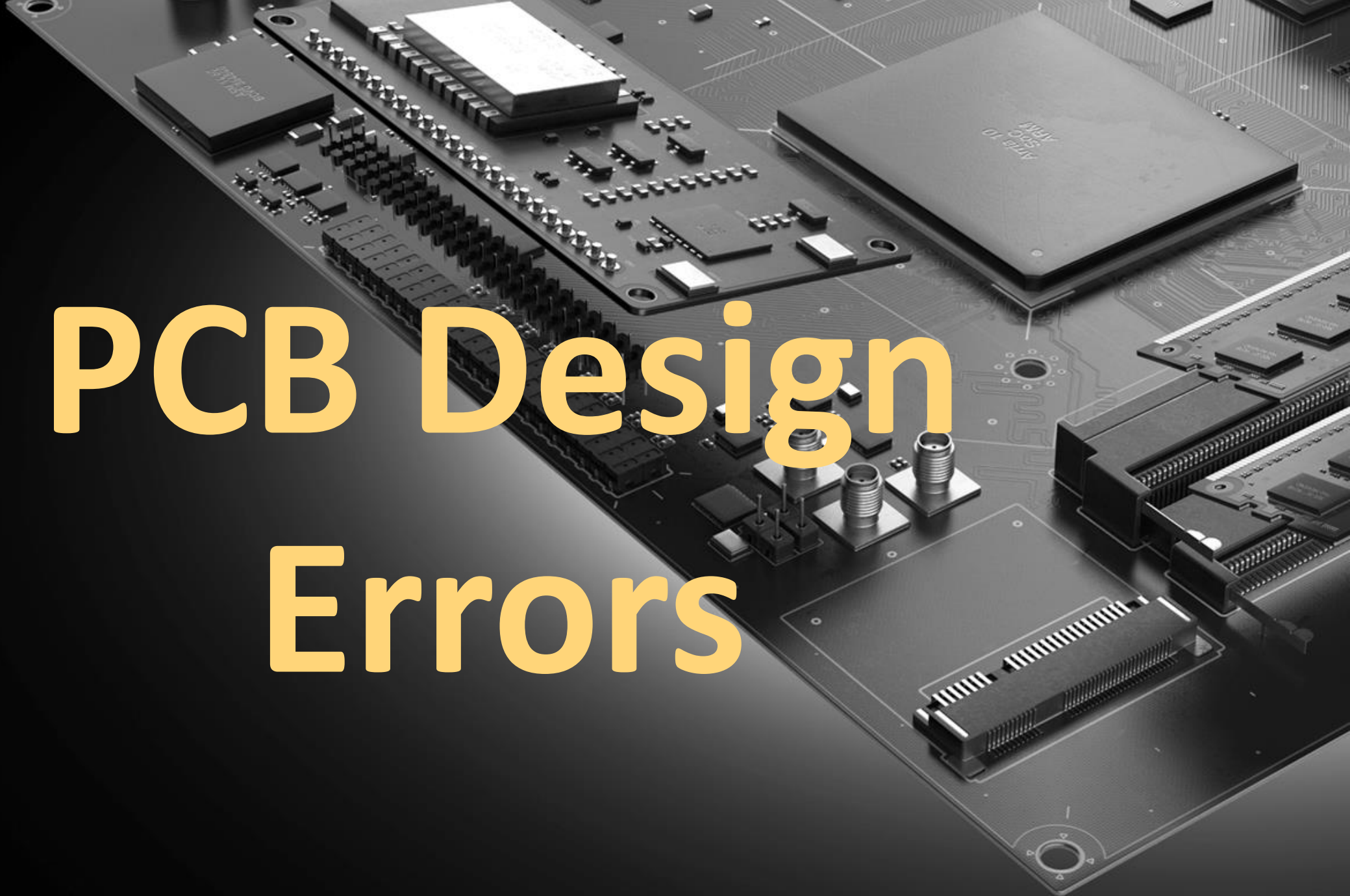


Altium

PCB Design Errors



1. PREVENTING CRITICAL PCB CONTAMINATION
2. COMMON CAUSES OF PCB SHORTAGES
3. COPPER LAYER DEFECTS
4. DON'T LET LITHOGRAPHY ISSUES CAST A SHADOW ON the PCB MANUFACTURING
5. HOW TO LOCATE SHORT CIRCUITS ON A PCB

1

Preventing Critical PCB Contamination

Preventing Critical PCB Contamination

On PCBs, contamination is more subtle—impairing PCB functioning, damaging insulation, and causing shortages. Knowing about it is the first step to avoiding it in future processing and designing.

Effects of Contamination

Contamination degrades elements enough to impair functioning.

Electrochemical migration is one of the biggest concerns.

