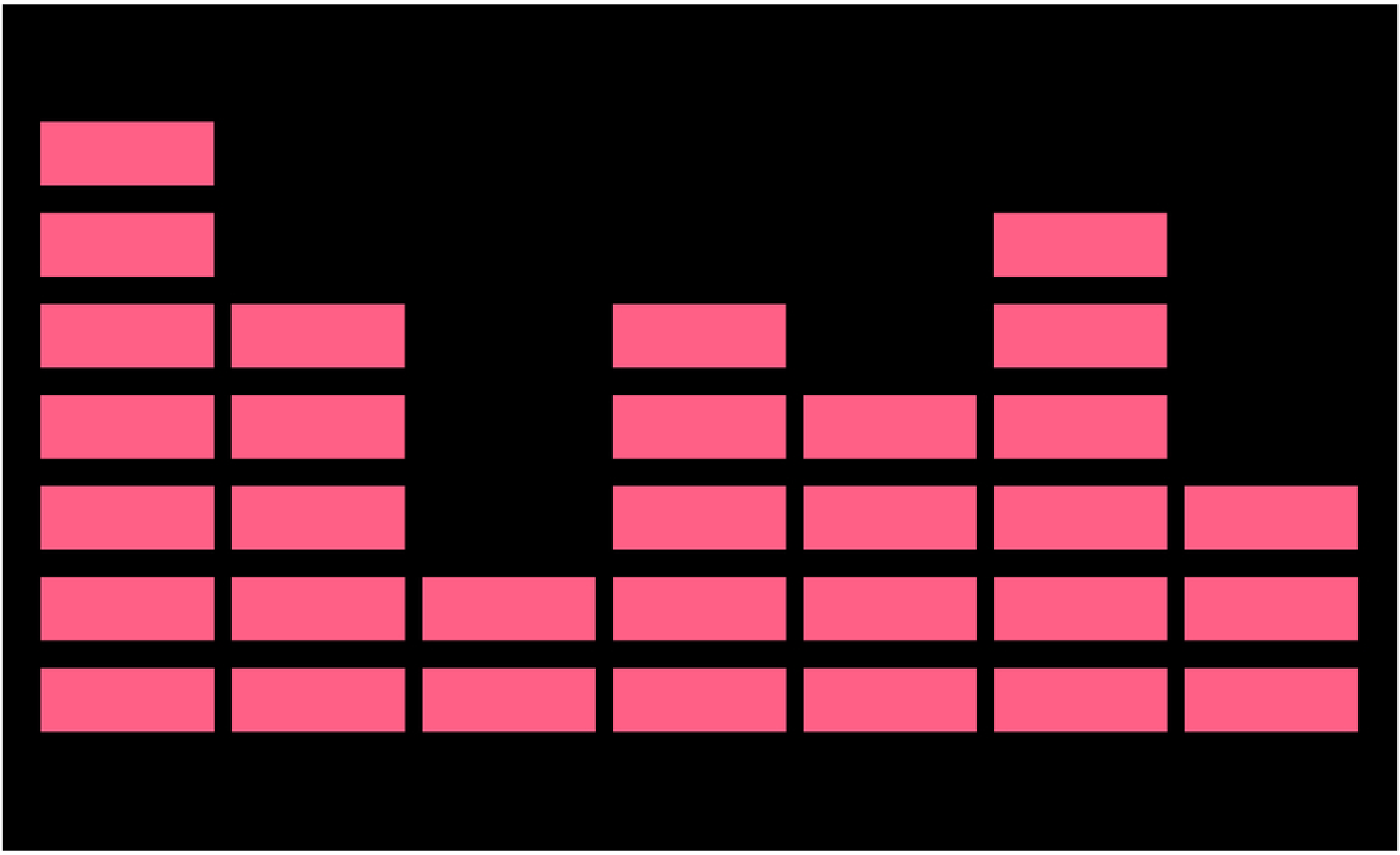
PCB Manufacturing Production and Process



PCB MANUFACTURING PRODUCTION AND PROCESS

Depending on your source, open circuits constitute almost a third of PCB defects during PCB manufacturing and design, especially in the form of open solder joints. Several issues can cause open circuits on your board; here we explore the most common causes. We discuss how trace routing can affect solder joints, what PCB etching solutions are available, understanding solder bridge shorts, and more. Even a solder paste stencil is subject to solder shortage problems, and we discuss when wave soldering is the best PCB soldering process as well as the benefits of manufacturing paperwork and documentation reviews.

Join us as we discuss a variety of topics to help you with PCB Manufacturing Production and Process, including:

- PCB Design and Fabrication Tips to Prevent Open Circuits During PCB Manufacturing
- Taxes Aren't All You'll Pay Without PCB Manufacturing Process Reviews
- PCB CAD Design for Manufacturing Guidelines: How Trace Routing Can Affect Solder Joints
- Wet and Dry PCB Etching Solutions: Which Process is the Right Solution?
- Keep Your PCB Assembly and Production Processes Like Your Best Pants: Seamless
- Understanding Solder Bridge Shorts: Bake Your Cookie and Eat It Too
- Paint by Numbers: Solder Paste Stencil Design Guidelines to Reduce Shortages
- Riding the Wave: When Wave Soldering is the Best PCB Soldering Process

PCB DESIGN AND FABRICATION TIPS TO PREVENT OPEN CIRCUITS DURING PCB MANUFACTURING

For a couple years, I lived in a town with a chocolate and candy factory. It was an amazing and terrible time, because you could go to the factory and buy "seconds," or substandard candies, for about 75% off the normal price. Usually, the mistake was cosmetic, like the chocolate cracking over the caramel, and everything tasted totally fine.

When a PCB manufacturer makes a mistake, sometimes it's cosmetic, and the board will still function. Something like misaligning the final screen printing probably won't affect the electrical performance, but a similar misalignment of a solder mask or copper layer could completely ruin your board. Since PCBs are intended to route electricity, most of the significant performance defects are electrical in nature, things like open circuits, shorts, and routing or material failures.

Depending on your source, open circuits constitute about a third of PCB defects, especially in the form of open solder joints. A number of issues can cause open circuits on your 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. You might be left with an open circuit, or a joint that's weak and prone to breakage. Another issue with solder paste is inconsistent reflow temperatures across the surface. If you've ever microwaved chocolate, you've probably seen hot spots that melt much sooner than the rest. The same kind of variability can happen during solder reflow. If some areas don't reach reflow temperature and bond completely, the electrical connection won't form, similar to leaving unmelted chunks of chocolate in your cocoa or frosting mix.

When solder paste is applied, if the aspect ratio (the aperture width to stencil thickness) is off, and you're more likely to see issues with solder paste deposits. Make sure to verify the layer thickness, especially of your solder mask, with your manufacturer.

Like melting chocolate, solder must reach reflow temperatures everywhere on your board.

CONTAMINATION

No one wants to eat contaminated chocolate. PCB components can get contaminated, too. 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. Check for quality controls from your manufacturer and use in-house handling to make sure parts stay clean and undamaged.

Fingerprints on a board are a common source of contamination, often leading to corrosion and poor solder joints.

