Key Processes used to Build a Quality Printed Circuit Board

Key Processes

There are over forty different processes and hundreds of individual steps needed to build a printed circuit board (PCB). It is often difficult to get an overall sense of how circuit boards are produced and the importance of each individual processes, because not all boards are built the same way. While all of the processes used to build a printed circuit board are important, some of the processes are key to understanding how a PCB is built. In this section, we will cover these key processes and how they affect a finished board.

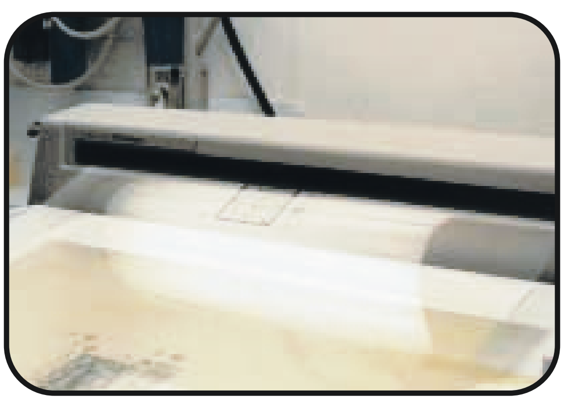
Phototooling

Phototooling is an essential part of a number of processes including: Inner layer printing, hardboard printing, soldermask, nomenclature, deep and immersion gold and carbon ink. The phototool is critical because it determines how a customer s circuitry gets built into a board. This means that each customer order must have its own set of phototools. If the phototools are wrong, the board will be wrong.

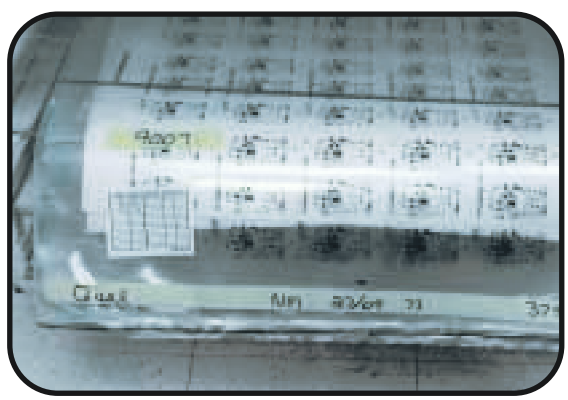
When a customer orders a board from JMACI, they send the specifications and circuitry in an electronic format called Gerber data. This Gerber data is read by computer programs in the Engineering department which converts this information and creates the necessary phototools. The device used to create the phototools is called a photo plotter.

A photo plotter is essentially a specialized printer. The photo plotter uses the electronic data to expose a photosensitive film. After being exposed, the film is developed. The pattern on the film contains the customer s circuitry.

Phototools consist of a sheet of clear mylar with a photosensitive film called emulsion on one side. The emulsion is very thin and is subject to damage from improper handling and storage. A scratch in the emulsion will produce a defect on the panel. Therefore, great care is taken to properly store and handle phototools.



A phototool as it exits the plotter.



Storage and handling are critical to making sure phototools are not damaged.

Exposing and Developing

When making a phototool, the areas of the emulsion that have been exposed to light in the photo plotter remain on the mylar after developing. The areas that were not exposed to light are washed away. This is what we mean when we say that something is photosensitive. If the material is exposed to light it causes the photosensitive material to change chemically. The areas that have not been exposed to light are not changed. When the film is processed through the developer, the developing chemicals leave the exposed material and remove the unexposed material.

This same process is used in printing the customer s image onto a circuit board. There are three major steps to this operation. The first is to apply the photosensitive material to the board. For inner layers, an even layer of liquid is applied to the surface. In hardboard printing, a dry film is laminated to the panel using heat and pressure. In the Soldermask area, soldermask is applied to the panel using a screen printer or an electrostatic sprayer. Despite the differences in the application process, the goal is to deposit a layer of photosensitive material onto the panel.