When ions are left on the board, usually by an etchant or other chemicals, an electric potential can cause “reverse plating” that pulls metal away, and redeposits new “traces”—only there to cause trouble and strife in their tiny, damaging forms.



Preventing Critical PCB Contamination

Contamination on a PCB

- Solder Paste Processing** - The solder paste applied to the board can also get contaminated. Contamination can get into the paste during the application and reflow process. Any impurity or foreign material is especially problematic for solder because it can cause tin whiskers, which will create electrical shorts between solder joints or traces.
- Etchant and Chemicals** - There are a variety of acids and potentially damaging chemicals that get used during processing. Etching is an obvious source of acids, but flux, electrolytic solutions, and water-soluble soldering can all leave chemical residue on the board after their application and use.



Preventing Critical PCB Contamination

Prevention - Humans are a common source of contamination, with skin cells and hair landing on boards throughout manufacturing, especially if they aren't in a clean room.

Human handling also leaves oil from skin and cross-contaminated sources on PCBs which are particularly difficult to remove. Taking precautions to ensure that the skin or hair does not contaminate the board, and by processing product in a clean environment is a good first step for avoiding these contaminations.

Making sure that manufacturer also has a **clean production environment** with strong **process controls** is the second step to prevent contamination.

Verify the quality of materials being used, especially solder masks, and solders that contain flux.



[See original content here](#)

2

Common Causes of PCB Shortages

Common Causes of PCB Shortages

Understanding how PCB shorts can occur is instrumental in designing PCBs that minimize the risk of shorting.

PCB Contamination and Electrochemical Etching - Contamination can take many forms. Incompatible chemistry or improper cleaning can leave films and residues on the surface of the board, between the laminates, or between copper layers internal to the PCB. This contamination can cause damage or degradation, and short circuits.

When metallic or conductive salts are present in the layers of PCB, electrochemical etching can cause them to form filaments that are officially called Conductive Anodic Filamentation.

Copper dendrites are more common on the surface of a PCB; they are also formed by electrochemical etching, and are more likely when the surface of the board is contaminated, especially by acids or ionic solutions.



Common Causes of PCB Shortages

Understanding how PCB shorts can occur is instrumental in designing PCBs that minimize the risk of shorting.

Ensuring Safety with Solder Paste Stencil Alignment - When a PCB is manufactured, the components are placed into position before going into a reflow oven.

If the components are placed incorrectly, either shifted or rotated just a little, then the leads may not match up with their corresponding pads. The stencil used to apply the solder paste might also have aperture sizes that are mismatched to the size of the PCB pad.



Common Causes of PCB Shortages

Avoiding Overcrowding and Reducing PCB Shortages – Component sizes are decreasing, resulting in smaller pad pitch sizes and higher component density.

With a high component density there is a higher potential for heat-damage, and interference between parts. While the size of the component pads isn't directly causing electrical shorts on the PCB, smaller pads make shorts more likely. The reason is that smaller pads have less margin than larger pads, hence there is less buffer in the event of layout issues.

But, if the pads are large compared to the spacing between them, it is more likely to have solder bridges form over that narrow distance, and Optimum Design reports that solder bridges cause as up to 15% of PCB defects.



[See original content here](#)

Common Causes of PCB Shortages

Layout Concerns for Trace Routing Designs -The trace routing can affect the quality of solder joints and depends on.

- Width of the trace connection to the pad
- Number of connections you have to a given pad
- Pad size (larger pads gives more margin to work with)
- Consistency of sizes between different traces



[See original content here](#)

3

Copper Layer Defects

Copper Layer Defects

Internal copper layers and plating are critical for getting current where it needs to go, but improper layer-stacking, routing, or design can leave strong design ineffective and powerless.

Compatibility and Causes of Copper Layer Defects - When a multilayer board is manufactured, the laminate and copper have to be carefully chosen for compatibility across multiple parameters.

Etching resolution - Thinner copper enables to have finer resolution. When etching copper, for sharper edges or more precise geometry, use a thinner copper. This is especially true for etched trace lines that are less than 0.005”.



Copper Layer Defects

Internal copper layers and plating are critical for getting current where it needs to go, but improper layer-stacking, routing, or design can leave strong design ineffective and powerless.

Coefficient of thermal expansion (CTE) mismatch - More heat is required to process thicker, heavier copper. A large enough CTE mismatch can cause delamination between the layers and start to warp the PCB surface.

Warping limits the feature size in imaging and thus the solder mask. Copper and FR-4 boards are at a higher risk of CTE mismatch, which can be compounded by the amount of resin used with the laminate.

Thicker copper layers also exacerbate the mismatch. The key is to select materials that can tolerate the harshest environment they will experience.



