

Werner Engelmaier: “Mr Reliability”

When it comes to printed circuit boards and related assemblies, Werner Engelmaier brings over 50 years of practical engineering experience in design, process and product evaluation, materials and assembly testing, failure analysis and reliability assessment.

In his very informative White Paper Report “**Recommendations for PCB FAB Notes and Specifications in Printed Circuit Board Drawings for SnPb and Lead-Free Soldering Assemblies, the Qualification of PCB Shops and Activities to Assure Continued Quality,**” Werner discusses

可靠性先生



what needs to be considered to have a successful assembly operation especially in the lead-free environment. For more information and to obtain your copy of this educational and timely report, visit: www.engelmaier.com. Werner can also be reached at Engelmaier@aol.com.

I spent some time discussing the causes of delamination with “Mr. Reliability,” as many in our industry have dubbed him, about this very concerning issue at the recent IPC Midwest Conference in Schaumburg. Here is our conversation:

GP: Why does delamination seem to be more prevalent with the use of the new materials that require lead-free or the RoHS processes?

WE: There are a number of reasons that combine—fillers are added to the resin to reduce CTE’s, this can reduce cohesive strength; the higher temperatures required for Pb-free soldering result in more than a doubling of the vapor pressure internal to the PCBs; there is evidence—at least anecdotally—of some of the resins with RoHS-survival capability to be more brittle.

GP: Baking was once a standard procedure prior to assembly, why have most assemblers seemed to have dropped this step?

WE: In short, cost reduction. And a belief that they (the assembler) can get away with it. On the latter, that may have been true for SnPb-soldering, but assuredly is not true for Pb-free soldering. As I said, from 220 to 260°C—the maximum peak temperatures reached by the PCBs in SnPb- and Pb-free soldering, the vapor pressure more than doubles. Adequate moisture bake-out is imperative.

GP: With the advent of lead-free and the new materials that are required for those processes, why is baking prior to assembly more important than ever?

WE: As I said, from 220 to 260°C—the maximum peak temperatures reached by the PCBs in SnPb- and Pb-free soldering, the vapor pressure more than doubles. Adequate moisture bake-out is imperative. There also are statements in the industry—without any data that I have seen, that some of these resin systems may absorb water vapor more readily.

GP: Assemblers are balking at the need for baking prior to assembly. The hesitation of an additional process that would lengthen the time required to produce an assembly is understandable, but whose responsibility is it to bake—the PCB manufacturer or the assembler?

WE: Baking at the PCB manufacturer makes little sense, because during most shipping and storage steps moisture is absorbed—only nitrogen- or vacuum storage would prevent moisture absorption. Thus, to be effective, moisture bake-out needs to happen just prior to assembly—but it is the customer, the OEM, who really needs to specify it.

GP: What would be a good suggestion as to the packaging and storage of PCBs?

WE: No packaging is truly hermetic; it can only slow down moisture absorption. Desiccants also help. But since none of this is guaranteed to prevent moisture absorption, baking, regardless of the packaging or storage conditions, is highly advisable just prior to assembly. The only exception is testing representative coupons that have been packaged and/or stored identically and not finding any problems.

GP: What would you suggest to be a proper bake cycle for PCBs prior to assembly?

WE: I would suggest 105°C (even though 125°C have seen successful use) for 24 hours; increasing the temperature may jeopardize solderability and the reduction in the required time is not in relation to the risk.

GP: Besides baking, are there any other suggestions or that today's PCB assembler should consider for a successful assembly operation?

WE: Yes, as an assembler, I would want to see the 'FAB Notes' for the PCBs I am asked to assemble to see what actually had been specified for the PCB. As an assembler, I can do everything right, and things still can go rather wrong, if the PCB specifications are inadequate for Pb-free solder assembly.