Printed circuit board

A **printed circuit board**, or **PCB**, is used to mechanically support and electrically connect [electronic components](http://en.wikipedia.org/wiki/Electronic_component) using [conductive](http://en.wikipedia.org/wiki/Electrical_conductor" \o "Electrical conductor)pathways, tracks or signal traces [etched](http://en.wikipedia.org/wiki/Industrial_etching) from copper sheets [laminated](http://en.wikipedia.org/wiki/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**). Printed circuit boards are used in virtually all but the simplest commercially-produced electronic devices.

PCBs are inexpensive, and can be highly reliable. They require much more layout effort and higher initial cost than either [wire wrap](http://en.wikipedia.org/wiki/Wire_wrap) or[point-to-point construction](http://en.wikipedia.org/wiki/Point-to-point_construction), but are much cheaper and faster for high-volume production; the production and soldering of PCBs can be done by totally automated equipment. Much of the electronics industry's PCB design, assembly, and quality control needs are set by standards that are published by the [IPC](http://en.wikipedia.org/wiki/IPC_(electronics)) organization.

## Manufacturing

### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=3)]Materials

A PCB as a design on a computer (left) and realized as a board assembly populated with 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.The Component Side of a PCB in a computer mouse; some examples for common components and their [reference designations](http://en.wikipedia.org/wiki/Electronic_symbols#Reference_designations) on the silk screen.

Conducting layers are typically made of thin copper foil. Insulating layers [dielectric](http://en.wikipedia.org/wiki/Dielectric) are typically laminated together with [epoxy resin](http://en.wikipedia.org/wiki/Epoxy_resin)[prepreg](http://en.wikipedia.org/wiki/Prepreg). The board is typically coated with a solder mask that is green in color. Other colors that are normally available are blue, black, white 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](http://en.wikipedia.org/wiki/Polytetrafluoroethylene) (Teflon), FR-4, FR-1, CEM-1 or CEM-3. Well known prepreg materials used in the PCB industry are [FR-2](http://en.wikipedia.org/wiki/FR-2) (Phenolic cotton paper), FR-3 (Cotton paper and epoxy), [FR-4](http://en.wikipedia.org/wiki/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.

FR-4 is by far the most common material used today. The board with copper on it is called "copper-clad laminate".

Copper foil thickness can be specified in ounces per square foot or micrometres. One ounce per square foot is 1.344 mils or 34 micrometres.

### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=4)]Patterning (etching)

The vast majority of printed circuit boards are made by bonding a layer of copper over the entire substrate, sometimes on both sides, (creating a "blank PCB") then removing unwanted copper after applying a temporary mask (e.g. 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](http://en.wikipedia.org/wiki/Electroplating) steps. The PCB manufacturing method primarily depends on whether it is for production volume or sample/prototype quantities.

#### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=5)]Commercial (production quantities, usually PTH)

* [silk screen printing](http://en.wikipedia.org/wiki/Silk_screen) -the main commercial method.
* Photographic methods. Used when fine linewidths are required.

#### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=6)]Hobbyist/prototype (small quantities, usually not PTH)

* Laser-printed resist: Laser-print onto paper (or wax paper), heat-transfer with an iron or modified laminator onto bare laminate, then etch.
* Print onto transparent film and use as photomask along with photo-sensitized boards. (i.e. pre-sensitized boards), Then etch. (Alternatively, use a film photoplotter).
* Laser resist ablation: Spray black paint onto copper clad laminate, place into CNC laser plotter. The laser raster-scans the PCB and ablates (vaporizes) the paint where no resist is wanted. Etch. (Note: laser copper ablation is rarely used and is considered experimental.)
* Use a CNC-mill with a spade-shaped (i.e. 45-degree) cutter or miniature end-mill to route away the undesired copper, leaving only the traces.