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. This is most common if you have component warping or solder mask irregularities, but can result from other thermal mismatch issues, problems in the layer stack up (like air bubbles from improper outgassing), or physically mishandling the board.

Sometimes gaps and cracks are severe enough to be visible, but most often you'll need to use a microscope or X-ray to find issues, especially with smaller packaging on components. Depending on the budget that you have for troubleshooting, you may have to use electrical testing to identify the location of the open circuit and have your manufacturer or a test lab do the final root cause analysis.

Something as simple as dropping your board can break solder connections especially if they were fragile to begin with, like a chocolate egg!

Mistakes during fabrication can be time-consuming and costly. You can improve the process by managing your designs and manufacturer information with quality design layout software, like Altium Designer and Altium Vault.

Do you want to learn more about how Alitum's capabilities can help you improve your design and manufacturing process? Talk to an Altium PCB design expert.

TAXES AREN'T ALL YOU'LL PAY WITHOUT PCB MANUFACTURING PROCESS REVIEWS

Some people are afraid of spiders or snakes and I'm afraid of those too, but one of my worst fears is being audited. Now it's time to start getting organized again and collect on all of my organization last year to make reviewing my notes and preparing for taxes easy. But, wait: my drawer full of notes and important receipts is also full of assorted shopping lists, grocery receipts, home repair goals, and that list I wrote in February for a workout routine that I stuck to for three days. I have a formal bookkeeping process, but, sometimes, I worry that it's not enough. I wish I knew what the IRS was looking out for in their reviews process.

On the other hand, being the auditor isn't just picking daisies. Especially if you're auditing your manufacturing process for your PCB manufacturer, there are hordes of things to keep track of and pay attention to. While small batches of boards or simple PCBs with large tolerances probably don't justify the time and travel for a full review, when you're designing high density PCBs or large volumes of them, a thorough review should definitely be within your considerations.

Since the manufacturing facility may contribute to defects, it's a good idea to audit your manufacturer's facilities and practices. When you visit, you should be prepared with a list of items you want to review, like my checklist of tax forms and documentation. Everything from the equipment, the environmental controls, and the handling procedures should be considered. It may seem like a lot, but if you're going to the effort to visit a manufacturing facility, especially if it's overseas, it will be better to get it all done at once.

PCB MANUFACTURING PAPERWORK AND DOCUMENTATION REVIEWS

One of the first steps to any review will be looking at the available documentation. This can range from questions such as if temperatures and ramp times are recorded each the time the solder reflow oven is used if issues with etching are recorded, and how different products are tracked and kept separate during manufacturing. Having an understanding of your manufacturer's documentation process will allow you to know what kind of information you'll want from them, and what you'll be able to have ready access to.

Just as important to understanding their documentation process, your manufacturer's design security is incredibly valuable too. Horror stories like manufacturers using your design to start producing counterfeit versions themselves, or having unsecured computers where your proprietary designs could be easily accessed by unauthorized entities are as much a part of your review of your manufacturer's process. Even if it seems redundant or insulting, it's significantly better to ask the specific questions now instead of seeing a competitor learning your trade secrets.

You should verify security procedures during your audit.

WHAT TO LOOK FOR IN MANUFACTURER HANDLING PROCESSES

When you get into checking the manufacturing facilities, you'll be looking into how the manufacturer handles the materials and the environment. Here are some of the key methods to check in your handling reviews:

Process Controls: Make sure that everyone who enters the area is wearing proper protection. That should include hair nets to keep static discharge from building up, as well as keeping hair off the boards in case of shorts. People should also be wearing wrist, and possibly shoe straps that are plugged into the ground at each workstation.

Environmental Controls: Along with process controls, you want to see good environmental controls. Contamination is a huge concern, and a dirty manufacturing environment will be almost impossible to compensate for. There shouldn't be dust collecting on any equipment. Any processes that produce dust, like CNC milling, should be separated from the rest of the production line and cleaned regularly. Chemical processes should be performed under hoods or appropriate ventilation. Operator contamination sources, like oil from your skin, hair, or clothing, should be minimized by gloves, goggles, hairnets, or other PPE.

ESD Protection: One of the largest concerns for your PCB design is ESD protection. Having environmental protections for ESD during manufacturing will keep components and boards from suffering discharge damage. As much as I've harped on paperwork, ESD areas shouldn't have any loose paper or packaging materials around that could get pushed around and generate a static charge.

Packaging: Always ask to see the packaging that gets used. Make sure that it meets your requirements for the product their currently packing, and that they are familiar with the type of packaging that your 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. Consistency and manufacturing safety is vital to the overall health of your manufacturing process.

Check if everyone in ESD safe areas is wearing the proper protections.

OPTIMIZE YOUR MANUFACTURING PROCESS BY KNOWING CAPABILITIES

Your first and main concern with a manufacturer's audit would most likely be the actual production capabilities of your manufacturer; skipping over their review would be like itemizing your deductions when you have no income to report: an awful lot of work to miss the main event. Here is a list of things to consider:

1. Go into your audit with a list of the facility's production capabilities.
2. Verify that they actually have every tool you think you'd be using. Ask what the purpose is when you aren't sure, because they may have capabilities you didn't know you could utilize.
3. Ask to walk through the specific manufacturing path for your proposed products, and find out if they have any recommendations.
4. Find out how old the tools are, how they get calibrated, and how often they are inspected.
5. 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?
6. Some manufacturers may also utilize off-site processing. Find out what, if anything, is processed elsewhere, and make sure you are comfortable with the details if you can't visit it yourself.

The most enlightening parts of a manufacturer audit might be in finding out that a manufacturer can produce one of your designs in a cheaper and more effective way than you had previously anticipated. Being able to know confidently what your manufacturer is capable of will give you a clearer gauge of the success of manufacturing any of the potential designs you come up with.

Unlike an IRS audit, manufacturing reviews should be informative and helpful for everyone involved. It's an opportunity to learn more about the production line, and how to improve your designs.

After you learn what your manufacturer's capabilities are, you can use those guidelines to help improve your PCB design process with assistance from your CAD system. For easy application of manufacturer capabilities to PCB design software, consider using smart layout tools such as those found within Altium's CircuitStudio.

If you'd like to learn more about how to apply knowledge about your manufacturer to your PCB designs, consider talking to an expert at Altium today.

PCB CAD DESIGN FOR MANUFACTURING GUIDELINES: HOW TRACE ROUTING CAN AFFECT SOLDER JOINTS

Editorial credit: Aija Lehtonen / Shutterstock.com

A couple of weeks ago I attended a concert that was a tribute to the big band leader Stan Kenton. I love big band jazz for many reasons, one of which is the configuration of musicians and instruments in the band. There are usually around 15 to 20 musicians on different instruments, and everyone plays a different part. If just one person makes a mistake, it can ruin the balance of the number that was so carefully arranged by the composer.

