AGC
Your Dreams, Our Challenge

nelco

Q&A

Q&A for those involved in Contract Manufacturing using Nelco PCB Materials

1. **Do Nelco laminates have any discoloration effects or staining issues after multiple high temperature exposures?**
   - Every laminate will darken after multiple high temperature cycles. How much is dependent on the temperature and duration of the cycle. Laminate exposed to temperatures approaching the decomposition temperature will darken more than those exposed to lower temperatures. It should be noted that the discoloration effects of the solder mask may obscure changes in the underlying substrate.
   - Laminates usually do not stain after multiple high temperature exposures unless they have been contaminated with chemicals. Certain water-soluble fluxes can cause staining and solder masks will retain high ionic contamination levels after processing with certain fluxes.
   - If the solder mask surface does display cloudy white stains, either as circular spots or swirling patterns, these spots can normally be removed with a post baking process of 220°F / 104°C for about 2 hours.

2. **Are there any contrast issues at AOI related to Nelco® laminates during assembly?**
   - If a laminate darkens during the assembly process it should not have a negative impact for most AOI processes, since these inspections normally rely on the contrast of the solder mask.
   - If the AOI works on reflectivity and/or fluorescence, these mechanisms are also largely unaffected by changes in color.

3. **Does the laminate contribute to the amount of warpage in the panel or finished board?**

   Laminates can contribute to PCB warpage in a number of ways:
   - First, the boards may have been built with an unbalanced construction. This generally occurs when boards are designed with odd number of layers or have unbalanced dielectrics on either side of the center plane of the board.
   - Second, the boards may have a balanced build but the cores themselves are asymmetrical and use prepregs that are of dissimilar styles. A 0.005" core, for instance can be made with a single ply 2116, two plies 1080 or with a ply of 2113 and 1080. The latter build could generate warpage if the orientation of the two dissimilar plies are not properly distributed around the center plane of the board.
   - Copper distribution on a board/panel, can also cause warpage. The designers should try to balance the copper around the center of the boards to minimize warpage. The fabricators should use a venting pattern that provides balanced amounts of retained copper without sacrificing the ability to vent out moisture and air during lamination.

- Exposing the boards to high temperature can cause warpage during PCB fabrication and at the assembly stages, especially if there is a front to back gradient. Warpage can also be induced during the lamination cycles of the multi layers. It can also occur at HASL at the board shop or during reflow at assembly. At the reflow, process panel thickness, panel size and component weight can impact warpage.

4. **What is the coefficient of thermal expansion for laminates and what impact does it have to the assembly process?**

The thermal coefficient of expansion is a dimensional change due to change in temperature.

- Each laminate may have a different coefficient of thermal expansion (CTE) in the X, Y and Z axes. The CTE in the X & Y axes are most strongly influenced by the reinforcement used. For boards built with E glass, the CTE's are generally 12-16 ppm/°C.
- Z axis CTE is also important. When a board is constrained in the X & Y axis by the reinforcement, it wants to move in the Z axis when it is exposed to heat. We generally report these differences as a percent change in expansion as a board is raised in temperature from 50°C to 288°C. The amount of expansion depends on the board design, the type of resin content, and reinforcement. One can expect to see expansions that range from a low of 1.5% to a high of almost 7%. For obvious reasons, you do not want to have a high Z axis expansion for high layer count and/or thick boards. If you do, you may introduce reliability issues during thermal cycling. Typical defects can include barrel cracks, delamination, etc.

5. **What is the impact of wave solder to a laminate?**
   - Wave solder has little impact on laminate if its process is properly controlled.
   - Moisture uptake needs to be considered to avoid delamination and therefore baking is recommended prior to wave solder.

6. **What is the impact to a board or panel during an inline / offline wash process?**
   - Bare laminate will absorb more moisture within a certain period of time than a finished board (i.e. one covered with a solder mask). Additionally, how often the boards/panels are washed and the duration of the washing affects the moisture uptake for each laminate.
   - Short wash cycles of properly stored laminates are not normally an issue if they proceed directly to the next process.
   - Check for chemical Resistivity (saponifier, acetone etc.)