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

1. [**Silk screen printing**](http://en.wikipedia.org/wiki/Silk_screen) 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](http://en.wikipedia.org/wiki/Hybrid_circuit).
2. [**Photoengraving**](http://en.wikipedia.org/wiki/Photoengraving) uses a photomask and developer to selectively remove a [photoresist](http://en.wikipedia.org/wiki/Photoresist) coating. The remaining photoresist protects the copper foil. Subsequent etching removes the unwanted copper. The photomask is usually prepared with a [photoplotter](http://en.wikipedia.org/wiki/Photoplotter) from data produced by a technician using CAM, or [computer-aided manufacturing](http://en.wikipedia.org/wiki/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. [**PCB milling**](http://en.wikipedia.org/wiki/PCB_Milling) 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](http://en.wikipedia.org/wiki/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](http://en.wikipedia.org/wiki/HPGL) or [Gerber file](http://en.wikipedia.org/wiki/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 through holes but still single sided were made with a process like this.[General Electric](http://en.wikipedia.org/wiki/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](http://en.wikipedia.org/wiki/Plating)-through of the holes (to produce conductive [vias](http://en.wikipedia.org/wiki/Via_(electronics)" \o "Via (electronics))) in the circuit board.

PCB copper electroplating machine for adding copper to the in-process PCB

 PCB's in process of adding copper via electroplating

The dimensions of the copper conductors of the printed circuit board is related to the amount of current the conductor must carry. Each trace consists of a flat, narrow part of the [copper](http://en.wikipedia.org/wiki/Copper)foil that remains after etching. Signal traces are usually narrower than power or ground traces because their current carrying requirements are usually much less. In a multi-layer board one entire layer may be mostly solid copper to act as a [ground plane](http://en.wikipedia.org/wiki/Ground_plane) for shielding and power return. For printed circuit boards that contain [microwave](http://en.wikipedia.org/wiki/Microwave) circuits, [transmission lines](http://en.wikipedia.org/wiki/Transmission_line) can be laid out in the form of [stripline](http://en.wikipedia.org/wiki/Stripline) and [microstrip](http://en.wikipedia.org/wiki/Microstrip) with carefully controlled dimensions to assure a consistent [impedance](http://en.wikipedia.org/wiki/Characteristic_impedance). In radio-frequency circuits the [inductance](http://en.wikipedia.org/wiki/Inductance) and [capacitance](http://en.wikipedia.org/wiki/Capacitance) of the printed circuit board conductors can be used as a delibrate part of the circuit design, obviating the need for additional discrete components.

### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=7)]Etching

Chemical etching is done with ferric chloride, ammonium persulfate, or sometimes hydrochloric acid. For PTH (plated-through holes), additional steps of electroless deposition are done after the holes are drilled, then copper is electroplated to build up the thickness, the boards are screened, and plated with tin/lead. The tin/lead becomes the resist leaving the bare copper to be etched away.

### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=8)]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.

### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=9)]Drilling

Holes through a PCB are typically drilled with tiny drill bits made of solid [tungsten carbide](http://en.wikipedia.org/wiki/Tungsten_carbide). The drilling is performed by [automated](http://en.wikipedia.org/wiki/Automation) [drilling machines](http://en.wikipedia.org/wiki/Milling_machine) 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](http://en.wikipedia.org/wiki/Excellon_file" \o "Excellon file)". 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](http://en.wikipedia.org/wiki/Via_(electronics)" \o "Via (electronics)). 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 partly due to embedded glass, which is harder than 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](http://en.wikipedia.org/wiki/Laser). 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* 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.

### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=10)]Exposed conductor plating and coating

PCBs[[2]](http://en.wikipedia.org/wiki/Printed_circuit_board" \l "cite_note-1) are plated with solder, tin, or gold over nickel as a resist for [etching](http://en.wikipedia.org/wiki/Etching) away the unneeded underlying copper.[[3]](http://en.wikipedia.org/wiki/Printed_circuit_board#cite_note-2)

After PCBs are etched and then rinsed with water, the soldermask is applied, and then any exposed copper is coated with solder, nickel/gold, or some other anti-corrosion coating.[[4]](http://en.wikipedia.org/wiki/Printed_circuit_board#cite_note-3)[[5]](http://en.wikipedia.org/wiki/Printed_circuit_board#cite_note-4)

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](http://en.wikipedia.org/wiki/HASL)). The HASL finish prevents oxidation from the underlying copper, thereby guaranteeing a solderable surface.[[6]](http://en.wikipedia.org/wiki/Printed_circuit_board#cite_note-indiumcorporation-5) This solder was a[tin](http://en.wikipedia.org/wiki/Tin)-[lead](http://en.wikipedia.org/wiki/Lead) alloy, however new solder compounds are now used to achieve compliance with the [RoHS](http://en.wikipedia.org/wiki/RoHS" \o "RoHS) directive in the [EU](http://en.wikipedia.org/wiki/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](http://en.wikipedia.org/wiki/IAg)), immersion tin, electroless nickel with immersion gold coating ([ENIG](http://en.wikipedia.org/wiki/ENIG)), and direct [gold plating](http://en.wikipedia.org/wiki/Gold_plating) (over nickel). [Edge connectors](http://en.wikipedia.org/wiki/Edge_connector), placed along one edge of some boards, are often nickel plated then [gold plated](http://en.wikipedia.org/wiki/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.[[7]](http://en.wikipedia.org/wiki/Printed_circuit_board#cite_note-6)[[8]](http://en.wikipedia.org/wiki/Printed_circuit_board#cite_note-7) 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](http://en.wikipedia.org/wiki/Tin_pest), the transformation of tin to a powdery allotrope at low temperature.[[9]](http://en.wikipedia.org/wiki/Printed_circuit_board#cite_note-8)

### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=11)]Solder resist

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

### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=12)]Screen printing

Line art and text may be printed onto the outer surfaces of a PCB by [screen printing](http://en.wikipedia.org/wiki/Screen-printing). When space permits, the screen print text can indicate [component designators](http://en.wikipedia.org/wiki/Circuit_diagram#European_and_Australian_codes), 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.

### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=13)]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](http://en.wikipedia.org/wiki/Bed_of_nails_tester), a fixture or a [Rigid needle adapter](http://en.wikipedia.org/wiki/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*](http://en.wikipedia.org/wiki/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.

### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=14)]Printed circuit assembly

After the printed circuit board (PCB) is completed, electronic components must be attached to form a functional *printed circuit assembly*,[[10]](http://en.wikipedia.org/wiki/Printed_circuit_board" \l "cite_note-9)[[11]](http://en.wikipedia.org/wiki/Printed_circuit_board#cite_note-10) or PCA (sometimes called a "printed circuit board assembly" PCBA). In [*through-hole*](http://en.wikipedia.org/wiki/Through-hole_technology) construction, component leads are inserted in holes. In [*surface-mount*](http://en.wikipedia.org/wiki/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](http://en.wikipedia.org/wiki/Soldering) techniques used to attach components to a PCB. High volume production is usually done with [machine placement](http://en.wikipedia.org/wiki/SMT_placement_equipment) 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 in. by 0.01 in.)[[12]](http://en.wikipedia.org/wiki/Printed_circuit_board#cite_note-11) by hand under a [microscope](http://en.wikipedia.org/wiki/Microscope), using tweezers and a fine tip[soldering iron](http://en.wikipedia.org/wiki/Soldering_iron) for small volume prototypes. Some parts are impossible to solder by hand, such as [ball grid array](http://en.wikipedia.org/wiki/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](http://en.wikipedia.org/wiki/Visual_inspection), [automated optical inspection](http://en.wikipedia.org/wiki/Automated_optical_inspection). [JEDEC](http://en.wikipedia.org/wiki/JEDEC) guidelines for PCB component placement, soldering, and inspection are commonly used to maintain[quality control](http://en.wikipedia.org/wiki/Quality_control) in this stage of PCB manufacturing.
* While the power is off, [analog signature analysis](http://en.wikipedia.org/wiki/Analog_signature_analysis), [power-off testing](http://en.wikipedia.org/wiki/Power-off_testing).
* While the power is on, [in-circuit test](http://en.wikipedia.org/wiki/In-circuit_test), where physical measurements (i.e. voltage, frequency) can be done.
* While the power is on, [functional test](http://en.wikipedia.org/wiki/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](http://en.wikipedia.org/wiki/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](http://en.wikipedia.org/wiki/JTAG)) standard. The [JTAG](http://en.wikipedia.org/wiki/JTAG) test architecture provides a means to test interconnects between integrated circuits on a board without using physical test probes. [JTAG](http://en.wikipedia.org/wiki/JTAG) tool vendors provide various types of stimulus and sophisticated algorithms, not only to detect the failing nets, but also to isolate the faults to specific nets, devices, and pins.[[13]](http://en.wikipedia.org/wiki/Printed_circuit_board#cite_note-12)

