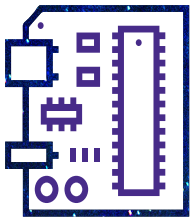
ASSEMBLY DATA & DOCUMENTATION

Process Overview: Part 2



SEGMENTS OF SANITY FROM THE GUIDE

EMA Design Automation®

Assembly Data & Documentation: Process Overview, Part 2

In part one of this two-part series on assembly data and documentation, we discussed how data drives the assembly package and listed the several items needed for it to be complete. Understanding of this and the manufacturing process will help you make sound design decisions. This article will provide a deeper dive into each step of this process.

Programming the Placement Machinery

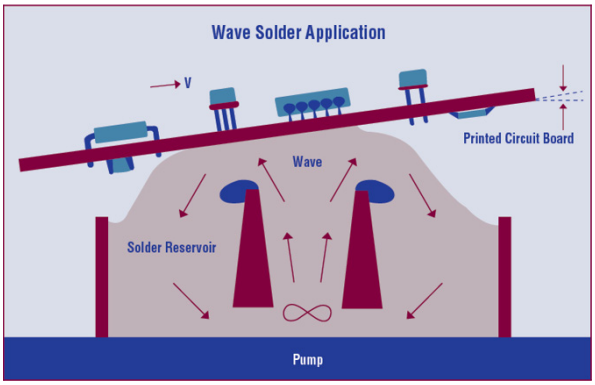
Once the parts (including the PCB panels) are purchased, the automated assembly process can begin. The data required for this phase of the manufacturing process is taken from a few valuable sources. For the manufacturing engineer, the BOM provides a limited view of the requirements. Along with the BOM output, the designer has hopefully provided an intelligent data format file for the design. With this information, the manufacturing engineer can get a good look at the design intent and begin planning the order of manufacturing operations. Invaluable to the process is the provision of X,Y placement data output from the design database.

Sometimes referred to as a "pick & place" file, the data can be fed into process programming to help select the proper tape reel and define the nominal location and rotation for each SMT part which will be automatically soldered onto the board as required.

If there are SMT parts on both sides of the board, the placement file will indicate this and the placement parameters will be divided into two operations (the first pass and the second pass) to place and solder the parts onto their respective sides.

All the remaining through-hole parts which are not to be run in the reflow oven process will be designated for secondary solder processing. This can either be done manually by a soldering technician or by two other common automated processes, wave soldering and selective soldering.

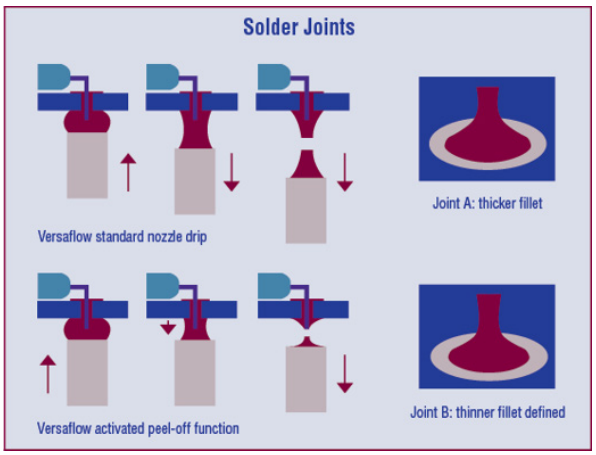
Wave soldering is widely utilized by the EMS provider to make solder joints on the side of the PCB in which the through-hole pins protrude. The best-case condition for wave soldering is when all the parts are mounted on the top side of the PCB. Only the through-hole pins needing solder and the secondary side of the PCB should meet the solder wave.



When SMT exists on both sides of a mixed technology PCB design, it is far more common to process both sides of the PCB SMT passes in the reflow oven. Then follow up with a selective solder operation for the remaining through-hole parts on the assembly.

Selective solder machinery can make accurate solder joints on through-hole component pins by positioning a small fountain of solder underneath the selected pin and automatically raising the fountain cup enough to make the solder joint. The process is highly accurate, but is not considered "speedy" due to the nature of the single-solder fountain head.

Many important decisions must be made regarding solder processing before a board ever begins to roll down the assembly line. Even though the EMS provider has many options to address the manufacturing challenges, the PCB design engineer's goal should be to communicate with the manufacturing engineering stakeholder. Seek out design solutions to reduce these multiple process requirements as much as possible; that's just good DFM.



