Device Shrinkage Achieved Through Consideration of Assembly Factors

Yousef Heidari, VP of Engineering, SigmaTron and Andrew Vo, Director of Test Engineering, SigmaTron

As medical devices continue to get smaller in size, the assembly of the internal electronics becomes more critical. Like a puzzle, fitting all the necessary pieces together in a smaller space is no simple task. When outsourcing is involved, synergy between teams is critical. The earlier a medical device manufacturer's design team and their contractor's team begin working together, the better.

One of the benefits of outsourcing is that many contract manufacturers have been seeing product footprints shrink in other industries for years. Constraints that may be new to a design team just beginning to develop miniaturized products are easily correctable issues to an engineering team experienced with densely populated boards and/or smaller component packaging. Key areas of focus are:

- Component selection considerations
- Design for manufacturability and test
- Assembly process planning and tooling: Stencil, fixture, and assembly considerations
- Cleanliness
- Appropriate assembly inspection processes

Component Selection Considerations

In addition to determining the best components for fit, form, and function,

designers should consider test strategy. Smaller printed circuit board assemblies (PCBA) often have limited test point access. Boundary scan testing provides one option for addressing this challenge. However, IC design must include boundary scan support and the PCBA must have a connector that complies with IEEE 1149.1 or .x connection standard. From a manufacturing and test standpoint, through-hole technology

may be the preferred package for the connector since it offers a stronger interconnection with the PCBA for plugging and unplugging test cables. If a through-hole connector is used, assembly processing should be defined. In some cases, these connectors can be reflowed rather than wave soldered or soldered with a selective soldering machine. However, if reflow is used, the connector must have high temperature plastic capable of withstanding peak reflow temperatures.

Design for Manufacturability

From a design for manufacturability (DFM) standpoint, layout must be carefully reviewed. Land pattern choices are critical because they drive the spacing. The pad size must support the density as well as the correct solder joint. IPC guidelines are a good starting point. In SigmaTron International’s [1] process, the manufacturing data sheet is referenced if the pad size appears incorrect for the specified component. In addition to the actual component’s termination, pin-to-pad and the component-to-component spacing is analyzed as part of the DFM review. If the PCBA is a mixed technology design, spacing between SMT and through-hole components is also reviewed for selective soldering and/or selective wave solder fixture design. The layout is also checked for sources of mechanical interference and to ensure that adequate spacing around the PCBA is present for automated handling. Fiducial mark locations are reviewed and fiducial marks are added, as needed, to support component placement and flying probe testing. Much of this

process is automated using Mentor Graphics Valor tools [2].

Stencil and Fixture Design

SigmaTron does stencil design in-house because of the potential impact it can have on product quality, particularly in smaller PCBAs. One of the first steps is to change the land pattern shown in CAD to the actual component footprint. This involves replacing the CAD symbol with footprint from the Valor Part Library.

The Valor software tool pairs component footprint geometry along with the layout land pattern in order to optimize the aperture’s size and locations in the stencil. Some modifications from the software-generated design may be necessary. Typically, the smallest component geometries determine thickness of stencil and necessary compensations to the aperture areas need to be made for the larger components.

A PCBA carrier is often necessary to support the PCBA correctly through the assembly operations. Typically, an SMT carrier is utilized to facilitate the set-up and to minimize warpage by supporting the PCBA during screen printing, pick and place, and reflow. Carriers also help ensure efficient processing in high mix, low volume production. They drive a repeatable, consistent process by minimizing the variability that can occur when the process becomes set-up or operator dependent.

Cleanliness

PCBA cleanliness is important in medical products. Low profile components and QFNs cannot be easily cleaned underneath the component package. No clean chemistry is recommended. If the product requires conformal coating or a higher level of solder joints cleanliness, an aqueous cleaner using Kyzen chemistry can be used. If no clean chemistry is used, any flux residue left behind under the low profile components will be no clean and benign.

Appropriate Assembly Inspection Processes

The first inspection point is normally solder paste inspection. The stencils are designed to have a good release and machines are equipped with automatic wipers that clean after every print. As a result, solder paste deposit area and height is normally consistent throughout the production run. However, PCBAs are visually inspected and sample solder paste height measurements are performed as part of standard assembly processes.

Ionic testing may be used on a sampling basis to ensure board cleanliness requirements are met. Test strategy varies based on product characteristics. For example, if many of the vias are masked, it can be more difficult to get good test coverage. Medical products often have less vias to minimize signal interference. The schematic is reviewed to determine if boundary scan is an option. Accessibility for flying probe testing is also an option.

The best test option may be boundary scan, if it is designed in and the design is compliant with the IEEE 1149.1 or .x connection standard. The boundary scan test technology present in the PCBA is to increase test coverage of the components' early manufacturing defects (shorts/opens); complex PCBAs with masked vias tend to have limited test access points. Softshell tests, electrical test for connectivity, functional test, x-ray laminography, and AOI can also be used either standalone or in combination depending on product requirements. In-circuit testing is very dependent on test accessibility. If the layout is heavily masked with low test coverage, ICT can be very high cost.

Ensuring good quality as medical products shrink ties back to the structural

integrity of the interconnections. The critical elements to focus on are ensuring correct spacing in the layout, printing the correct amount of solder, and validating quality via a good automated inspection process.

Source URL (retrieved on *01/29/2015 - 2:50pm*):
http://www.mdtmag.com/articles/2013/06/device-shrinkage-achieved-through-consideration-assembly-factors

Links:
[1] http://www.sigmatronintl.com/
[2] http://www.mentor.com/