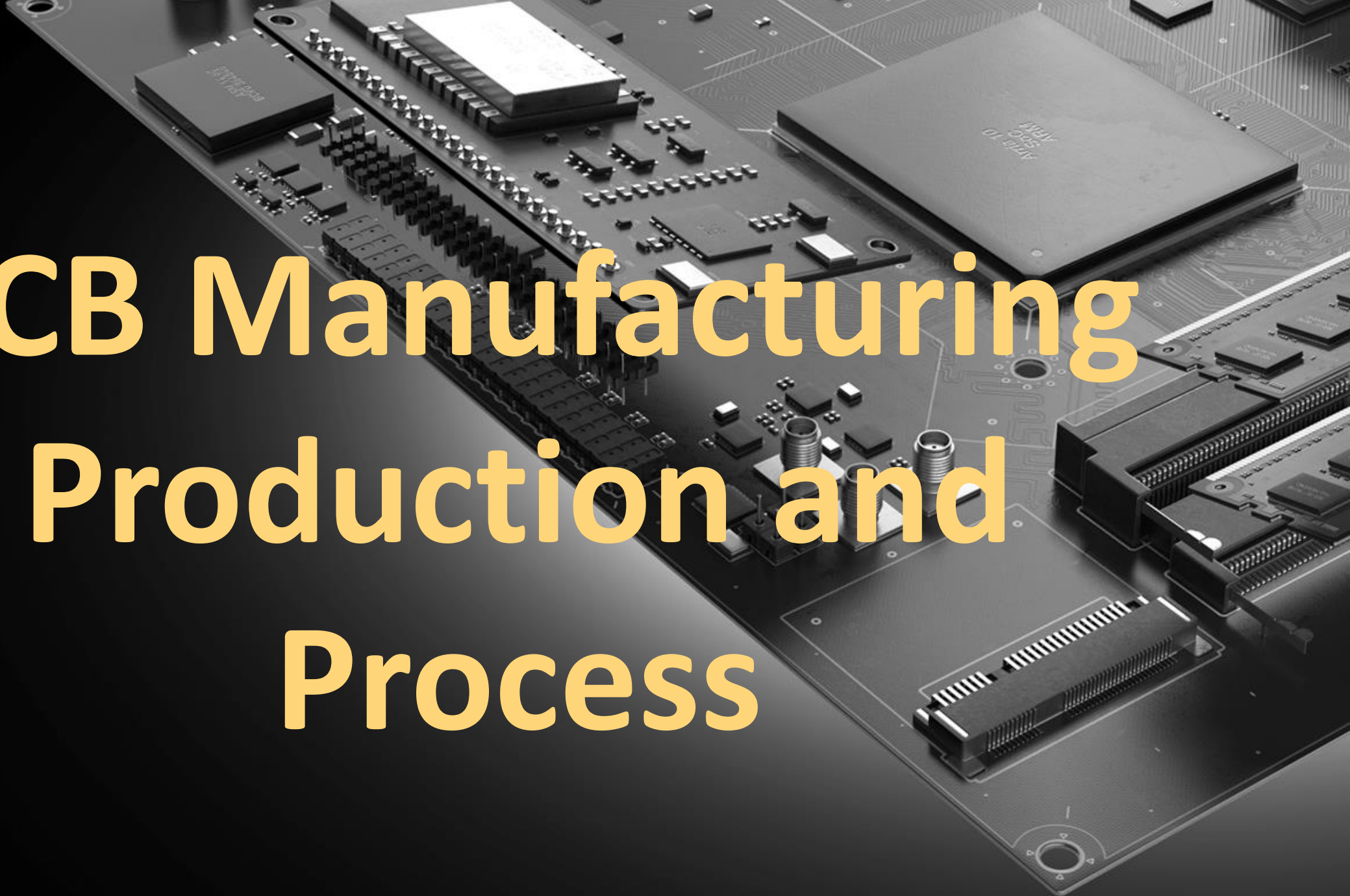


Altium

PCB Manufacturing Production and Process



1. TIPS TO PREVENT OPEN CIRCUITS DURING PCB MANUFACTURING
2. PCB MANUFACTURING PROCESS REVIEWS
3. PCB CAD DESIGN FOR MANUFACTURING GUIDELINES
4. WET AND DRY PCB ETCHING SOLUTIONS - WHICH PROCESS IS THE RIGHT SOLUTION?
5. PCB ASSEMBLY AND PRODUCTION PROCESSES
6. UNDERSTANDING SOLDER BRIDGE SHORTS
7. SOLDER PASTE STENCIL DESIGN GUIDELINES TO REDUCE SHORTAGES
8. WHEN WAVE SOLDERING IS THE BEST PCB SOLDERING PROCESS

1

PCB Design and Fabrication Tips to Prevent Open Circuits During PCB Manufacturing

Prevent Open Circuits During PCB Manufacturing

A number of issues can cause open circuits on the board, varying from materials to processing to handling. Here are the most common causes.