7. **How sensitive are Nelco® laminates to moisture?**
   - The amount of moisture a laminate absorbs depends upon temperature, humidity and duration of processes. We generally test our laminates using standard IPC specifications.
   - It should also be noted that each laminate family has a different absorption rate. This is a critical point because it is necessary to bake some laminates longer than others to remove moisture that may cause delamination during high temperature excursions such as reflow or wave soldering.

8. **How often can a board (laminate) be reworked?**
   - The amount of rework a circuit board can tolerate depends on several factors. The type of material used is one obvious key, but things such as copper peel strength, heat, mechanical force, pad size, components size, component weight, distribution of copper, construction and thickness can also be important.
   - In order to reduce the problems caused by rework, the laminate should be processed based on the properties specified in the product literature, especially with regard to the maximum thermal exposure temperature (reflow, wave solder & hand rework).

- Laminates with low CTE and high thermal reliability, as determined by their glass transition temperature (Tg) and time to delamination (T_{260}, T_{288} or T_{300}), will perform better in multiple rework cycles.

9. What is the maximum temperature and duration that Nelco laminates can be exposed to?

- The laminate should be processed based on the properties specified in the product literature, especially with regard to the maximum thermal exposure temperature (reflow, wave solder & hand rework).
- However, some factors such as exposure time and circuit construction may affect the maximum use temperature. When these factors are present it is important to consider the RTI (Relative Thermal Index) and the MOT (Maximum Operating Temperature) for best results. These values can be found in the "Interactive Product Comparison Table" on the website.

10. How many temperature cycles can a Nelco laminate go through?

- Laminates will degrade with each applied heat cycle. The closer the temperature comes to the thermal decomposition temperature, the faster the degradation process. Even at lower temperatures the degradation is in progress but at a slower speed.

11. When does peel strength affect rework? How often can a pad be reworked, and at what temperature?

- Rework requirements depend on several factors, such as copper peel strength, applied heat, mechanical force, pad size and the specific rework process.

12. What is the importance of the chemical resistance of laminates?

- Chemical resistance can be important for cleaning the boards (clean, no-clean flux). When certain chemicals are added to the wash process they can cause loss of performance. Often boards are contaminated and it is impossible for a human eye to detect. This contamination can cause field failures and should be avoided. When using inline/offline cleaning, it becomes very important to carefully consider any applied cleaning chemicals. Generally, DI-water and alcohol are good solutions to choose.

13. How do Nelco laminate properties affect Scoring or Break-off related issues in an assembly process?

- Some panels have to be broken apart into boards at some point. Depending on panel size, thickness, weight, overall weight of assembled panel or scoring depth, break-off can make the assembly process more challenging.
- Some laminates are softer than others after heat exposures, which can affect the stability of the panel. For a certain panel size and/or thickness it might be necessary to determine a specific handling procedure (ex. middle support for placement, reflow process etc.). Generally we recommend a review of the mechanical properties to determine if additional handling procedures are required.

14. What are the root causes of delamination during assembly?

- The root causes of delamination are often low pressure in the press, moisture absorption of the laminate, contamination of the prepreg or core surface (oil, grease etc.), entrapped moisture in plugged via's, under cured laminate, non-optimized oxide and desmear chemistry.
- The weakest bond link is typically the interface between the oxided copper surface and the prepreg. Most delamination occurs at this interface.

- Usually in an assembly environment, moisture absorption combined with high temperature cycles can cause delamination.
- Generally laminates with high Tg, high decomposition temperature, low Z axis-CTE and laminates with good T_{260}, T_{288}, or T_{300} properties are better for multiple high temperature cycles.