In the hardboard print area, resist is applied to the panel using a vacuum lamination process.

Resist is applied to inner layers as they pass through a sheet of liquid resist. The coating is then cured in an oven.

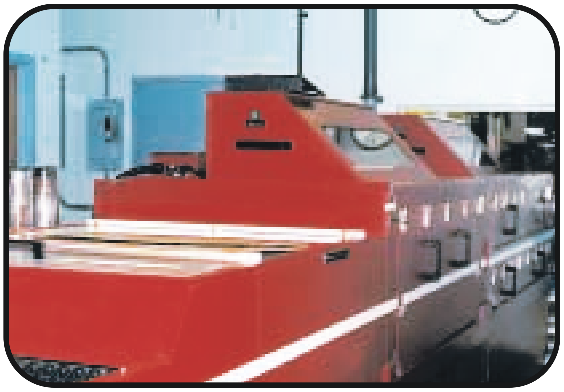
Once the photosensitive material has been applied to the panel, the phototool is used to expose the areas where we want the material to remain after developing. The goal is to exactly duplicate the phototool on the panel. In the areas where there is emulsion on the phototool, the light will be blocked. Light will pass through the clear areas on the phototool exposing the photosensitive material. To get the best possible image, the emulsion on the phototool must be next to the panel. This keeps the light from spreading as it passes through the mylar, blurring the image. Most of the printers used at JMACI expose photosensitive material by applying a specific amount of light energy to the panel. Light energy is the intensity or brightness of a light over a specific period

Phototools are cleaned and held against a glass frame by vacuum. Panels or layers are placed between two phototools and exposed to ultraviolet light.

* The lights in the print areas are filtered to remove ultraviolet light. Black & white photographs are used to increase the clarity of the image.

of time. By controlling the amount of light energy, the exposure process can be properly controlled.

The final step in the process is developing. The developer typically consists of a series of chambers containing chemicals which remove the unexposed photosensitive material and leave the areas that were exposed. Cleanliness is very important to the printing process. If dirt or loose copper is on the panel while it is being exposed, it will change the customer s circuit pattern. If this occurs in the inner layer area it will cause an open between circuit connections. The dirt will block the light from the resist and it will wash off in the developer. All the areas of the layer that do not have resist will be etched away during the etch process.



A developer completes the printing process by removing the unexposed resist.

All photographic printing areas at ACI are inside cleanrooms to control errors due to dust and dirt.

Cleanliness is very important to the quality of a finished circuit board.

The opposite is true in the hardboard area. Here the areas that do not have resist will be plated. Dirt will expose more copper during the plating process. Since it is the plating that protects the circuitry during the etch process, the board will have a short circuit across the circuit connections.

There are several important things to remember about exposing and developing. Cleanliness is very important to the quality of a finished printed circuit board. The phototool must be handled with care so the emulsion is not scratched or damaged. The image must be printed with the emulsion of the phototool next to the panel and exposed with the correct light energy. Finally, the photosensitive material must be applied correctly and the developer must be operating correctly.

Etching

Etching is the process of removing copper from selected areas of the panel. Etching is the process that turns a layer of copper into a pattern that matches the customer s circuitry. The copper that makes up the customer s circuitry must be protected from the chemicals in the etching solution. This is done one of two ways.

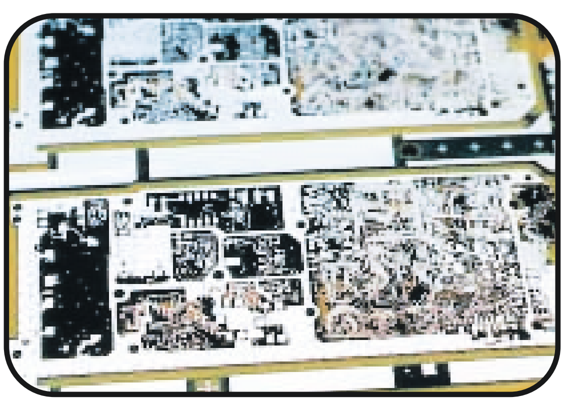
In the Inner Layer area resist is printed on the layer. The resist image matches the customer s circuit connections. When the layer enters the etch chamber the etchant attacks

The resist on inner layers protects the copper under it from the etching process.