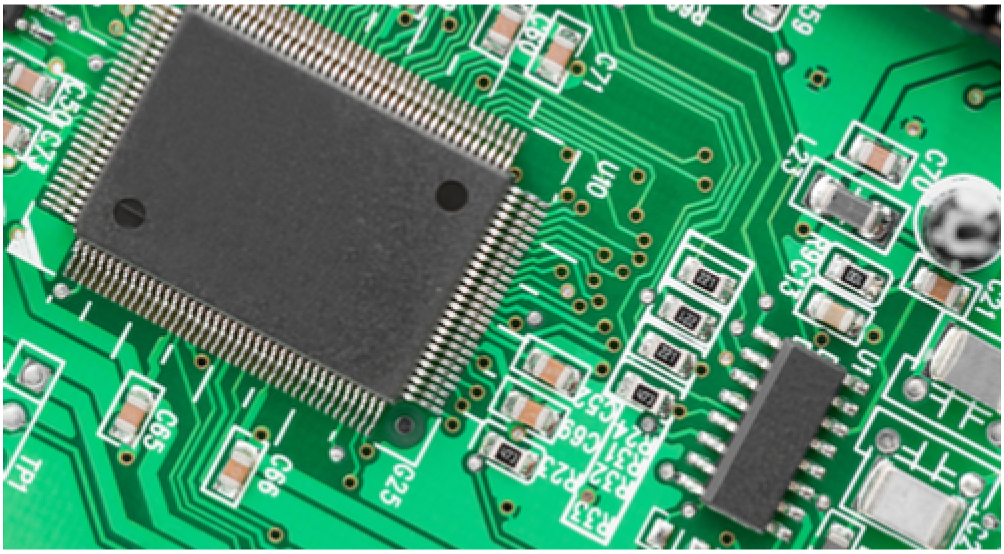
The importance of each member of the band playing harmoniously together reminded me of the importance of a correctly manufactured printed circuit board. If just one part isn't soldered correctly, the finished circuit board may have intermittent failures, or perhaps not work at all. Just as a saxophone playing a sour note can ruin the whole number, a bad solder joint can ruin the entire board. Fortunately, design for manufacturing (DFM) rules can help you to avoid hitting sour solder joints on your circuit board.

One area where DFM rules can help your board may come as a surprise. How you route traces on your PCB can have a direct effect on solder problems, and DFM rules offer some guidance there. Take a look with me now at how trace routing can cause problems like cold solder joints or tombstoning so that you will know what to avoid in the future.

ACUTE ANGLE TRACES

The first problem that we'll look at is acute angle traces. Although this situation doesn't specifically lead to a solder problem, it is a routing problem noted in PCB DFM guidelines.

Acute angles in traces are traces which have corners that are greater than 90 degrees. This causes the trace to come back on itself. 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.



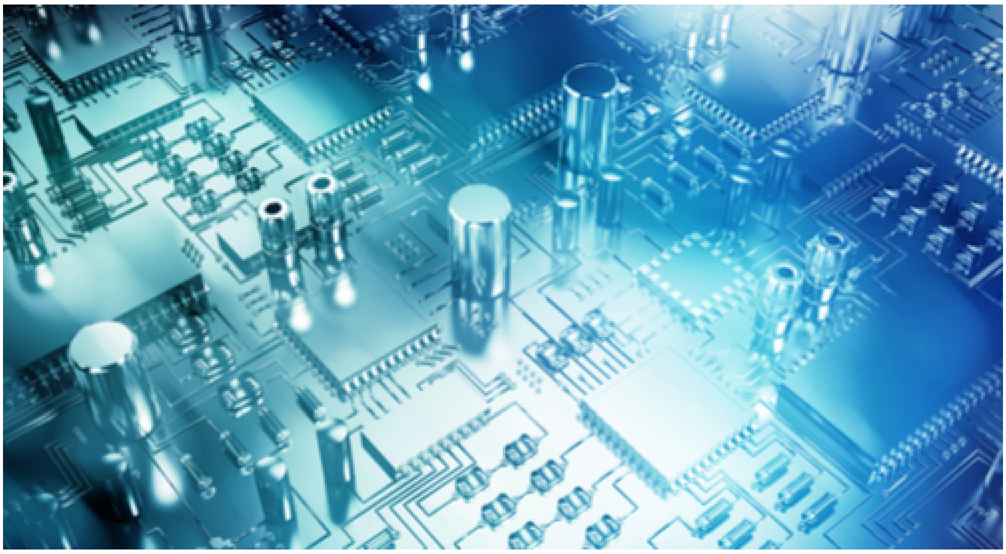
Trace routing on a PCB

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.

One of the factors that can cause this heating imbalance is using different sized traces on the two pads. The wider the trace, the longer it will take for the pad that it is connected to heat up. If one pad of the part has a very narrow trace, and the other pad has a very wide trace, you will likely have a solder reflow imbalance and one pad will melt and reflow before the other one.

Often electrical engineering will want a power trace that is too wide for the manufacturer to reliably solder. PCB design for manufacturing guidelines have recommendations for the minimum and maximum trace widths to use on different sized parts, but that might not solve your problem. The key for you is to balance the requirements of both electrical engineering and manufacturing and come to a common agreement between the two. In this way, you can meet the needs of both sides on your design.



DFM rules can help you to design out manufacturing problems on your board

COLD SOLDER JOINTS

Another problem that can happen when routing thicker traces is the creation of a cold solder joint. 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.

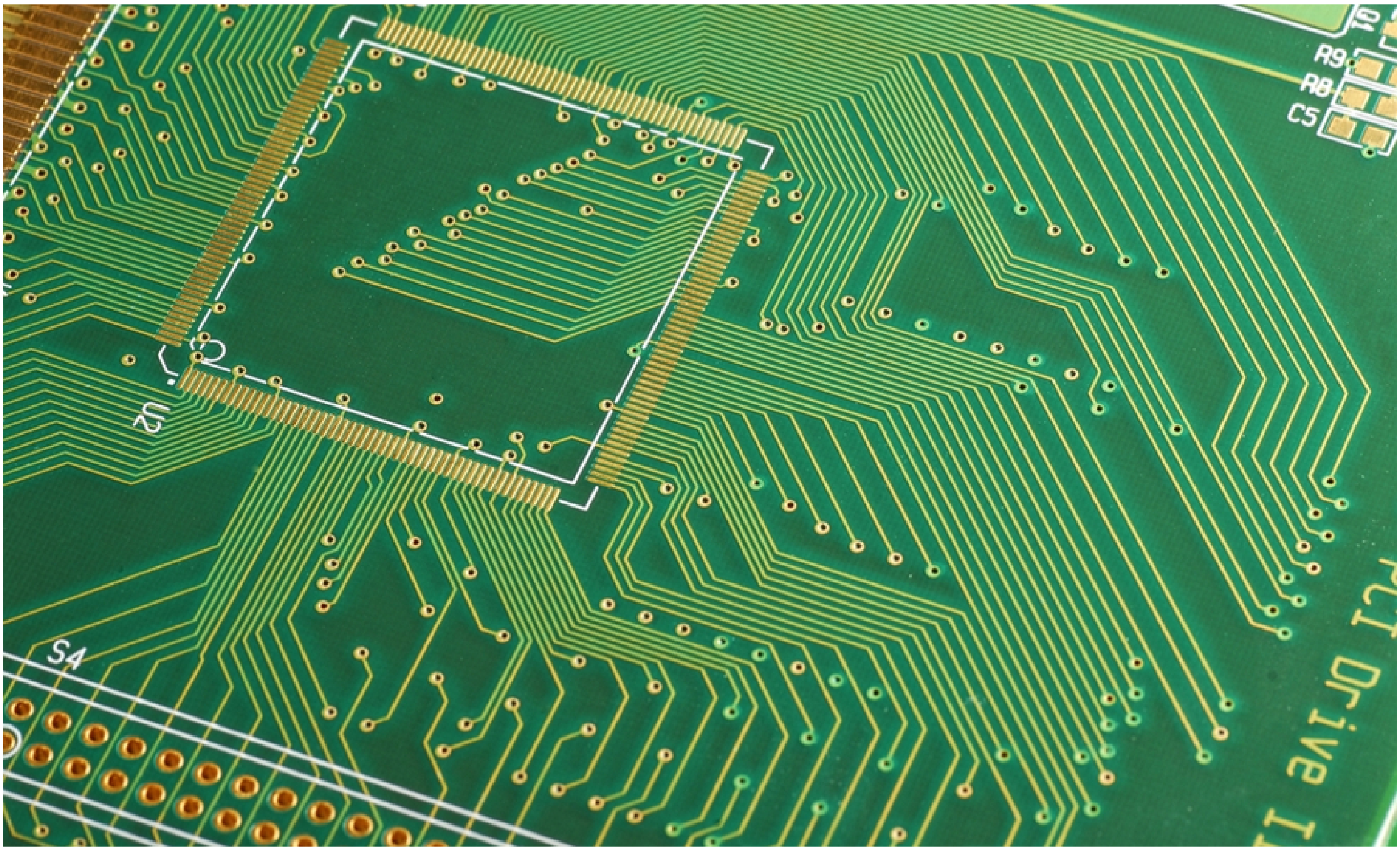
The solution is to use traces widths that are smaller than the pad size. Some DFM guidelines recommend a trace no wider than 0.010 mils, although this again must be worked out to balance the needs of both electrical and mechanical engineering.