15. Which Nelco® laminates can be used for lead free soldering?

- The details of the board fabrication process play a very important role in whether or not a specific material is lead free compatible. The copper plating thickness and the amount of moisture retained in the PCBs are two, of many things, that need to be considered.
- In a high temperature, lead free assembly process, it is important to be cognizant of the HDI design attributes. For instance PTH size, hole to hole distance, layer count, etc. also can affect the product capabilities.
- The following Nelco resin systems are recommended for lead-free soldering depending on design and reflow conditions:
  - N4000-7/7 SI
  - N4000-29
  - N4000-12/12 SI
  - N4000-13/13 SI
  - N4000-13 EP/13 EP SI
  - N4800-20/20 SI
  - N5000
  - N7000
  - N8000
  - N9000 series
  - Mercurywave 9350
  - Meteorwave family of products

- The following resin systems are not recommended as lead free materials, but under certain designs and reflow conditions have shown to be lead free compatible:
  - N4000-6
  - N4000-6FC

16. Printed Circuit Fabrication and Assembly

- Storage of a finished Printed Circuit Board (PCB) is critical for its ability to survive down stream processing and should not be discounted. There are many different resin systems on the market today, and each one of them will absorb moisture at different rates.

  As the industry has moved to higher reflow temperatures (220°C – 260°C) with the implementation of the RoHS initiative, and the increased use of lead free solders (SnPb to SAC305), the vapor pressure of absorbed moisture increases greatly as the assembly is exposed to higher temperatures. This higher vapor pressure can cause delamination or cracking of the PCB if not removed prior to SMT or wave soldering. To limit moisture uptake it is recommended that fabricated PCBs be shipped in approved moisture barrier bags that provide a layer of ESD protection

  Determining what packaging is most appropriate for the fabricated board is dependent on a variety of factors. These include, but are not limited to storage conditions of the PCB, laminate material, final plating finish, and overall board thickness. The addition of a desiccant pack along with Nitrogen gas will help minimize moisture absorption and keep the surface finish on the board solderable. Including a humidity indicator card inside the packaging will provide visual evidence once the packaging was opened.

Moisture bags should be hermetically sealed immediately after the N2 gas is introduced with the appropriate heat seal equipment. After the bags are opened, they should be limited to the open air environment to limit moisture absorption. Dry packing should be capable of limiting moisture absorption up to 1 year of useful life if the bags are stored in a stock room environment not to exceed 40^{0}C and 90% RH per the JEDEC-STD-033 standard.

Baking PCBs which are run under both a lead and lead-free soldering process is recommended for optimal performance and reliability. However you should consult with your chemical supplier to ensure that baking will not have adverse consequences with regard to solderability. As mentioned earlier, different resin systems will absorb moisture at different rates (Polyimide resins being one of special note). The design, layout and metal finish of the PCB plays a role in determining the necessary baking time. For instance a board which has been plated on all sides may need a very long bake to remove any trapped moisture if it is possible at all.

Prior to assembly, it is recommended that boards be *****rack baked in a convection oven to allow for proper air flow. **The following is a guideline for determining the minimum baking time to remove trapped moisture in a PCB. Longer bake times may be required, but may have adverse effects on solderability:**

OSP/Immersion Tin	1-2 hrs at 110C-120C (230F – 250F)
ENIG/Immersion Silver	4-6 hrs at 150C (300F)

***** The PCBs should be placed on racks and not stacked on top of each other. Boards in the middle of the stack may not see the minimum temperature to allow for proper moisture removal.

A properly profiled and calibrated reflow oven along with any hand soldering operations will be critical in achieving high reliable finished PCBs. Boards which are to see secondary or post solder processing at a later date should be stored and resealed in their appropriate moisture barrier bags or nitrogen cabinet.

The information contained in this document is believed to be reliable and accurate. However there are many factors which can affect the final outcome and it is up to the individual customer to determine what processes are best suited for their manufacturing operation to yield high reliable assemblies.