High-Reliability PCB Design Tips for Aerospace and Other Harsh Environments

SUMMIT
INTERCONNECT

High-Reliability for Aerospace

3, 2, 1 ...Liftoff

Harsh-environment and high-reliability engineering often go hand-in-hand. Whether a sensor package is placed on an airplane, a spacecraft, or down the borehole of a wellhead, the cost of failure can often be measured in hundreds of lives or millions of dollars lost.

For these situations, engineers have to pay particular attention to design choices. The following article presents design considerations for your next aerospace project.

Quality and Reliability

When a pinted crcuit board is manufactured by an accredited facility, an employee is assigned the task of ensuring the board meets or exceeds expectations in all aspects of the design: plating thickness, surface finish, planarity, etc.... These are measurable, quantifiable metrics, and the board either meets them, or it does not. The PCB manufacturer's responsibility for quality usually ends the moment the PCB is delivered to and accepted by the customer. At delivery, 100% of the boards are expected to function as designed.

Reliability covers the lifecycle of the product from delivery to dumpster -- and it is a much harder metric to quantify because board failure dates are unknown and variable. Several factors affect reliability, including PCB design, manufacturing process control, and material variability. Even if the design follows all best practices, and the processes are carried out flawlessly, there is always a bit of material variability that introduces slight variations to manufactured printed circuit boards. This variability, along with poor design choices and poor process control, can hide latent defects inside a design.

Quality is easy to determine -- either the board meets specifications, or it does not. Reliability is hard -- because a board might fail 1-year, 5-years, or 20-years into its service life.

High Reliabilty Fabrication

IPC Specifications

IPC specifications applicable to the PCB design process include:

- IIPC-2221 Design Guidelines for Printed Circuit Boards
- IPC-2222 Design Guidelines for Printed Circuit Boards for Rigid Boards
- IPC-2223 Design Guidelines for Printed Circuit Boards for Flex Rigid-Flex Performance Requirements
- IPC-6011 IPC-6012, IPC-6013, IPC-6018

No one individual designer or even a group of designers will know all of these specifications inside and out, which is why it is important to work with a PCB manfuacturere that understands how the requirements within these specs are levied.

Engage with Manufacturers Early

Product developers can enage with manufacturers at any stage. Some do it very early when they are doing the stackup and material selection. Here it is important to know the kind of drill spans they are using; whether or not they are using multiple sublams; where they are starting blind vias, buried vias and micro vias and if those vias are stacked or staggered.

At a minimum, the required specs for reliable PCBs are the IPC-222X series design and the applicable IPC-601X series performance document. Any exceptions to they guidelines should be agreed upon in writing.

PCB Reliability Requirements

Circuit boards used in aerospace applications or other harsh environments should be able to withstand up to 2000 cycles from -40 degrees Celcius and up to 30 cycles at 570 degrees Celcius.

In order to meet these reliabilty requirements, designers need to select the right high-quality materials.

High-Speed Materials

Materials capable of withstanding extreme heat are ideal for aerospace applications. These can include aluminim, anodized aluminum, and copper.

Thermal Management

It is often advisable to consider incorprating heavy copper technology into high-reliability designs. Heavy copper allows for improved heat dissapation without requiring additonal cooling systems.

Conformal Coating

PCB finishes also contribute to dependibilty and reliability, helping to withstand harsh operating conditions. Preferred conformal coating includes electrolytic nickel gold, ENIG (Electroless Nickel with Immersion Gold Coating), chemical silver, HASL (Hot Air Solder Leveling) and lead-free HASL.

Maintaining Connectivity

Electronic components must be connected to each other to function. Silicon dies are connected to metal lead frames with bonding wires, lead frames are connected to copper pads with solder, and copper pads are connected to each other with traces, copper pours, and vias. To maintain a high-reliability board, all of these connections must remain intact over the lifespan of the device.