There is a lot more to PCB design for manufacturing guidelines than the trace routing recommendations that we have given you here. DFM guidelines will also help you with proper component placement techniques, footprint sizes, and other aspects of your design. This will ultimately help your design to be manufactured with as few errors as possible. A circuit board that is error free during manufacturing is a reflection of a good and solid design, kind of like hearing the Stan Kenton band playing an error-free rendition of Intermission.

PCB design software, like Altium Designer, has advanced routing capabilities and other features to better help you to design according to your DFM rules. This will help you to deliver a design that is DFM compliant to your manufacturer the first time.

Would you like to find out more about how Altium can help you with your next design to assure its DFM compliance? Talk to an expert at Altium.

WET AND DRY PCB ETCHING SOLUTIONS: WHICH PROCESS IS THE RIGHT SOLUTION?



I have always been a very visually-oriented thinker. When I was much younger, I used to look up maps to video game levels online and put a sheet of tracing paper up against the screen to copy them down. The Zelda games specifically would always leave me with a notebook full of maps and, on the more challenging dungeons, color-coded warnings. If I sneezed once in my tracing process or the screen moved or I was called away mid-tracing, my map would be restarted.

One time, I spent hours in a dungeon only to realize that I'd done it all wrong because of a mistake in my process and had to backtrack to understand exactly where I messed up. Now, we don't have to worry about etching mistakes so much with more intuitive dungeon-layout logic in games and, more generally, the availability of phone and computer screens.

It does make sense, though, that I've worked in positions that make me recognize the PCB etching process. When I was just getting started in design, I had very little idea about etching and how it worked. But as my experience grew, I was put into positions which required me to have more of a direct understanding of production and manufacturing processes. Having this knowledge will allow you to have a smoother transition to production.

KNOW THE PROCESS OPTIONS: WHAT PCB ETCHING SOLUTIONS ARE AVAILABLE?

Etching your PCB is a mandatory step for any sort of PCB production. Whether it is a simple prototype or a full-blown manufacturing batch, your board's connection paths will need to be precisely defined and laid out so that your components will be able to successfully communicate with one another.

There are various styles of PCB etching that can occur in order to achieve similar results. The style that you choose, however, could depend on many factors involved. Is this simply a one and done PCB? Is time a factor in your decision? What about environmental impacts to larger production runs? How tightly packed and complex is your design?

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 your predefined copper paths (once cleaned).

The other, slightly newer 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. Of course, knowing which is right depends on your application.

Knowing which etching choice is best for your PCB can be a difficult decision.

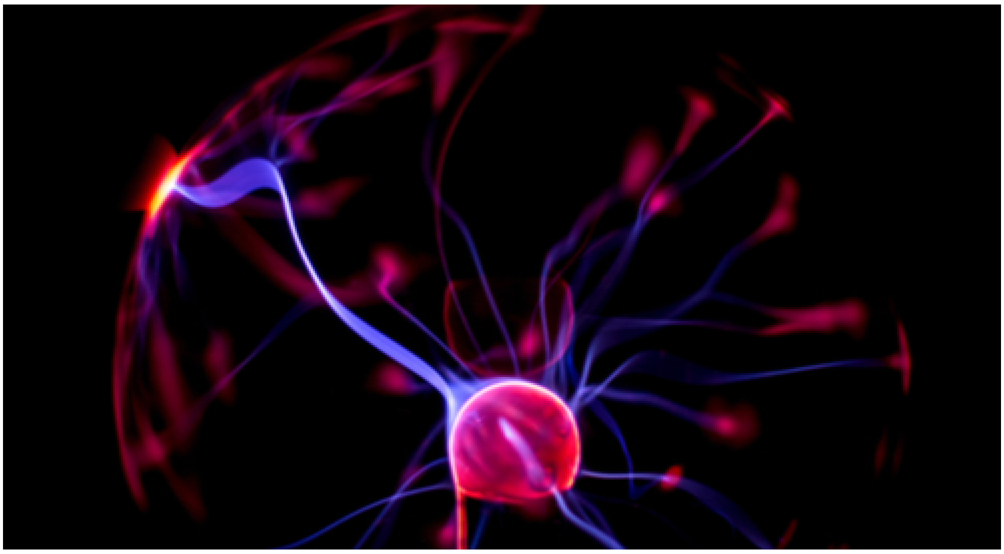
APPLICATIONS OF WET ETCHING

If you perform a quick search of the term 'PCB wet etching', you'll find a plethora of DIY videos and instructions on how to etch your own boards with simple, easy to attain items. The materials involved can range from simple household items such as the use of hydrogen peroxide, vinegar, and salt. Other, more robust applications involve the use of more 'potent' chemicals such as copper chloride.

One advantage of using the wet method is that it is relatively simple to perform and is ideal for one-off DIY designs. If you are in a crunch for time and can't wait for the fab house to get you a prototype board out, you will certainly be able to procure your own etched board within a day or so. This method, of course, is rather messy and does require some knowledge of the chemicals and process involved as you will literally be soaking your board for a short amount of time.

Using the household materials as listed above is obviously a cleaner set up, however, using the industry standard copper chloride will involve some forethought on proper handling and disposal of the solution as it is terribly harmful to the environment (especially with all that dissolved copper floating around in it).

