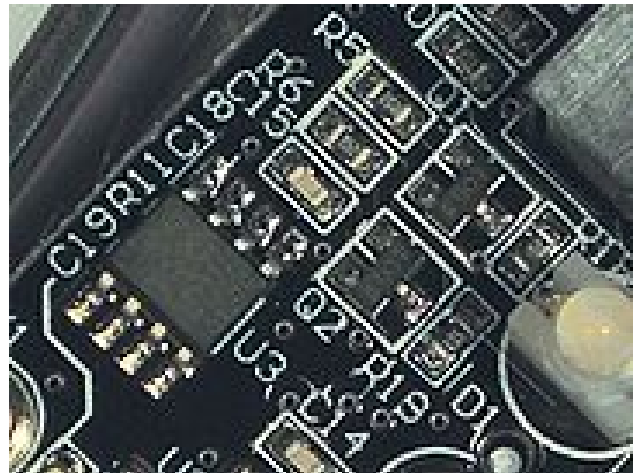
Multiwire is a patented technique of interconnection which uses machine-routed insulated wires embedded in a non-conducting matrix (often plastic resin). It was used during the 1980s and 1990s. (Augat Inc., U.S. Patent 4,648,180)

Since it was quite easy to stack interconnections (wires) inside the embedding matrix, the approach allowed designers to forget completely about the routing of wires (usually a time-consuming operation of PCB design): Anywhere the designer needs a connection, the machine will draw a wire in straight line from one loca-

tion/pin to another. This led to very short design times (no complex algorithms to use even for high density designs) as well as reduced **crosstalk** (which is worse when wires run parallel to each other—which almost never happens in Multiwire), though the cost is too high to compete with cheaper PCB technologies when large quantities are needed.

Surface-mount technology

Main article: [Surface-mount technology](#)

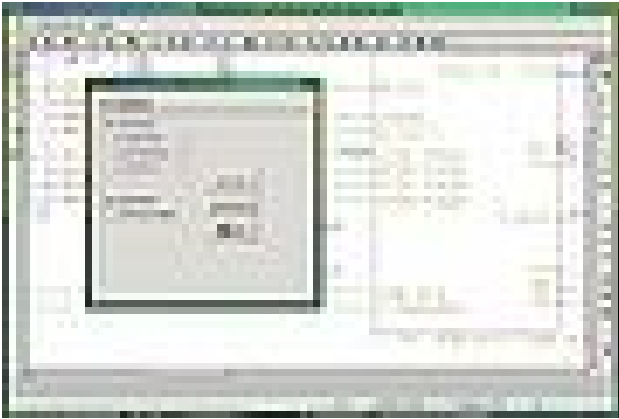


Surface mount components, including **resistors**, **transistors** and an **integrated circuit**

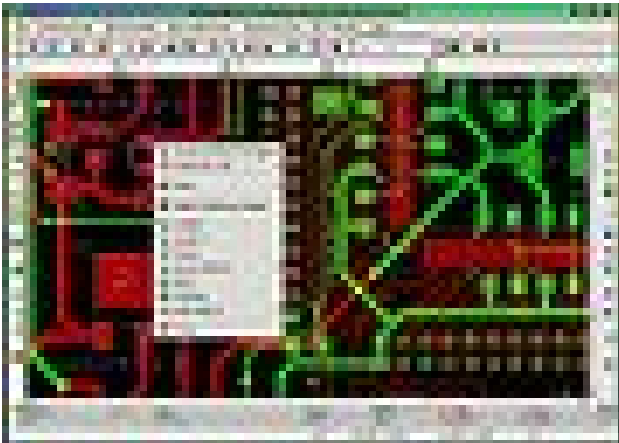
Surface-mount technology emerged in the 1960s, gained momentum in the early 1980s and became widely used by the mid 1990s. Components were mechanically redesigned to have small metal tabs or end caps that could be soldered directly on to the PCB surface. Components became much smaller and component placement on both sides of the board became more common than with through-hole mounting, allowing much higher circuit densities. Surface mounting lends itself well to a high degree of automation, reducing labour costs and greatly increasing production and quality rates. Carrier Tapes provide a stable and protective environment for Surface mount devices (SMDs) which may can be one-quarter to one-tenth of the size and weight, and passive components can be one-half to one-quarter of the cost of corresponding through-hole parts. However, integrated circuits are often priced the same regardless of the package type, because the chip itself is the most expensive part. As of 2006, some wire-ended components, such as small-signal switch diodes, e.g. **1N4148**, are actually significantly cheaper than corresponding SMD versions.

See also

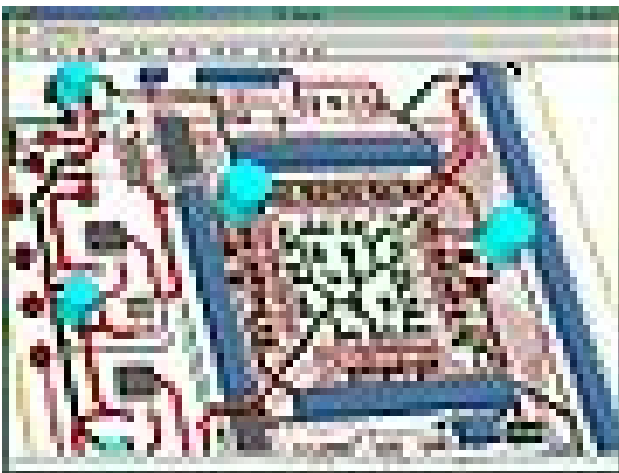
- Breadboard**
- C.I.D.+**
- Hybrid Integrated Circuit**
- Electronic packaging**