Copper Layer Defects

- **Outgassing** - Manufacturing the layer stackup should include rigorous outgassing processes to remove any air bubbles from the laminate. Make sure the outgassing during the board manufacture is enough to eliminate trouble later in processing.
- **Through hole plating** - Vias and through holes can also experience plating failures. If the via or through hole doesn't have enough material internally, it will change the resistance of the pathway and can affect the board's performance.
- **Pad lifting** - The pads on the surface of the PCB can delaminate from the surface, just like any other layer of the board. Pad lifting also happens when the pads get too hot, almost always due to heat applied during rework.



[See original content here](#)

**4**

Don't Let Lithography Issues Cast a Shadow on the PCB Manufacturing

Don't Let Lithography Issues Cast a Shadow on PCB Manufacturing

Learn how to properly plan out the lithography process, as well as understand the factors which might affect it and cause defects.

How is PCB Manufacturing Affected By Lithography?

The etching and electroplating processes used in PCB manufacturing aren't very directional. Without a patterned protective coating on the board, everything would be etched, plated, or coated indiscriminately. The protective pattern may be a metal stencil, a sheet of polyamide, or a resist.

Resists are light sensitive (called photosensitive in the "biz") to specific wavelengths of light, usually ultraviolet. A photolithography process is used to expose specific areas to light, so the resist in those areas is cured or hardened.



Don't Let Lithography Issues Cast a Shadow on PCB Manufacturing

Sources of Lithographic Defects

- Shadows** - Sometimes, there are particles falling on the mask and casting shadows when the light should be curing the resist, like putting a chair in front of a projector and cutting off a small part of the image. That means some areas won't harden the way they're supposed to and instead will get etched during the next processing steps.

- Aspect ratios** - A common issue is a slight angular misalignment of either the light, the mask, or the PCB. On the PCB, that means the resist mask will be applied with an incorrect aspect ratio . When the board is done, there will be a higher risk of poor quality solder joints, which frequently result in open circuits. If the board isn't cured properly, there will be inner and outer layers experiencing shrinkage, expansion and warping at different rates, which can cause aspect ratio issues, too.



Don't Let Lithography Issues Cast a Shadow on PCB Manufacturing

•**Dose and exposure issues** - The “dose” of light that a resist gets exposed to is calculated like an integral—it’s the brightness of the light over the total exposure time. If the timing is off, or the light is too dim, the resist might not get a high enough dose to harden all the way.

•**Alignment** - Misalignment can also be a horizontal or vertical translation, where patterns are laid down millimeters or less away from where they should have been. As feature sizes decrease, small misalignments have larger impacts, since it’s easier to miss a solder pad or via entirely. With decreasing tolerances, make sure the manufacturer can produce and align with features small enough to match the board specifications.

•**Tolerance** - The tolerance around feature size depends on the process. Some etchants are prone to over-etching into regions ,need protected, so design the mask layer to account for that. Stenciling a paint or silkscreen layer is much more forgiving.



[See original content here](#)

5

How to Locate Short Circuits on a PCB

How to Locate Short Circuits on a PCB

Explaining major hardware failures on burned fingers is one of the worst meetings. The best thing to do is offer a plan forward. Here's a starting point for tracking down that short circuit.

Visual inspection - The first thing to do is to carefully look at the entire surface of the PCB. Look for tin whiskers between pads or solder joints. Any cracks or blobs of solder should get careful attention. Check all the vias.

If the short is really bad, burned spots on the PCB can be seen. They can be quite small but will be dramatically discolored brown instead of the normal green solder mask. If there are multiple boards, a burned PCB can help narrow down the specific location without powering up another board to sacrifice in the search.



How to Locate Short Circuits on a PCB

Potential Locations of Short Circuits on the PCBs

Bad Components - Some components have a tendency to go bad, like electrolytic capacitors. In case of suspect components, check those first. If not certain, then usually do a quick google search for components that suspect “failed” to find out if it’s a common issue.

Infrared Imaging - Using an infrared camera can help to locate a short circuit, even if the short is between internal layers. The short is usually higher resistance than a normal trace or solder joint since it didn’t have the benefit of being optimized in the design . That resistance means the short circuit will heat up, and announce itself in an infrared view.



[See original content here](#)

How to Locate Short Circuits on a PCB

Potential Locations of Short Circuits on the PCBs, cont.

Digital Multimeter Testing - The multimeter approach gets mixed reviews in most electronics forums, but tracing the test points can help figure out what isn't the problem. A very good multimeter is needed with milliohm sensitivity, and it's easiest if it has a buzz function to alert when probing a short.

The Last Straw - Destructive Testing - Destructive testing is obviously a measure of last resort. Going back to the multimeter confirmed that most sections did *not* have Vcc and ground tied together. But that single quarter of the board was a little black hole of mystery. Make sure there is a robust rule checking, for errors, design issues, and manufacturer tolerances.



[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 the attention!