If you are in need of a higher quantity production, further considerations should be made accordingly. The amount of material alone in a production level wet etch will surpass that of the dry method, along with floor space required for bathing, proper training and education of your workers, as well as governmental permits required for disposal.



Plasma (dry) etching acts like a plasma lamp when etching away the unwanted copper on your PCB

APPLICATIONS OF DRY ETCHING

On the other side of the coin, the dry etching method is a far more complex operation requiring a dedicated machine for its operation; however, the process is a slightly cleaner way of etching in my opinion. 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. The disposal of the residue will be released straight to the atmosphere which can be viewed in a negative context, but when comparing to the wet method, it appears that less waste will impact the environment as a whole.

Drawbacks to this method of etching are limited if you intend to reproduce boards in multiple quantities, but if you are simply looking to develop a handful of one-off prototypes, the investment in these machines may outweigh the return.

WHICH PROCESS WILL GET YOU TO THE TREASURE?

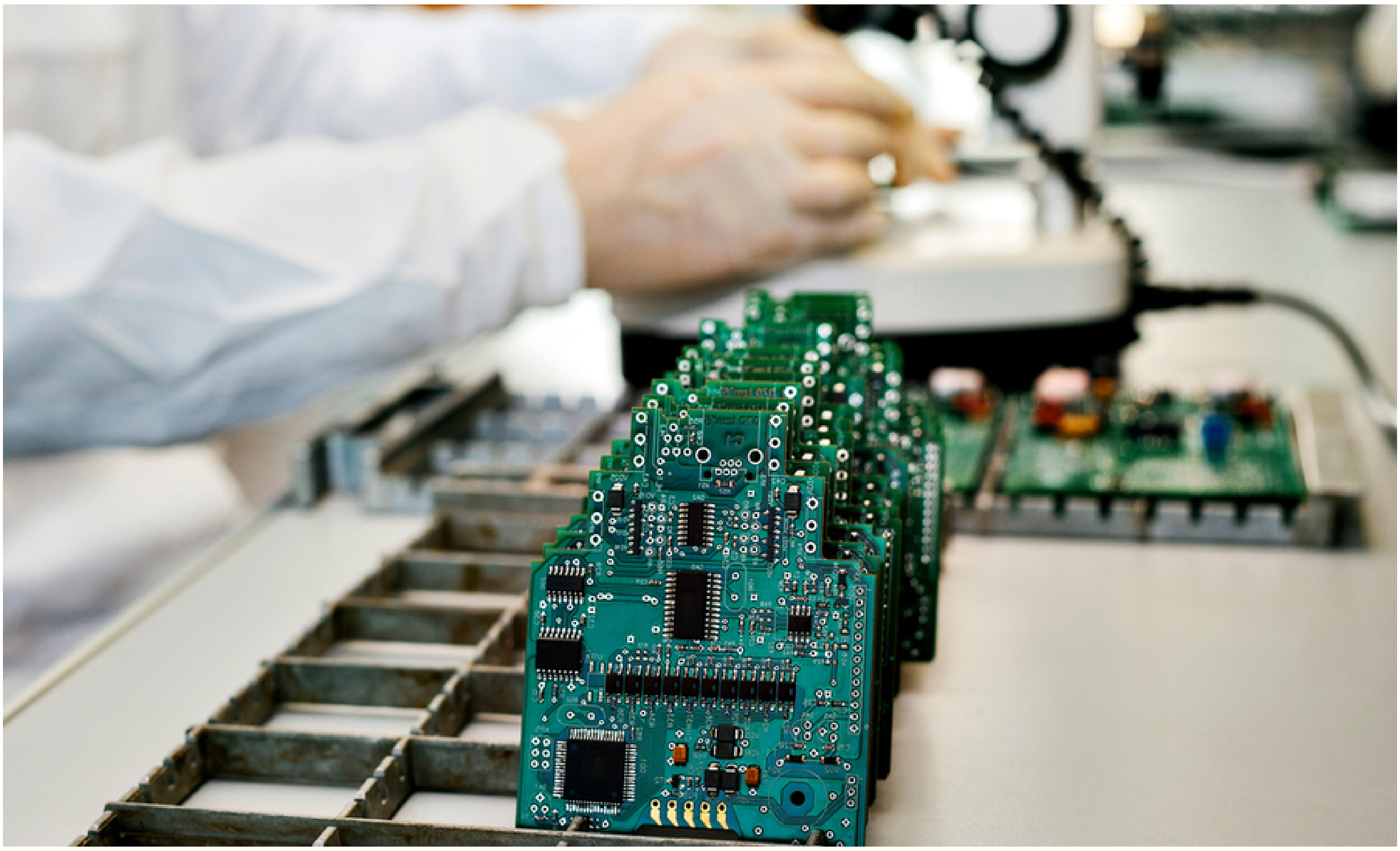
If you are in a time crunch with a need for a specific PCB layout, you can easily get away with a quick and dirty DIY wet etching operation. If you are in need of higher quantities of boards, or if you are to the point in your production where timing, accuracy, and environmental concerns arise in your decisions, then utilizing dry etching methods with the use of dedicated etching machines will likely be for you.

Take careful review of your manufacturers and ask for an overview of their process in all aspects like preparation, operation, and disposal; ensure their standards align with your needs and concerns before jumping in. To get you to a point where you can consider your manufacturing and production options for your designs though, you should be using PCB design software which can contend with your design demands, and prepare you for next steps.

A strong design software will allow you to move seamlessly between schematic, footprints, layout, and manufacturing outputs without painful transferring and translating processes. Through a unified design environment, you won’t need to rely on a horde of different softwares or tools to find the treasure at the end of the dungeon. With the idea of a smart, intuitive design approach in mind, Altium Designer would be the right fit for any designer.

If you want to further discuss etching concerns or progressing your designs through all phases of the process, talk to an expert at Altium today.

KEEP YOUR PCB ASSEMBLY AND PRODUCTION PROCESSES LIKE YOUR BEST PANTS: SEAMLESS



I lost a button on an old pair of jeans the other day and I was totally unable to repair it. The seams were all sorts of wonky, and while normally you can find extra buttons along the waist there were none to be found. My mom taught me to always check the seams, but sometimes it's easy to get lazy and forget to do some of the most basic procedures when you want to get an otherwise grueling experience done with. Some brands are more trustworthy and reliably durable than others, but even then it's still a good practice to check the clothes you're about to buy for any obvious manufacturing defects.

In an ideal world, all the most rigorous checks are outsourced back to the manufacturer. After all, they're directly involved in the production process and should have the equipment to review themselves with ease. In a realistic world, your 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 your manufacturer relationship. With the right knowledge, you'll know what to look for and ask about when verifying manufacturing procedures as well as ensuring some initial product testing.

WHY INSPECT YOUR PCB ASSEMBLY AND PRODUCTION PROCESSES?

Inspecting a manufacturing facility will enable you to develop a more trusting rapport with your manufacturer as well as give you confidence that your manufacturing choice is the best for your design considerations. When you visit your manufacturing facility, you should check all of the tools and facilities that will be used to produce your board. You also want to look at handling between tools, and materials being used. However, your work is far from done once you've verified all the tools for a build.

