Using Stencils to Simplify the Printed Circuit Board Assembly Process

Author:
Nolan Johnson
CAD/EDA Manager
njohnson@sunstone.com

The process of creating a prototype circuit board requires multiple phases and a large number of steps. The process starts with the specification of the design, then proceeds through electrical design (often done with a schematic or similar symbolic notation), into the physical design (layout), manufacture of the printed circuit boards themselves, and culminates in the assembly of these boards. Errors can be induced at any one of these steps.

Specification → Electrical Design → Physical Design → Manufacture → Assembly

With the increase in the number of components available only in Surface Mount packages, assembly skills – at least for prototypes – will continue to move away from the realm of hand-assembly by the design team. If you've tried, then you know: it's one thing to hand-solder an 0804 component - that's tough enough. Trying to work a soldering iron in and around anything smaller is difficult indeed. Especially considering that the weight of the component can be so light that a mis-directed exhale would move the components around the board.

And, of course, hand-soldering a Ball Grid Array (BGA) or other similar package is just not feasible. The engineers who blog about soldering leadless packages overwhelmingly are doing nothing more than creating breakout packages with the leadless component so as to facilitate bench testing of the component.

So, like it or not, machine-based assembly is increasingly becomiing a requirement. The trouble is that purchasing pick-and-place machinery can be cost prohibitive. Even if you have the pick-and-place equipment, you still need a stencil to guide the application of solder paste prior to placing the components.

Where is the gap between hand assembly and outsourced assembly? And where does the purchase of a stencil fit in bridging that gap?

The Board Assembly Process:

Apply Solder Paste → Pick-and-Place Components → Heat / reflow solder → Cool → Test
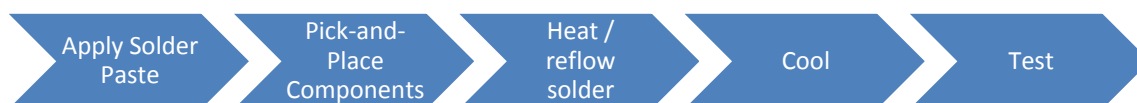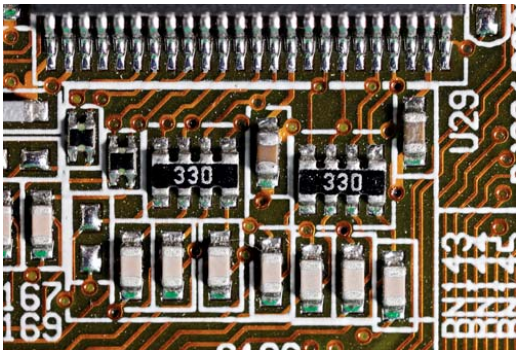
Apply Solder Paste: To start the assembly process, solder paste needs to be added to those areas of the bare, unpopulated PCB board. Solder paste only goes where a component must be in electrically conductive contact with a metal landing pad on the board itself. Typically, this is achieved with a Solder Stencil. Using the solder screen placed directly onto the board and registered in the correct position, a runner is moved across the screen squeezing a small amount of solder paste through the holes in the screen and onto the board. As the solder screen has been generated from the printed circuit board files, it has holes on the positions of the solder pads, and in this way solder is deposited only on the solder pads.

There are two basic types of solder stencils: framed stencils for use with solder paste machinery, and prototype stencils for hand-application of solder paste.

Pick-and-Place Components: Pick-and-Place can occur either by hand, or by machine. With hand pick-and-place, a technician uses tweezers to arrange each component on top of the appropriate bits of solder paste.
As suggested by its name, with machine pick-and-place, a machine loaded with components *picks* the components and *places* them onto the board. The tension of the solder paste is usually enough to keep the components in place. The pick-and-place machine uses location and orientation information (commonly held in a Centroid file) as the reference for placing the components onto the board correctly. Centroid files can usually be derived from the CAD design data for the board design.

Heat/Reflow Solder: Once the components have been added to the board, the next stage of the assembly process is to pass it through the soldering machine. Especially for prototypes, reflow soldering techniques are the most common technique for SMT soldering, though wave soldering machines are sometimes employed.

Test: Functional testing to ensure a working board.

As we can see from this flow, the assembly process starts with the application of the solder paste. This photograph of a board shows how surface mount components can be not only close together but also prone to misalignment or shorting if not precisely soldered.

It's one thing to be able to solder a through-hole component, filling the hole with a bead of solder. It's another thing entirely to tease solder under a chip type component such as the typical SMT package, without having solder or components slide around or solder create unintended short circuits.

The Advantages of Solder Paste Application

The process of pre-applying solder paste to the board, then resting the component upon the solder paste areas, starts to make sense. The likelihood of a quality placement increases, since the solder paste is evenly spread under the component prior to reflow.

The question now, however, is how to apply the solder paste in such a successful way. That's where the stencil comes into play.

What is A Stencil?

Stencils are customized to the board you're assembling. Generally speaking, the stencil is made out of a solid sheet of material (aluminum, copper, and mylar are common). There are generally two ways to obtain your stencil: make one yourself, of have a stencil company make one for you.

Making Stencils Yourself

Limor Fried, the founder of Adafruit Industries, and a highly influential force in the DIY community, has a great page on how to make your own solder paste stencils.