Solder Paste – If solder paste is applied inconsistently, either varying with the amount deposited or with some locations being missed entirely, then there won't be enough to form a solid joint. It will be left with an open circuit, or a joint that's weak and prone to breakage. Another issue is inconsistent reflow temperatures across the surface.

Contamination – Environmental contamination can come from a variety of sources, either on the board or in the solder paste. Obvious causes are chemical spills, dust and particulates in the air, and oils from being touched. Even moisture in the air can lead to accelerated corrosion. Any contamination or corrosion of the pad surface or the component lead can keep the solder joint from bonding correctly.

Gaps and Cracks – Gaps caused by surface irregularities can cause areas of the PCB to lose planarity, making the distance between different leads on the same component vary widely, and keeping leads from even making contact with solder paste during reflow. Sometimes gaps and cracks are severe enough to be visible, and will need a microscope or X-ray to find issues, especially with smaller packaging on components.



[See original content here](#)

2

PCB Manufacturing Process Reviews

PCB Manufacturing Process Reviews

Since the manufacturing facility may contribute to defects, it's a good idea to audit the manufacturer's facilities and practices.

PCB Manufacturing Paperwork and Documentation Reviews – One of the first steps to any review will be looking at the available documentation. Having an understanding of the manufacturer's documentation process will allow to know what kind of information is needed from them, and what there'll be to have ready access to.

What to Look for in Manufacturer Handling Processes

- Process Controls** – Make sure that everyone who enters the area is wearing proper protection.
- Environmental Controls** – Along with process controls, check for good environmental controls. Contamination is a huge concern, and a dirty manufacturing environment will be almost impossible to compensate for.
- ESD Protection** – One of the largest concerns for PCB design is ESD protection. Having environmental protections for ESD during manufacturing will keep components and boards from suffering discharge damage.



PCB Manufacturing Process Reviews

Packaging – Always ask to see the packaging that gets used. Make sure that it meets requirements for the product they are currently packing, and that they are familiar with the type of packaging that the product will need.

As a final step, check how the materials and products are actually unpacked and packed to make sure the handling processes are consistent throughout the production process.

Optimize Your Manufacturing Process by Knowing Capabilities

- Go into audit with a list of the facility's production capabilities.
- Verify that they actually have every tool which will be used.
- Ask to walk through the specific manufacturing path for the proposed products, and find out if they have any recommendations.
- Find out how old the tools are, how they get calibrated, and how often they are inspected.
- Ask about the performance of the tools and how it's tracked. For example, when solder mask is applied, what's the tolerance? How much variability do they see in a single mask, and across an entire manufacturing batch?
- Some manufacturers may also utilize off-site processing. Find out what, if anything, is processed elsewhere, and get comfortable with the details if the place can't be visited.



[See original content here](#)

3

PCB CAD Design for Manufacturing Guidelines

PCB CAD Design for Manufacturing Guidelines

How trace routing can cause problems like cold solder joints and what to avoid in the future.

- **Acute angle traces** – The wedge that is created by the acute trace angle can trap acidic chemicals during the fabrication process. These trapped chemicals don't always get cleaned up as they should during the cleaning phase of fabrication and will further eat away at the trace. This can eventually result in the trace breaking or cause intermittent connections.
- **Tombstoning parts due to trace widths** – Tombstoning happens when a small two pin part, such as a surface mount resistor, stands up on end on one of its pads during soldering. This results from a heating imbalance between the two pads during solder reflow. Whichever side melts first pulls the part towards that side, and causes the tombstoning effect.
- **Cold solder joints** – A cold solder joint is one where the solder has not reflowed correctly to make a good connection, or that the solder has pulled away from the connection. When routing a thick trace out of a pad, the thick trace size may end up pulling the solder off of the pad where it is needed to make the connection to the part.



[See original content here](#)

4

Wet and Dry PCB Etching Solutions – Which Process is the Right Solution?

Wet and Dry PCB Etching Solutions

Experience in PCB etching process and understanding of production and manufacturing processes will allow to have a smoother transition to production.

Know the Process Options- What PCB Etching Solutions are Available? – Etching PCB is a mandatory step for any sort of PCB production.