After assembly, and probably at several stages of fabrication, your manufacturer should be inspecting the boards and conducting process checks. Ideally, they will even be able to do some function tests and find any issues before the final boards are shipped back to you. After all, if your pants were being sewn with a hole instead of pockets you would want that to be found early so the pockets could be replaced easily.

Unlike PCBs, jeans can pass inspection with holes in them unless you have my mother inspecting your outfit.

WHAT ARE YOUR METHODS OF INSPECTION?

Using the fabrication flow path for your PCB and the processes that are activated during manufacturing, you can determine inspection tools and procedures. You should find out about the following inspection processes used by your manufacturer, and watch them in action if you can:

Mechanical inspection: Some mechanical tests include shock and thermal creep, but regardless of what (if any) additional stress you apply to the boards, it should still be physically intact and undamaged before you send it out.

Optical inspection: Looking at the boards with a bare eye or under a microscope allows you to inspect for larger scale physical damage. Solder bridges, burns, and misalignments from reflow are usually easily visible if you have the patience to search.

Adhesion testing: If the mix of materials in the layer stack doesn't have the right ratios, you can have delamination between the copper and the laminate or the solder mask. This should be checked during manufacturing; if you discover it later you may have to scrap many parts and many more hours of fabrication work. 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.

SEM inspection: Scanning electron microscopes (SEM) are another very expensive piece of equipment. If you have extraordinarily fine features, the SEM imaging might be used to inspect the board. However, it's more often used to evaluate metallization. The electron beam will damage certain types of materials, even stripping them away. Gold-coated pads are tested this way, with a fast stripping time indicating a very thin plating layer and revealing a poor surface quality.

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 until you know for sure what and where the problem is. Not every manufacturing plant has x-ray inspection capabilities, so it's something you should ask about when you visit.

X-ray inspection isn't usually this glamorous, but it can be useful for internal issues in your PCB.

HOW MUCH CAN YOUR MANUFACTURER TEST?

The testing that can be done by your 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 you have specific standards you need to meet, like MIL-SPECs, or consumer safety requirements, you should also find out if there's any pre-compliance testing that the manufacturer can attend to. If they can produce and test an initial batch it will be much cheaper than finding out in the middle of certification that you've got a mysterious resonance or can't meet voltage safety requirements.

While it's admirably gutsy, and maybe saves you a little time in the present, buying jeans without trying them on is just asking for trouble in the future. Having PCBs produced without being familiar with your manufacturer isn't even admirably gutsy, it is going up to trouble's doorstep and begging for it to come visit you.

By choosing the right PCB design software, you can select the tools that will save you in the long run and ensure quality, repeatable processes. With smart PCB layout, and real-time component placement corrections, Altium's CircuitStudio will be able to give you the CAD system you need to get your job done right.

If you've got any advice for finding good jeans, or want to chat about your CAD software needs, talk to an expert at Altium today.

UNDERSTANDING SOLDER BRIDGE SHORTS: BAKE YOUR COOKIE AND EAT IT TOO

I love the idea of baking. Especially around the holidays, there's something thrilling and therapeutic in planning to bake up elaborate platters of cookies and bring them to parties. In reality, what I like is to eat cookie dough. The actual rolling or spooning out cookies onto a pan and waiting for them to bake is an exercise in both tedium and restraint. When I lose my patience, I try to cram as many cookies onto the tray as possible; of course, this results in a single, giant cookie that, begrudgingly, I have to eat all myself.

Whether you're wallowing in the winter planning of too-many holiday cookies, or just wishing for a more convenient way to plan out your space-constraints, the goal is always going to be designing the layout of your baking sheet to avoid scrapping that whole tray of cookies. If you spent the time making them, you should earn the reward of eating them. That lesson ought to be true for a lot of things though, shouldn't it?

Unfortunately, if I overcrowd my PCB it doesn't just turn into a cookie. No, being impatient and not working within space constraints can be much more costly than ruining a batch of cookies. 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 your design to effectively work within space constraints will save (and probably earn) you more money and time that could be better spent on other things (like, for example, eating more holiday cookies).

ELECTRICAL SHORTS

The most common issue with high-density designs is frequent electrical shorts. An unfortunate reality (an inconvenient truth?) is that oftentimes electrical shorts can occur despite how well you designed your PCB. Solder bridges are likely so long as you have a high-density design; however, high-density designs are often unavoidable with the increasing demands from PCBs.

When you decrease 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. Like accidentally baking a giant cookie, the only way to avoid those bridges is by having buffer space between the pads that is enough to protect them from a little bit of spillover from their neighbors. It isn't designing for failure, but rather allocating design resources into understanding shortage risk and managing it appropriately.

Overheating your PCB and components is just as bad a burning cookies, and more expensive to redo.

MANUFACTURING

Other manufacturing stages can also be more difficult with higher density PCB designs. Stencil issues compound the risk of solder bridging, especially in tight arrays of small components. When there is high component density, you have the highest risk of solder paste smearing or misalignment which results in inadequate buffer space. Having buffer space enables your PCB design to prevent solder paste bridges from forming. Without that buffer space, solder bridges become more likely and encourage the occurrence of errors like shorting.

Pick and place machine accuracy also limits the density you can use in your PCB design. If the accuracy of the machine is very high, you'll have more flexibility to pack in your components. However, also consider the tooling used on the pick and place. Placement tools can extend beyond the edge of the component, which can add difficulty to tightly-packed or small component placing.

Depending on the place I'm living, sometimes I need a stepstool to reach whatever I've stored in the top cupboards. This might impede my cookie-baking speed, but it won't ultimately keep me from baking. Component height also becomes an issue if the pick and place can't reach the location next to a taller component like in the case of resistors next to headers. Imagine if you were decorating a cookie and put giant marshmallows down, then needed to draw in delicate icing without moving the cookie. I usually eat the cookie before the decorating process, but I can't just eat a PCB.

Variable height components make cookies as tricky as a PCB

INSPECTION, REWORK, AND REPAIR

After manufacturing, the challenges of a high-density design aren't over. Inspection is more challenging because edges of components aren't easily visible when they are close together like they are in high-density designs. You may be blinded from viewing solder joints and verifying standoff heights. If you have varying heights, then the components might actively block each other from view.

Finding the issues in a high-density design isn't the entire problem either—when you have components so close together, you can encounter other processes that will slow down or can even cause errors of their own. You might have to remove other components to get at whichever component you need to fix, and even with the tiniest tweezer and most stable hands, reheating sections of the board will assuredly remelt the solder and cause slippage. You may also need to add a heat shield for nearby components since there's less total room for heat to dissipate.

OPERATIONAL REQUIREMENTS ON SPACING

Before any scenarios in which errors, bridges, or shorts occur exist, the foremost demand on component spacing is from operational requirements. There are formalized specifications for spacing: the most commonly cited is IPC-D-279, section 3.3.9, but there are also manufacturer recommendations, like this chart from OCM, often for a specific packaging, or application. Once you have varied component and packaging types, the specifications often aren't extensive enough to provide the necessary guidelines.