The resist is then stripped from the panel exposing the bare copper. The bare copper is removed in the etch chamber. In this case, the solder or tin plating rather than the resist is what protects the copper under it from the etch solution.

the bare copper. The resist protects the copper under it from being etched away.

Once the inner layers are etched and laminated into a panel, the panel is drilled and run through the sensitize line to plate a thin layer of copper in the holes. A resist image is then printed on the panel. Unlike the resist that is printed on the inner layers, this resist covers the areas that will eventually be removed during the etch process. The panel is then electroplated with a layer of copper followed by a layer of either solder or tin. The plated areas on the panel then match the customer s circuitry or pattern. This is why it is often called pattern plating.



The solder plating from the pattern plating process protects the copper under it from the etchant.

The parameters that are critical to the etch process include: the chemistry of the etch solution, the speed of the conveyor, the thickness of the copper being etched and the pressure of the etch spray. If there is something wrong with the etch process it will usually show up in line width. If the panel or layer is underetched, the line widths will be bigger than the customer s circuitry. If the panel is overetched, the line widths will be too small or will disappear altogether.

Plating

Plating is a process that deposits a relatively thin layer of metal onto a surface. There are two basic types of plating needed to make a multi-layer circuit board. The first is electroless. It is called electroless because it doesn t require an electrical current to work. The sensitize line uses the electroless plating process. The advantage of electroless plating is that any surface which is properly prepared will get plated with the desired metal. Electroless plating is used to plate metal on a surface that is not conductive.

Key Processes

Electroless plating doesn t require an electrical connection to the panel.

The drawback to electroless plating is that it is only useful to deposit a very thin layer of metal. It is used in the process of building a printed circuit board to coat the inside of a drilled hole with copper.

The goal of the sensitize line is to make the inside of a drilled hole conductive so one layer can be connected to another layer.

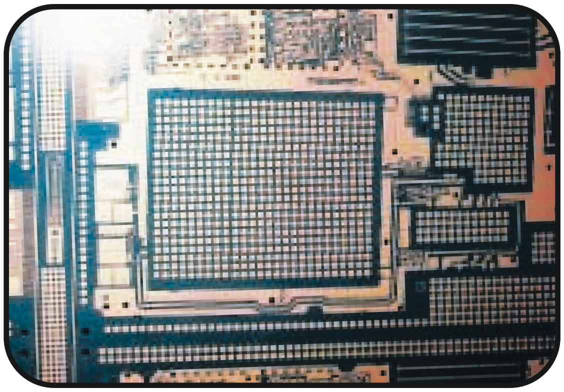
The other type of plating used at JMACI is electroplating. Electroplating uses electrical current to deposit metal on the conductive areas of the panel. Any surface that is nonconductive will not have metal deposited on its surface. The areas to be plated can be controlled by applying resist to a panel prior to electroplating. The resist image printed on the panel in hardboard print exposes only the surfaces of the panel that will eventually become the customer s circuitry. The areas covered by the resist are not plated. This process is called pattern plating.

Electroplating is used because we can precisely control the areas to be plated and because it has the ability to deposit metal quickly. These two features reduce the cost and the time required to build a printed circuit board. It is especially important when gold is being plated onto a panel. Since gold is very expensive it is important to plate only the areas on the board where the customer needs gold plating.

Both plating processes require the panel be chemically conditioned before plating. In the sensitize line the panel must be processed through over 20 different baths before it is ready to accept plating. Of course, the chemistry of the plating baths is critical to proper plating.

Electroplating requires an electrical connection to the panel.