There are two main schools of application when it comes to PCB etching. The first and most widely understood method of etching is wet etching. This involves the use of chemical solutions that the board essentially bathes in which chemically removes a specified area of the original copper plate leaving behind predefined copper paths. The other, method of etching is known as dry, or plasma etching. This process involves the use of a chamber filled with a positively charged gaseous solution that essentially eats away the unwanted copper material leaving behind a similar looking path to that created by the aforementioned wet method

Applications of Wet Etching – The materials involved can range from simple household items such as the use of hydrogen peroxide, vinegar, and salt. One advantage of using the wet method is that it is relatively simple to perform and is ideal for one-off DIY designs



Wet and Dry PCB Etching Solutions

Applications of Dry Etching – The amount of training that is required for operation is far less than any wet method making it a better solution for a more agile operation. The use of positively charged gas, or plasma, that is used for the physical etching of the copper leaves no residue for any further cleaning stage making it more suitable for higher quantity production runs.

Which Process Will Get You to The Treasure? – If there is in a time crunch with a need for a specific PCB layout, easily get away with a quick and dirty DIY wet etching operation. In need of higher quantities of boards, or if to the point in production where timing, accuracy, and environmental concerns arise in the decisions, then utilizing dry etching methods with the use of dedicated etching machines will be of use.



[See original content here](#)

5

PCB Assembly and Production Processes

PCB Assembly and Production Processes

PCB manufacturer should be doing all the necessary inspection and process checks. Being able to verify these processes are happening is a critical component of manufacturer relationship.

Why Inspect Your PCB Assembly and Production Processes? – Inspecting a manufacturing facility will enable developing a more trusting rapport with manufacturer and give confidence that manufacturing choice is the best for the design considerations.



PCB Assembly and Production Processes

PCB manufacturer should be doing all the necessary inspection and process checks. Being able to verify these processes are happening is a critical component of manufacturer relationship.

What Are Your Methods of Inspection?

Mechanical inspection – Some mechanical tests include shock and thermal creep, but regardless of additional stress on the boards, it should still be physically intact and undamaged before sent out.

Optical inspection – Looking at the boards with a bare eye or under a microscope allows to inspect for larger scale physical damage.

Adhesion testing – If the mix of materials in the layer stack doesn't have the right ratios, have delamination between the copper and the laminate or the solder mask. An adhesion test is often as simple as putting scotch tape over a layer and checking that nothing from the board gets peeled up with it.



PCB Assembly and Production Processes

SEM inspection – Scanning electron microscopes (SEM) are another very expensive piece of equipment. If there is an extraordinarily fine features, the SEM imaging might be used to inspect the board. However, it's more often used to evaluate metallization.

X-Ray inspection – If there are issues with opens or shorts after the layer stack up has been fabricated, a non-destructive testing option is preferable .



[See original content here](#)

PCB Assembly and Production Processes

How Much Can Your Manufacturer Test? – The testing that can be done by manufacturer varies, but at a minimum, they should have some rudimentary electrical testing and thermal imaging to find areas of high-current draw (indicating leakage). Ideally, they'll also have electron microscopy or the capability to do elemental analysis, which can be used to drill down deeper into identifying the root cause of an electrical issue.

If there are specific standards that need to be met, like MIL-SPECs, or consumer safety requirements, find out if there's any pre-compliance testing that the manufacturer can attend to.



[See original content here](#)

6

Understanding Solder Bridge Shorts

Understanding Solder Bridge Shorts

Size and complexity of design are major factors that affect PCB manufacturing costs; however, more than that, overcrowded designs can lead to malfunctions and PCB shorts. Being able to manage the design to effectively work within space constraints will save more money and time

Electrical shorts – The most common issue with high-density designs is frequent electrical shorts. An unfortunate reality is that oftentimes electrical shorts can occur despite how well designed the PCB. When there is a decrease in the size of pads and the distance between them, there is more solder paste in a smaller volume. That paste is more likely to spill out of the appropriate stencil defined areas and create solder bridges to neighboring pads.

Manufacturing – Stencil issues compound the risk of solder bridging, especially in tight arrays of small components. When there is high component density, there is the highest risk of solder paste smearing or misalignment which results in inadequate buffer space. Having buffer space enables the PCB design to prevent solder paste bridges from forming. Pick and place machine accuracy also limits the density that can use in the PCB design. If the accuracy of the machine is very high, it'll have more flexibility to pack in the components



Understanding Solder Bridge Shorts

Inspection, rework, and repair – Inspection is more challenging because edges of components aren't easily visible when they are close together like they are in high-density designs. Finding the issues in a high-density design isn't the entire problem either—when components are so close together, encounter other processes that will slow down or can even cause errors of their own.