The requirements for component spacing depend on the function of the board, the operating parameters, and its environment. Consider isolation requirements (based on voltage, RFI, or EMC) before you try and reduce spacing, or you'll be shortening the life of the entire product, and possibly introducing safety issues, to save a little bit of upfront cost. You might also encounter handling and safety requirements that will determine edge spacing. Initially, that buffer seems like an easy place to cut excess, but cutting your buffer could have catastrophic impacts on the lifespan on the board.

Spilled icing is delicious. Spilled solder paste is disastrous.

On top of all the other information that you need to keep track of while designing, component spacing and being aware of solder bridge shorts may seem like a troubling task. Trust me, coming from someone who often forgets to preheat the oven before putting anything into it, I understand the difficulty of keeping track of multiple instructions. But when you are doing high-density design, use Altium Designer with its robust design rules checking, can make sure "keep out," areas and other physical restrictions for components are all accounted for, and that can import manufacturer rules to check for compatibility before designing.

All this and other great design tools can be at your fingertips to make your design life more manageable. Learn more about solder bridge shorts and component spacing by talking to an Altium Expert today.

PAINT BY NUMBERS: SOLDER PASTE STENCIL DESIGN GUIDELINES TO REDUCE SHORTAGES

I’ve never had much artistic talent, but when I was a kid I loved those paint-by-numbers kits. Up until I saw those I’d only used watercolors, but I was enthralled with the thick, slow flow of the paint. I may also have killed some brain cells from breathing all the fumes. I was completely obsessed with keeping the paint in the lines, so I didn’t use the brush they gave you. Instead, I traced the outlines with a toothpick that had barely been dipped into the paint, then dipped the toothpick again and dripped paint into the outlines, relying on surface tension to distribute the paint evenly.

Although that level of neuroticism can improve your PCB design, it's probably not healthy to apply in all areas of your life. Try and limit it to one or two places, like getting a totally perfect PCB stencil for your solder paste application.

Solder paste is a common cause of shorting in PCBs since the paste covers a large amount of your board surface and bridges can form easily. However, 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 your stencil will cause problems with the application of the solder paste. The stencil is a mask usually made from metal but sometimes out of Kapton (polyimide) for prototypes or low quantity runs. The stencil marks out where solder paste should be applied to the surface and is reused. It designates where solder paste should be applied to the surface like a spray paint stencil. No matter what it's made of, the stencil is reused as long as possible. That might be less than five times for polyimide or thousands of times for a metal stencil.

A stencil for solder paste is like a stencil for spray paint, marking out where you want your material applied.

EVEN A SOLDER PASTE STENCIL IS SUBJECT TO SOLDER SHORTAGE PROBLEMS

While it is tempting to think that with a solder paste stencil, you do not need to worry about potential problems or interferences, that is not always the case. Here are two primary ways in which your board may encounter problems, despite using a stencil:

- **Uneven solder paste:** Pads that have too much solder paste applied are, unsurprisingly, more prone to forming solder bridges between neighboring pads. If too much paste is being applied consistently, either across a board or between different boards, you should consider decreasing the stencil aperture dimensions or the stencil thickness in your design.

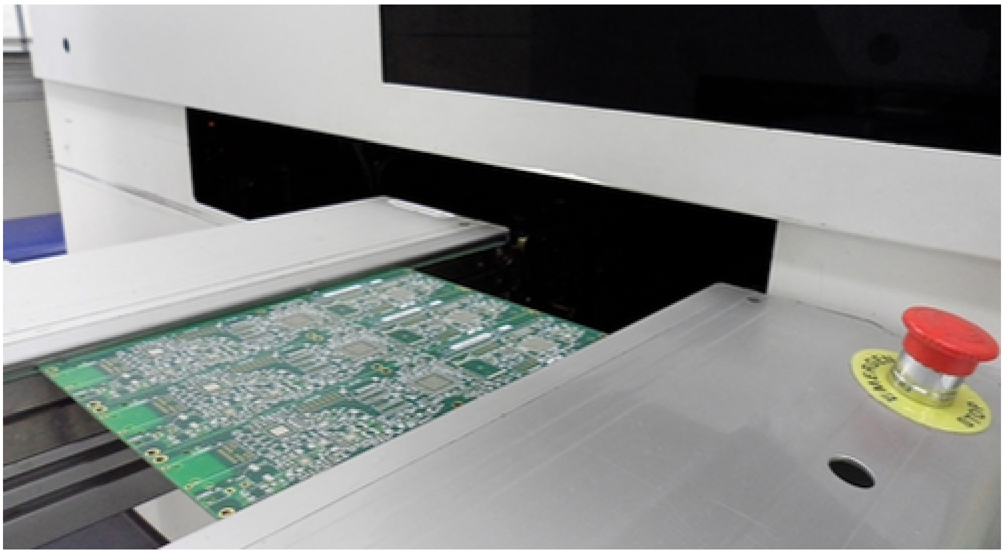
- **Stencil resolution:** Another common stencil issue is having poor resolution or jagged edges. If the edges of the stencil aren't clean, you're more likely to have too much solder applied, uneven application, or a lack of definition around the edge of the joint. All of these situations are an open invitation to solder bridges shorting your pads together.

STENCILS AREN'T ALL TO BLAME FOR YOUR PCB DESIGN GOING FROM PAINTING TO PALETTE

Like working with paint, you can end up with solder paste in locations where you don't want it. The finished product is also affected by how the materials are handled after their application, though. Like paint cracking when it dries too fast, solder reflow temperatures have to be just right for a proper outcome.

- Smearing: A less common issue is a solder paste application that smears or slumps on the surface of the board. If you have a smear, you'll lose clean edges no matter how sharp your stencil. Part of smearing is getting solder applied in unwanted areas. 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 you use for solder reflow isn't correct, the solder paste may not be adequately melted when you need it to bond to components. Initial temperature ramp rates are particularly troublesome. Too hot and too fast and your solder paste may flow away from designated pads.



Make sure to optimize your equipment to the exact specifications to avoid damages of excess.

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. It is possible to heat and remove extra solder, but it isn't recommended as that, too, may damage nearby surfaces and joints.

When you design your PCB, the stencil is one of the layers in your Gerber file. You should choose a PCB design software that allows you the best control over your design. Look into better control of your designs with CircuitStudio, and by talking to an expert at Altium.

RIDING THE WAVE: WHEN WAVE SOLDERING IS THE BEST PCB SOLDERING PROCESS

Everybody loves the beach and I am no different. Sitting on the edge of the ocean being mesmerized by the ebb and flow of the water is a favorite pastime of mine. This is especially true when there are surfers riding the waves. It is awesome to watch a huge wave appear intent on engulfing them only to see them emerge upright on their boards having ridden the wave towards the shore. However, when too many surfers are bunched together they are inevitably toppled over due to a lack of water to support them all. This is similar to what happens when wave soldering is used to secure the components on a PCB.

The objective of wave soldering is to process the PCBs quickly. This is done by applying a wave of solder that encompasses the width of the board and the components ride the liquid solder upright in place until it is cooled and they emerge with connections firmly secured. If the board design is especially suited for the wave soldering process, the parts can be soldered in a matter of seconds. If the part clearances are not sufficient, the solder wave may not reach everywhere or connections may wind up fused together. In the past, when circuit boards and the components mounted on them were large, the use of wave soldering was the primary technique for quick PCB assembly.