Electroplating has the additional requirement of providing the proper amount of electrical current. The amount of current required for plating is determined primarily by the surface area to be plated. To obtain an even thickness of plating on the panel the current must be evenly distributed. In an ideal world, the customer s circuitry would spread the current evenly throughout the panel. This is not always the case. This is why the Engineering department adds robber dots to some jobs. The robber dots evenly distribute the plating current across the entire panel.

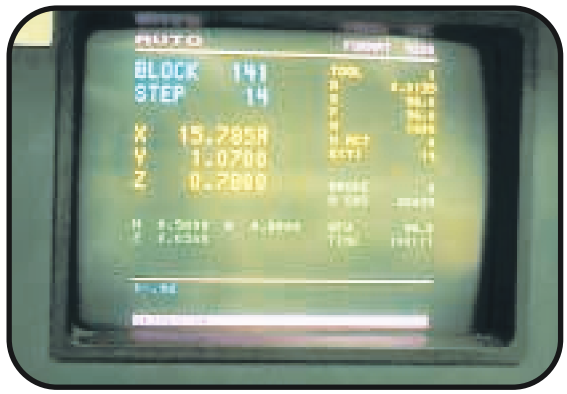


Robber dots are used to evenly distribute the plating current.

Drilling and Routing

When a customer sends the Gerber data to JMACI it contains information on the size and location of each hole. In a separate file is information on the outer dimensions of the finished board. This information is first checked by JMACI s Engineering department. The engineer checks the data to make sure that each hole lines up with the customer s circuitry. Once the engineer is sure the drill data is correct, it is turned into a drilling program. The drilling program instructs the computerized driller where to drill the holes and what size drill bit to use. The same thing is done for the routing program.

The drilling of holes and the routing of panels are very important aspects of building a printed circuit board. Each hole must be drilled in the proper location on the panel. If the hole is off by just a few thousands of an inch, the board will be rejected. Routing is equally important because routing determines the final dimensions of a finished board. The holes on the board must line up with the circuit paths and the outside dimensions must be correct. If the holes are in the wrong location when a customer inserts a component, it will not make a reliable electrical connection. If the board dimensions are not correct, the board may not fit into the customer s housing. To meet these exacting customer requirements JMACI uses computer-controlled drillers and routers.



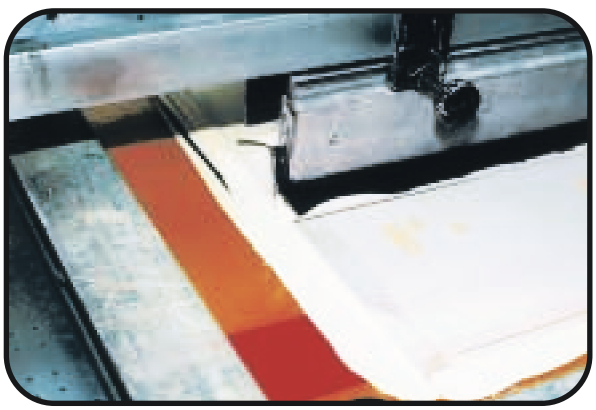
The precise location of each hole or rout path is specified by programs generated in Process Engineering. They are then down loaded into a computer-controlled driller or router.

Drillers and routers use a coordinate system to determine the correct location of holes to be drilled or the path of a rout bit. This system consists of three coordinates: X, Y and Z. The X coordinate determines the location left to right. The Y coordinate determines the position back and forward and the Z coordinate determines the movement up and down. A typical drilling instruction might tell the drill head to pick up a .062 drill bit, raise the bit up, move to location X = 1 inch Y = 2 inches, lower the drill bit and drill the hole.

Drillers and routers are precision machines. They are able to control the location and movement of bits to .001 of an inch. This level of precision is necessary to maintain the correct registration between drilled holes and the circuit pads on the board.