When boards fail the test, technicians may [desolder](http://en.wikipedia.org/wiki/Desoldering" \o "Desoldering) and replace failed components, a task known as [*rework*](http://en.wikipedia.org/wiki/Rework_(electronics)).

### [[edit](http://en.wikipedia.org/w/index.php?title=Printed_circuit_board&action=edit&section=15)]Protection and packaging

PCBs intended for extreme environments often have a [conformal coating](http://en.wikipedia.org/wiki/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](http://en.wikipedia.org/wiki/Wax); modern conformal coats are usually dips of dilute solutions of silicone rubber, polyurethane, acrylic, or epoxy. Another technique for applying a conformal coating is for plastic to be [sputtered](http://en.wikipedia.org/wiki/Sputter_deposition) onto the PCB in a vacuum chamber. The chief disadvantage of conformal coatings is that servicing of the board is rendered extremely difficult.[[14]](http://en.wikipedia.org/wiki/Printed_circuit_board#cite_note-13)

Many assembled PCBs are [static](http://en.wikipedia.org/wiki/Electrostatic_discharge) sensitive, and therefore must be placed in [antistatic bags](http://en.wikipedia.org/wiki/Antistatic_bag) during transport. When handling these boards, the user must be [grounded (earthed)](http://en.wikipedia.org/wiki/Ground_(electricity)). 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](http://en.wikipedia.org/wiki/Multi-Chip_Module) and microwave PCBs.

. SOLDERING

Soldering is a process in which two or more metal items are joined together by melting and flowing a filler metal into the joint, the filler metal having a relatively low melting point. Soft soldering is characterized by the melting point of the filler metal, which is below 400 °C (752 °F). The filler metal used in the process is called solder. Soldering is distinguished from brazing by use of a lower melting-  
temperature filler metal; it is distinguished from welding by the base metals not being melted during the joining process. In a soldering process, heat is applied to  
the parts to be joined, causing the solder to melt and be drawn into the joint by capillary action and to bond to the materials to be joined by wetting action. After the metal cools, the resulting joints are not as strong as the base metal, but have adequate strength, electrical conductivity, and water-tightness for many uses. Soldering is an ancient technique mentioned in the Bible and there is evidence that it was employed up to 5000 years ago in Mesopotamia.

Soldering filler materials are available in many different alloys for differing applications. In electronics assembly, the eutectic alloy of 63% tin and 37% lead (or 60/40, which is almost identical in performance to the eutectic) has been the alloy of choice. Other alloys are used for plumbing, mechanical assembly, and other applications.

In high-temperature metal joining processes (welding, brazing and soldering), the primary purpose of flux is to prevent oxidation of the base and filler materials. Tin-lead  
solder, for example, attaches very well to copper, but poorly to the various oxides of copper, which form quickly at soldering temperatures. Flux is a substance which is  
nearly inert at room temperature, but which becomes strongly reducing at elevated temperatures, preventing the formation of metal oxides. Secondarily, flux acts as a  
wetting agent in the soldering process, reducing the surface tension of the molten solder and causing it to better wet out the parts to be joined