Today, with the ever-increasing push toward smaller and smaller boards and the prevalence of surface-mount technology (SMT) more precise selective methods of PCB soldering are being applied. However, wave soldering is still very much in use. It is a very fast process that produces reliable, secure connections and can be applied to PCB designs with through-hole components, surface mount devices (SMDs) or both. By looking at wave soldering advantages and disadvantages we will see that when it can be applied, wave soldering is probably the best PCB soldering process for your design.

KNOW THE TIDE: WAVE SOLDERING ADVANTAGES AND DISADVANTAGES

Just as a surfer has to be mentally and physically prepared before taking to the wave, you have to prepare your PCB design before the wave soldering process can be successfully applied. An essential requirement is that the design files include a solder mask layer.

For most PCB design software programs this is taken care of for you. However, if you have to create your own part library (symbol and pattern) for a component, you have the option of determining what is on each layer. Care should be taken to ensure that areas that require solder, for example, pads, 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. This can occur for parts that do not have common traces but are close together. Pads without sufficient spacing can be fused together during the soldering process, which will cause shorts and possibly part destruction during operation.

Preparing for the process

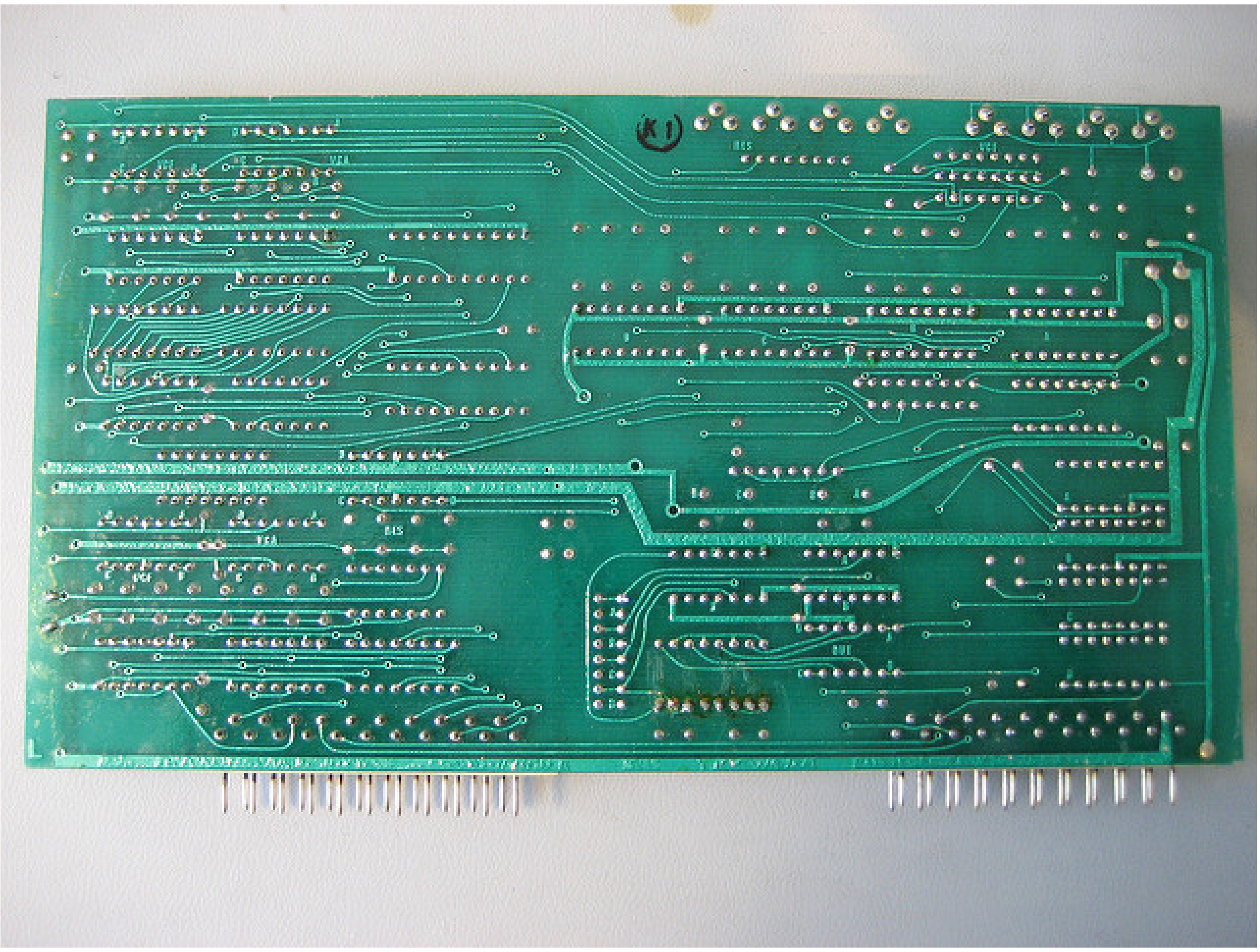
SURF'S UP! THE WAVE SOLDERING PROCESS

Wave soldering has been around for a long time and was probably the most common PCB soldering process when circuit boards, as well as the components, were larger than they typically are today. The basic process is sliding the PCB over a flow of solder and the solder feels in the exposed areas on the surface, which are the pads and other areas that require solder. The major steps involved are:

Flux Application: Flux is applied to the board to ensure that the places to be soldered are clean and free of oxidation. As with all electrical connections, dirty connections can interfere with the current flow that can cause problems during circuit operation.

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.



PCB assembled using wave soldering

For smaller boards, especially with components that are spaced close together, the wave soldering process is disadvantageous. It can be difficult to keep the soldered connections separated which can lead to circuit failure. However, when your PCB design calls for a large PCB, with components that are spaced apart wave soldering is the best soldering process. It has the advantages of reliability and speed over selective methods.

Using PCB design software that is intuitive to use, and gives you access to smart component placement will make your design-life hang loose. With Altium's Circuit Studio PCB design program you can design boards to take advantage of the PCB wave soldering process.

For more information on wave soldering and when and how to design your PCB to take advantage of this fast PCB soldering process, contact an Altium PCB design expert.

ADDITIONAL RESOURCES

Thank you for reading our guide on PCB Manufacturing Production and Process. To read more Altium resources, visit the Altium resource center here or join the discussion at the bottom of each original blog post:

- PCB Design and Fabrication Tips to Prevent Open Circuits During PCB Manufacturing
- Taxes Aren't All You'll Pay Without PCB Manufacturing Process Reviews
- PCB CAD Design for Manufacturing Guidelines: How Trace Routing Can Affect Solder Joints
- Wet and Dry PCB Etching Solutions: Which Process is the Right Solution?
- Keep Your PCB Assembly and Production Processes Like Your Best Pants: Seamless
- Understanding Solder Bridge Shorts: Bake Your Cookie and Eat It Too
- Paint by Numbers: Solder Paste Stencil Design Guidelines to Reduce Shortages
- Riding the Wave: When Wave Soldering is the Best PCB Soldering Process