Automated Placement

After solder paste is applied, the panel moves onto the placement line. The panel is designed to include rails, which not only provide stability to the panel structure, but support the panel on the conveyer as it is moved into position for automated placement. The pick-and place machinery optically calculates the exact position and rotation of the panel. The coordinate starting point is provided by the PCB designer via the placement file, allowing the machine can populate all the components onto the panel with lightning speed.

Moving Through the Reflow Oven

Once all the parts are placed onto the side under assembly, the panel moves along the conveyor into the reflow oven to heat the assembly, melt the solder, and form perfect solder joints. (Ref: IPC-A-610.)

A few variables must coalesce for solder joints to be in spec:
- The footprint land geometry must be designed properly with enough toe and heal and side extension to allow for a perfect solder fillet. (Ref: IPC-7351)
- The optimal amount of solder paste must be applied to the land by accurately calculating the stencil opening in relation to the stencil thickness and type of solder used on the PCB.
- The oven's thermal profile (or heat zones) must be correctly adjusted so the PCB and components receive the proper thermal ramp, soak, and cooling times to allow the solder to flow and solidify properly.

Second Op Assembly Processing

As mentioned, parts not suitable for the reflow oven must be processed using different manufacturing practices. It is usually up to the manufacturing engineer to determine how these remaining components will be soldered. The PCB assembly drawing the designer provides should specify the components be "soldered" without being specific with regards to how. This is intended to allow manufacturing flexibility for the owner of the assembly process.

There are through-hole parts which must be addressed specially with alternative soldering processes such as wave and selective soldering, but there are many other types of assembly actions required on the PCBA after all the parts are soldered. It is important for these second-op conditions to be carefully considered by the PCB designer during the layout phase, hopefully in collaboration with the manufacturing engineering stakeholder. Ultimately, it is up to the design and engineering stakeholders to define the performance requirements of the product. How the manufacturer gets there is ideally left up to them.

Assembly Test

Once all the assembly operations and processes have been run, verification must take place for the PCB assembly to be validated for performance. This is the point where all the front-end investment of time and expense to add test points and create fixtures and software mentioned in our article on design for test is going to pay off. If everything has gone according to plan, the PCBAs are taken from the line and

Assembly Data & Documentation: Process Overview, Part 2

placed into the assembly test fixture. There is a program to check for continuity between parts to ensure all the soldering and placement has gone smoothly. Functionality checks will supply power in order to light LEDs and run displays. The test engineers have collaborated with the PCB design engineer to make certain every function of the PCBA will be tested and perform within specifications.

If a problem is found such as a mis-located part or even an incorrect part, the board can be set aside for rework. After rework, the board will have to run through the entire assembly test process again to ensure complete conformance.

Final Inspection (IPC-A-610)

There are a few assembly notes which may be deemed necessary on an assembly drawing. Adding a simple note: "Acceptance criteria per IPC-A-610," says all the right things to your final inspection department. The pictorial, easy-to-read specification covers conditions of every basic process which your PCBA will have been exposed to during its journey through the assembly shop.

IPC-A-610 and the many other specifications which are utilized to compare manufactured hardware with conforming or non-conforming criteria are invaluable guidelines for final assembly test technicians to measure how each stakeholder has done their job. Once all the solder joints and parts are inspected, and the PCBA is run through its required burn-in testing and has passed, the manufacturing process is complete and the PCBA is ready to be shipped out.

Conclusion

Within the PCB assembly process or any PCB related process, tens of thousands of things can go wrong. The fully engaged stakeholder knows the processes inside-and-out and has learned from each assembly, each cycle, and has made the required adjustments. So must the PCB design engineer. Data and documentation created by the PCB design engineer on the front-end must be clean—garbage in equals garbage out.

Every topic and character description throughout this series illustrates a relative interconnectivity between stakeholder responsibilities and the design manufacturing and assembly processes. It is the hope of all the contributors of this series that its message of collaboration, communication, and understanding will serve the designer well as they assume an influential role as the hub of the entire design and manufacturing process for the PCBA. Learn from your assembly manufacturing colleagues to incorporate the tangible causes of success into your own PCB design process.

For more, download The Hitchhikers Guide to PCB Design.