Thermal Expansion

When the temperature of a printed circuit board or its components increases, the inter-atomic distance increases as well -- this is called the Coefficient of Thermal Expansion (CTE). The relationship between separation distance and temperature varies with material composition. Additionally, for PCBs, the CTE is often anisotropic -- which means the value depends on the direction.

FR-4, the basic building block of most boards, is made of a two-dimensional weave of fiberglass strands. The weft-and-warp fibers help to minimize in-plane expansion (x-axis, y-axis). But they do little for out-of-plane (z-axis) thermal expansion.

Not all weft-and-warp fibers are identical, some dielectrics purposely use different glass fibers which will result in slight CTE variability by axis. Check dielectric's slash-sheet[1] to find out more about your laminate materials.

Via Failures

The CTE for copper is low compared to the CTE for PCB dielectric and base materials. When a via is formed in a circuit board, and the board is heated, differential z-axis expansion leads to shear stress between the via wall and the dielectric layer. This force will eventually cause a failure in either the via or the copper foil.

The via aspect-ratio, the amount of temperature fluctuation, and the quality of the PCB plating process will determine whether the via will fail on the tenth thermal cycle or the ten-thousandth thermal-cycle.

Vias and Pads

Vias are holes in printed circuit board that are electroplated with copper. The least expensive design option is to leave those holes open to the air. This can be an unreliable design option for several reasons. For example, open vias can capture contaminants such as acidic flux that will slowly eat away at the copper over time.

But one often overlooked problem associated with open vias is solder thieving. During the reflow process, solder paste will liquify and follow exposed copper to a hole where it will enter and fill the hole. This leaves the copper pad with an insufficient amount of solder metal to form a proper bond between the part and the PCB.

Adding unfilled vias to a thermal pad can cause issues as well. Not only does the via allow solder thieving, but it can also allow outgassing into the space between the part and the PCB, forcing solder away from the thermal pad.

The solution to this problem is to not leave open vias beneath or near solder pads.

For vias near pads, use the solder-resist layer to provide a barrier between the solder pad and the via. Via-tenting is another option, but it is bes suited for the smallest of vias. A more reliable option is to fill the vias with LPI solder-resist.

For vias beneath pads, the most reliable option is to use filled and capped vias where a layer of copper is electrodeposited atop an epoxy fill material. Another option is to use small diameter vias (<12 mil) that are then filled during electroplating with 1-oz or thicker copper -- there will still be holes or voids present that can allow outgassing, but the volume of the remaining hole should be small enough to minimize solder thieving.

Solder

Before the RoHS act, the majority of components were attached to circuit boards with a Lead/Tin eutectic solder. Eutectics are combinations of metals that when mixed have a low transition temperature that allows a direct solid-to-liquid phase transition. This is important to electronics manufacturers beacuse high tempertures will destroy or severly shorten the lifespan of printed circuit boards and their components.

Engineers were concerned that they would never find material as ductile or wet table as Sn-63 Pb-37 mixture, so metallurgists got to work mixing various compositions of Aluminum, Antimony, Bismuth, Cadmium, Chromium, Copper, Gold, Indium, Lead, Molybdenum, Nickel, Palladium, Platinum, Silver, Tin, Tungsten, and Zinc to better understand the phase diagrams and material properties.

Engineers discovered that there were dozens of alloys suitable to act as solders in an RoHS application. And, since these solders all had different material properties, they could be used for different applications. For example, Transient Liquid Phase Sintering Paste (TLPSP) is a type of solder paste that liquifies once. Once the inter-metallics have formed, the phase-diagram changes and the liquid-phase disappears. Once melted, TLPSP cannot be re-melted without destroying the components and PCB. This makes the paste useful for multiple-reflow cycle and harsh-environments.

Summary

Designing for reliability entails a wide array of design choices, material selections, and carefully controlled manufacturing processes. Focusing on thermal management will help improve your board's lifespan and reliability. But it is just one part of the equation. You also have to make decisions that limit the temperature rise of your board by either limiting heat input or increasing heat-sinking.