Silk Screening

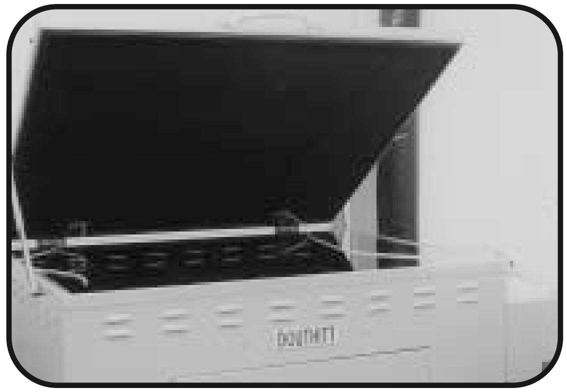
Silk screening is a process which prints an image directly onto a panel. The process is used to apply soldermask, nomenclature, peelable mask and carbon ink pads. As its name implies, printing is done using a screen. Each screen is made up of rows of holes which can either be closed or open. During printing a squeegee is used to force ink through the open holes onto the panel.



As the squeegee passes over the screen, ink is forced through the open holes and is deposited on the panels.

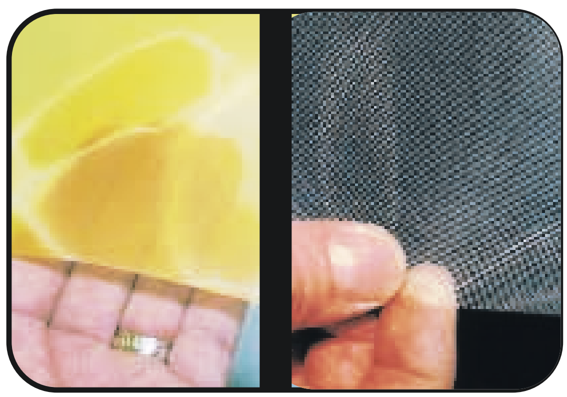
When making a screen, a photo-sensitive emulsion is placed on the underside of the screen. The emulsion is selectively exposed using a phototool (see Exposing and Developing) then developed. This is how the desired image is transferred to the screen.

There are a number of factors which influence the quality of the screen printing oper-



An exposure unit transfers the phototool s image to the screen.

ation. One important factor is the number and size of the holes in a screen. This is specified as mesh. The higher the mesh the greater the number of holes in the screen. This is desirable if what you are trying to print contains thin lines and a lot of detail. The drawback is that since the holes are small, not as much ink can pass through the screen. Screens with a mesh over 200 threads per inch are used in nomenclature.



Examples of a fine and coarse screen mesh.

The opposite is true of peelable mask where you are trying to get a build up of ink and have very little detail. Peelable mask uses meshes under 100.

Other factors include the viscosity of the ink. Viscosity is the measurement of how easily the ink will flow through the holes in the screen. The tension of the screen, the distance between the screen and the panel to be printed, pressure applied to the squeegee and even the sharpness of the squeegee blade are important for getting a good image.

Maintaining Registration

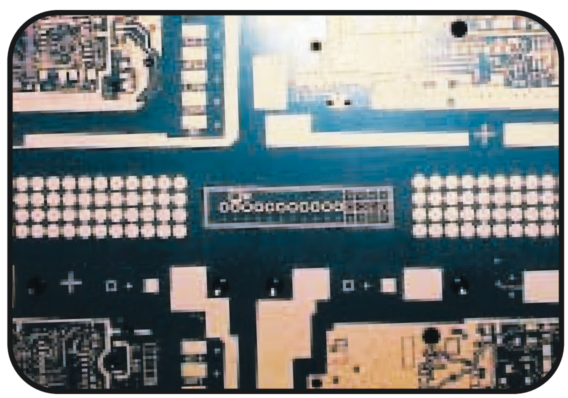
When we talk about the registration on a printed circuit board, we are describing how well each circuit layer lines up with the other circuit layers and how the drilled holes line up with the circuitry. Perfect registration is when the pads on each layer are directly on top of each other and the hole is drilled in the center of the pad. Misregistration is when one or more layers are not lined up with the other layers or the holes are drilled off center.

Perfect registration is when the pads on each layer are directly on top of each other and the hole is drilled in the center of the pad.