Schematic Capture. (KICAD)



PCB layout. (KICAD)



3D View. (KICAD)

- Multi-Chip Module
- Electronic waste
- Electrical conductor
- Signal trace - signal lines on PCBs
- Paul Eisler
- The Occam Process - another process for the manufacturing of PCBs

PCB Materials

- **Laminate materials:**
 - FR-4, the most common PCB material

- FR-2
- Composite epoxy material, CEM-1,5
- Polyimide
- BT-Epoxy
- Cyanate Ester
- PTFE, Polytetrafluoroethylene (Teflon)
- Conductive ink
- Heavy copper

PCB layout software

Main article: [List of EDA companies](#)

- [Altium Designer](#) by [Altium Limited](#)
- [AutoTRAX EDA](#)
- [EAGLE](#) by [Cadsoft](#)
- [Edwinxp](#)
- [FreePCB](#) by [Allan Wright](#) (open-source Win2K/XP)
- [FreeRouting](#) by [Alfons Wirtz](#) (Java, Autorouter with free angle support)
- [Cadstar](#) by [Zuken](#)
- [CR5000](#) by [Zuken](#)
- [Electronics Workbench](#) (now owned by [National Instruments](#))
- [gEDA](#), open-source PCB software project
- [OrCAD](#) by [Cadence](#)
- [Allegro](#) by [Cadence](#)
- [TARGET 3001!](#)
- [Kicad](#), open-source suite
- [PADS](#) by [Mentor Graphics](#)
- [Proteus](#)
- [Board Station](#) by [Mentor Graphics](#)
- [Expedition Enterprise](#) by [Mentor Graphics](#)

References

- [1] http://books.nap.edu/openbook.php?record_id=11515&page=77
- [2] http://epa.gov/dfe/pubs/pwb/tech_rep/p2_report/p2_sec3.htm
- [3] IPC Publication IPC-TR-476A, "Electrochemical Migration: Electrically Induced Failures in Printed Wiring Assemblies," Northbrook, IL, May 1997.
- [4] S.Zhan, M. H. Azarian and M. Pecht, "Reliability Issues of No-Clean Flux Technology with Lead-free Solder Alloy for High Density Printed Circuit Boards", 38th International Symposium on Microelectronics, pp. 367-375, Philadelphia, PA, September 25-29, 2005.
- [5] Ayob M. and Kendall G. (2008) A Survey of Surface Mount Device Placement Machine Optimisation: Machine Classification. *European Journal of Operational Research*, 186(3), pp 893-914 (<http://dx.doi.org/10.1016/j.ejor.2007.03.042>)
- [6] Ayob M. and Kendall G. (2005) *A Triple Objective Function with a Chebychev Dynamic Pick-and-place Point Specification Approach to Optimise the Surface Mount Placement Machine*. *European Journal of*

Operational Research, 164(3), pp 609-626
(<http://dx.doi.org/10.1016/j.ejor.2003.09.034>)

[7] See appendix D of IPC-2251

- Coombs, Clyde F., Jr. (Ed.) (1995). *Printed Circuits Handbook, Fourth Edition*, McGraw-Hill ISBN 0-07-012754-9.
- Coombs, Clyde F., Jr. (Ed.) (2001). *Printed Circuits Handbook, Fifth Edition* ', McGraw-Hill Professional ISBN 0-07-135016-0.
- Coombs, Clyde F., Jr. (Ed.) (2007). *Printed Circuits Handbook, Sixth Edition* ', McGraw-Hill Professional ISBN 0-07-146734-3.

External links

Design guidelines

- PWB/PCB Design – Analog, RF & EMC Considerations in Printed Wiring Board Design
- High Frequency Laminate Information by Chet Guiles
- EMC Design Guideline Collection on the Clemson Vehicular Electronics Laboratory web site
- PCB Design/Layout Tutorial (PDF)

Standards and specifications

- MIL-PRF-31032, Performance Specification Printed Circuit Board/Printed Wiring Board

- MIL-PRF-55110, Performance Specification for Rigid Printed Circuit Board/Printed Wiring Board
- MIL-PRF-50884, Performance Specification for Flexible and Rigid-Flexible Printed Circuit Board/Printed Wiring Board

Do-it-yourself (DIY) guides

- Easy Printed Circuit Board Fabrication Using Laser Printer Toner Transfer
- How to make PCBs at home in 1 hour & W I T H O U T special materials ?
- PCBs Fabrication Methods
- Making PCBs using DIY cnc hardware
- Making PCBs with a laser printer and photo paper, no UV machine needed anymore.
- More detailed PCB DIY making with laser printer
- Making PCBs by hand using an etch-resist pen (no printer or special paper needed)
- Open Source Hardware Initiative ([open hardware](#)).

Others

- PCB Glossary of Terms
- First PCB Patent - Patent N° 2,756,485 - Process of Assembling Electrical Circuits (pdf)
- General Information on Printed Board Programs
- PCB Trace Impedance Calculator
- Free Online PCB Gerber File Viewer

Retrieved from "http://en.wikipedia.org/wiki/Printed_circuit_board"

Categories: Electrical engineering, Electronics manufacturing, Electronic engineering

This page was last modified on 5 January 2010 at 02:15. Text is available under the Creative Commons Attribution-ShareAlike License; additional terms may apply. See [Terms of Use](#) for details. Wikipedia® is a registered trademark of the Wikimedia Foundation, Inc., a non-profit organization.[Contact us](#) [Privacy policy](#) [About Wikipedia](#) [Disclaimers](#)