I love how her instructions start....

> "You'll need: 1. A laser cutter."

Now, some of us have access to a laser cutter, but that becomes a pretty big barrier for many of us. Some searching on the internet shows reasonable desktop models for sale in the $5,000 range. I suppose if you're really mechanically inclined, you might consider building your own laser cutter for less. A deeper search for laser cutters you can build yourself uncovers the instruction set that explains how you can, for about $50, build your own laser cutter out of parts. The last item on the parts list? "A bushel of patience."

Further research leads us to an open-source, build-your-own laser cutter kit. Quote from the Lasersaur people, developers of an open-source laser cutter:

> "Laser cutters are traditionally expensive ($30,000 to as much as you can spend) and there are a lot of ... designers... who could do great things with them - if they could afford one, or even get regular access to one. Unfortunately, turnkey systems are expensive, and there isn't really a clear and simple way to build one. We can change this: with roughly six months of R&D time we can develop a laser cutter which anyone can build, use, and maintain. Most importantly this system will be open source which means anyone can improve and modify the design."

So, faced with this sort of an environment for getting your own equipment to help support you in creating your own stencils, the make vs. buy decision takes on another dimension: using a service to provide your stencil as a part of the

manufacturing process.

Buy a Custom Stencil

Faced with the purchase of some pretty serious capital equipment to make stencils in-house, it often pencils out best to use a stencil manufacturer to create stencils as you need them.
Benefits of a custom stencil from a stencil manufacturer:

- Accuracy
- Timeliness
- Reorders

Aside from the differing materials available, stencils come in two basic classes: framed and prototype. Framed stencils are built with a specific style of mounting hardware (the frame) to attach to a solder paste screen printing machine.

Prototype stencils are not mounted to a frame. Instead, three edges of the stencil are typically bent at a right angle to create a "sink" for the solder paste, to stiffen the stencil, and to provide handles for the user. Prototype stencils are optimized for use with pre-release and prototype work. Prototype stencils are:

- Affordable
- Effectively as accurate as production stencils
- Manufactured quickly
- Compact
- Easy to Handle
- Easily disposable / recyclable

When Is A Stencil Cost-Effective?

Granted, there are plenty of times or reasons that an assembly house is your best solution for assembly. And, on the other end of the spectrum, some designs are simple enough, or straight-forward enough (all through hole components), that it's just easiest to pull out the soldering iron and hand-solder the whole lot of components.

But where's that gap? When does it become easiest to make use of prototype stencils?

- A significant number of surface mount components
- Multiple revs planned, but component placement is finalized
- Easier (higher quality) soldering

Let's start by looking at the work flow improvements that come along with using a stencil. In the traditional hand-solder workflow, we can estimate the following times for board assembly:

Pick/place/solder time: 2:20 per part

This is an average. For passives, it'll be less, for large discrete, or connectors, it'll be significantly more. But when you factor in the steps of priming the footprints with solder and an iron, then placing the component with tweezers, then soldering the ends, 2:20 per part is a reasonable estimate. Because the technician is preparing and soldering each component individually, the prep time and reflow time are built into the per-piece time estimate. This changes with a solder stencil, however.

Using a stencil, in contrast, the technician will apply a thin coat of solder paste across the whole board in a swift motion. The time needed to apply solder paste becomes fixed – the same length of time for a larger, complex board as it is for a smaller, simpler board. For estimating, we'll assume it takes 2:30 per board to apply solder paste using the stencil.
The time it takes to place the components onto the PCB with tweezers is probably about the same, except that you'll be placing them all at once, in a big batch when you use the stencil. This part of the estimate scales up and down with the number of parts you need to place, but an estimate of 20 seconds per part to pick, place and nudge the part onto the proper footprint seems reasonable. Remember, this 20 seconds is built in to the 2:20 hand-soldering estimate.
Finally, the stencil technique means you can use a reflow oven process to heat the solder paste and attach all components at one time. Again, this is a fixed-time step. Because you can be assembling the next board while you reflow the previous board, I left the reflow time out of the estimate.

Finally, let's assume it costs your team $25/hr in labor for assembly. And, let's assume that you spent $120 to purchase the stencil.
The numbers are quite telling – it only takes a small handful of parts, on even a single board, before the stencil starts paying for itself – first in time saved, then in overall quality of work and increased yields from assembly. Let's look at the time-saved analysis first:

50 parts/3 boards	$120 saved
50 parts/ 3 boards	$242 saved
10 parts/ 2 boards	$14 saved
10 parts / 10 boards	$70 saved
80 parts / 5 boards	$320 saved
80 parts / 2 boards	$130 saved

In addition, if the cost of your bare PCB and components exceeds $100, all it takes is one mis-assembled board to equal the cost of the prototype stencil. The secondary savings in increased yield can often save your project dollars even if the time and labor are comparable.

Conclusion

Sunstone Circuits® has added stencils to our product line to help make your in-house board assembly process faster, easier, and more accurate. The use of stencils will only become even more commonplace with the migration to SMT style packaging for an ever increasing number of components.

Contact information:
Molly O'Hara
Marketing Media Manager
Sunstone Circuits
503.759.2196