Operational requirements on spacing – The requirements for component spacing depend on the function of the board, the operating parameters, and its environment. Consider isolation requirements before trying and reduce spacing, or it'll be shortening the life of the entire product, and possibly introducing safety issues, to save a little bit of upfront cost. There might also encounter handling and safety requirements that will determine edge spacing.



[See original content here](#)

7

Solder Paste Stencil Design Guidelines to Reduce Shortages

Solder Paste Stencil Design Guidelines to Reduce Shortages

The solder paste itself is rarely the root cause of the shorting issues. At each stage of manufacturing, there are opportunities for defects to be introduced- starting with the stencil, during the solder paste application, or during reflow.

Solder Paste Stencil Design Guidelines for Clean PCB Layouts – Often, issues with stencil will cause problems with the application of the solder paste. The stencil marks out where solder paste should be applied to the surface and is reused. No matter what it's made of, the stencil is reused as long as possible.



Solder Paste Stencil Design Guidelines to Reduce Shortages

The solder paste itself is rarely the root cause of the shorting issues. At each stage of manufacturing, there are opportunities for defects to be introduced- starting with the stencil, during the solder paste application, or during reflow.

Even a Solder Paste Stencil Is Subject to Solder Shortage Problems – Here are two primary ways in which the board may encounter problems, despite using a stencil.

- **Uneven solder paste** – Pads that have too much solder paste applied are more prone to forming solder bridges between neighboring pads. If too much paste is being applied consistently consider decreasing the stencil aperture dimensions or the stencil thickness in the design.
- **Stencil resolution** – Another common stencil issue is having poor resolution or jagged edges. If the edges of the stencil aren't clean, it is more likely to have too much solder applied, uneven application, or a lack of definition around the edge of the joint.



Solder Paste Stencil Design Guidelines to Reduce Shortages

Stencils Aren't All to Blame for Your PCB Design Going from Painting to Palette – The finished product is affected by how the materials are handled.

- **Smearing** – A less common issue is a solder paste application that smears or slumps on the surface of the board. Sometimes this isn't a major problem—just weakening the joint—but it is easy for the solder to form a short too.
- **Reflow** – If the temperature profile used for solder reflow isn't correct, the solder paste may not be adequately melted when it needs to bond to components . Too hot and too fast and solder paste may flow away from designated pads.



[See original content here](#)

Solder Paste Stencil Design Guidelines to Reduce Shortages

Don't Be Short-Sighted with Your Selective Soldering – The best way to fix solder paste shorts is to verify stencil quality and reflow recipes with manufacturers early in production. This avoids the reworking of surface areas after excess solder is used and removes the potential for unnecessarily damaging and weakening the surface or neighboring joints.



[See original content here](#)

8

When Wave Soldering is the Best PCB Soldering Process

When Wave Soldering is the Best PCB Soldering Process

Wave soldering is a very fast process that produces reliable, secure connections and can be applied to PCB designs with through-hole components, surface mount devices or both.

Know the Tide- Wave Soldering Advantages and Disadvantages – Prepare the PCB design before the wave soldering process can be successfully applied. An essential requirement is that the design files include a solder mask layer. Care should be taken to ensure that areas that require solder are not on this layer in the design files. Another important consideration is pad spacing. There must be sufficient gap between pads for a single component and between pads on different components.



[See original content here](#)

When Wave Soldering is the Best PCB Soldering Process

Wave soldering is a very fast process that produces reliable, secure connections and can be applied to PCB designs with through-hole components, surface mount devices or both.

The Wave Soldering Process

- Flux Application** – Flux is applied to the board to ensure that the places to be soldered are clean and free of oxidation. Dirty connections can interfere with the current flow.
- Heating** – For the solder to flow, it must be at a high temperature. This heats up the board significantly. To reduce the temperature shock to the PCB, it is preheated prior to exposing it to the solder wave.
- Solder Wave Application** – Once the board is at a reasonable temperature it is run over the wave and the pads are filled with solder. Excess solder may be blown off and the board cools forming nice smooth soldered connections.



[See original content here](#)



ALTIUM
DESIGNER18

Altium Designer 18

New to Altium Designer?
Don't be!

Try the latest release and get time saving workflows and enhanced team
collaboration options today at-

www.altium.com/free-trials



Altium

Thanks for your attention!