Unlike the other processes described in this section, registration is not a single process but a combination of individual steps. The first step is in the Process Engineering area. A process engineer checks the customer s data on Computer Aided Design (CAD) work stations. One of the things that is verified is that all of the circuit pads and holes are correctly registered. If they are not, it is guaranteed the finished parts will be misregistered.

Process Engineering also performs an important adjustment to the circuitry for the inner layers. A multi-layer circuit board is made up of several individual layers that are laminated together using heat and pressure. When a panel has been laminated and cooled to room temperature, the inner layers are slightly smaller than they were before lamination. This means the entire circuit pattern is about 1% smaller than it should be. To compensate for this shrinkage, the process engineer changes the scale of the artwork for each inner layer. The exact percentage is determined by the type of materials used to build the panel and experience from previous orders.

As the customer s board is panelized (several boards are put onto a single panel) a registration coupon is added to the corners of the panel. The registration coupon is used to verify that registration is maintained at different steps in the manufacturing process.



Registration coupons are used to check registration during the manufacturing process.

For example, after lamination the coupon is removed from the panel, placed in a mold and ground down so each layer is visible under a microscope. This process is called microsectioning.

Microsectioning checks the registration of the inner layers. The registration coupons are also used as a visual check during the hardboard printing process to verify that the image is properly registered to the drilled holes.

Registration tooling holes are an important part of making sure that registration is correct. Registration tooling holes are punched in each of the layers. The holes fit on registration pins that hold the layer or panel in a precise location. The registration holes and pins are used in the Inner Layer Printing, Lamination, Drilling, Hardboard Printing, Soldermask and Nomenclature departments. By punching holes in the layers and using fixture pins at each point in the manufacturing process which directly affects registration, the layer or panel is physically placed in the proper position.

As mentioned earlier, the phototools used to print the circuit image onto a panel are made up of mylar backing and a photosensitive emulsion. One of the characteristics of the mylar backing is that it changes size with changes in temperature and humidity. If the size of the phototool is allowed to change, it will change the scale of the artwork and will cause misregistration.

Temperature and humidity are tightly controlled and monitored to minimize misregistration due to changes in the size of phototools.

To compensate for this problem, each area that stores or uses phototools has special temperature and humidity controls. A chart recorder is used to make sure the proper temperature and humidity is maintained.

Meeting an Impedance Specification

Boards that have an impedance specification are usually used for radio frequency or high speed digital circuits. Examples of products which have this type of circuitry are cellular phones, pagers and computers.

Impedance is specified in ohms. An ohm is a unit of measure which describes a circuit's ability to control the flow of current. In a direct current (DC) circuit, the ability to control the flow of current is called resistance. A DC measurement of a circuit will tell you only the resistance of the circuit.

For PCBs, capacitance is a major factor in determining a board's impedance. The larger the capacitance the lower the impedance. When engineers design a board to meet a specific impedance requirement, they are changing the factors that influence the capacitance. This is why it is important to have an understanding of how capacitance is controlled.

The simplest definition of a capacitor is: two conductors separated by an insulator. There are three factors that influence the value of a capacitor. The first is trace width. The greater the area of the conductors, the greater the capacitance. This factor is controlled by maintaining the proper line width during the etch process.

The second factor is insulation thickness. The smaller the distance the greater the

Line widths are measured to make sure that the circuity is not over or underetched.

capacitance. This is controlled by selecting the proper core and adjusting the thickness of the B-stage used in lamination.

The third factor is the dielectric constant of the insulator separating the conductors. Air has a dielectric constant of 1. The typical dielectric constant for PCBs is 4.4. This means that if a capacitor has a value of 1 pico farad using air as an insulator, it will have a value of 4.4 pico farads after lamination. Soldermask also has a dielectric constant greater than air. This is why impedance readings drop about 8% after the soldermask is applied.

Selection of the correct inner layer material and the B-stage used in the layup process is critical to meeting an impedance specification.

Impedance is controlled by using the proper core and B-stage in lamination and controlling the line width during the etch process.