The Top 10 Issues that Cause Bad Prototypes

Engineering teams are constantly challenged to speed product development with fewer internal resources. As a prototype assembly house we are regularly called upon to fix issues driven by compressed product development cycles and thought it would be valuable to create a "Top Ten" checklist of issues that we see on a regular basis. Feel free to keep this as a reference list for your product development process. Additional resources can be found at www.screamingcircuits.com.

Issue 10. Ambiguous Part Substitutions

Some applications are tolerant of "fuzzy" part substitutions, but others need parts that are exactly to specifications. Components that can cause problems include: resistors that are close to specification values but not exact, passive component sizes that are close but don't match up, barrier diode ratings that don't match. Motor control and RF-based products are two examples where variance from specifications will likely cause significant product performance issues. Don't take for granted that it doesn't matter. Check potential application issues before substitutions are made.

Issue 9. Silk Screen Marking

Silk screen markings for diodes sometimes include ambiguous symbols making it difficult to determine whether anode goes to negative or positive. Properly identifying components in a non-confusing manner helps ensure correct placement. While a silkscreen is not an absolute requirement, if you are outsourcing boards without silk screen, you should provide some other form of documentation to verify component placement and rotation.

Issue 8. Poor Raw PCB Storage and Handling Practices

Different PCBs finishes are susceptible to damage as a result of incompatible handling or storage practices. Silver boards are sensitive to corrosion, light and oxidation. Gold boards, particularly those with thin layers of gold, are also sensitive to defects caused by poor handling or storage practices. Most boards need to be moisture-sealed. Storage in humidity-controlled, dark environments is best.

Issue 7. Excessive Moisture Exposure of Components

Moisture sensitivity is predominately an issue with RoHS-compliant components, but can be an issue in some leaded components, as well. After improper storage, moisture-sensitive chips may popcorn or crack subtly underneath. This can create hard to find or intermittent defects. It is often more of an issue with prototypes because components are often consigned in partial lots. This may result in impaired moisture sensitive packaging or storage beyond recommended shelf life. Be sure to evaluate parts in opened packages and bake moisture-sensitive that have not been stored properly.

Issue 6. When RoHS is not RoHS

Some components exist on the ragged edge of RoHS. While they meet the RoHS-compliance specifications in terms of materials content, they have limited ability to withstand the higher heat requirements of RoHS products. For example, a single metal can capacitor can solder acceptably at RoHS temperatures, but the thermal mass of several can capacitors close together may drive a longer heating requirement, which can destroy the components. Similarly, PCBs can also have this issue. FR4 typically can only take 1-2 heat cycles. In situations where component layout creates areas of high thermal mass or if multiple heat cycles are required, specify components and substrates equal to the challenge.

Issue 5. Layout Issues Which Cause Tombstoning

Layout inconsistency is one trend that drives tombstoning issues. For example, when a small trace is going to one pad and a large trace is going to another pad, the large trace will act as a heat sink. The smaller pad melts first and surface tension causes the large trace's side to pop up. Inner copper layers under one pad, but not another can also cause the same effect. Larger thermal mass components can also cause this. Another common cause of tombstoning is a thick solder mask. Good mask registration will help. Using a board vendor that has tight tolerances and delivers a thin flat mask surface will also help. In some cases, with the smallest parts, you may want to keep the soldermask off the pads by using non-solder mask defined (NSMD) pads.

Issue 4. Lead vs. No Lead

Older components don't always have a lead-free cross and newer components may only have a lead-free version. In the case of older BGAs, microBGAs, CSPs mixing lead and no lead doesn't work. If lead and no lead must be mixed, it is easier to put a lead-free BGA on a leaded board than the converse. The end result is not a reliable board, but may be acceptable for prototyping. The converse situation overheats and evaporates all the flux out and overcompresses. Reballing may be cost prohibitive, but is sometimes considered as an option with more costly BGAs.

Issue 3. Parts Library Mismatch

CAD system parts libraries often contain land patterns that are close to that needed for the actual component, but just enough off to create issues. For example, parts with uneven solder pads cause headaches on a fairly regular basis when matched with PCB land patterns that don't also follow the uneven dimensions. The surface tension of the molten solder will act on the bigger surface area of the pad and cause the part to shift during reflow. That can lead to reliability issues. In some cases, it can also lead to shorts with nearby components or mechanical structures. Sometimes there is variation between metric and English measurement systems in parts libraries. That difference can be irrelevant in a part with a few leads but significant in a part with a large number of leads.

Issue 2. Floating QFNs

Tiny QFN stencils with a fully opened area for the thermal pad can place too much solder paste and QFN can float up, preventing some of the side pins from connecting. The center pad area in the stencil should

be segmented to deliver about 50% solder paste coverage. With large QFN parts, the paste squeegee can deform and leave too little paste resulting in excess void. Again, segmenting the can preven the problem.

Issue 1. Via-in-Pad

Poorly designed vias-in-pads are the number one cause of bad prototypes that we see. In many cases, it is unavoidable as discrete components. QFNs and fine pitch BGAs often require via-in-pad. But if done poorly, solder capillary action sucks the solder through the via and down to the other side of the board. Small parts can tombstone, large parts end up with poor mechanical connections, BGA balls can be sucked off the component. If via-in-pad must be used, cap it and/or mask it. Capping the chip-side is better than capping the bottom because it reduces chance of outgassing and voids. At the least, make it as small as possible. For QFNs, cap or tent the vias.

The Screaming Circuits blog http://blog.screamingcircuits.com/ regularly discusses the issues listed here in greater detail. Feel free to visit and learn more.

Screaming Circuits (www.screamingcircuits.com) is an industry pioneer in the online quick-turn prototype pc board assembly market. Located in Canby, OR, the company offers assembly, prototype, pilot production and short-run production pc boards in as little as 24 hours for short runs. Screaming Circuits is a subsidiary of the Milwaukee Electronics Corporation, which also offers full service contract manufacturing through facilities in Wisconsin, Oregon and Mexico, and design services through its subsidiary, MEC Innovation.