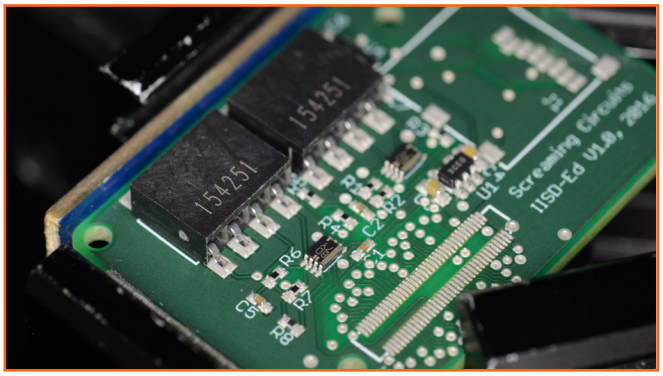
PCB Assembly for Smarties



How best practices at every stage of the process help you avoid common pitfalls and ensure success.

Our For Smarties series concludes with the fun part—PCB assembly. It is time to stuff the bare board with components and put it to work. You've successfully navigated the process from board design through manufacturing, but even experienced makers can hit a snag at assembly.

This paper will offer strategies for ensuring a smooth assembly process that produces a quality product.

Clearly each of these steps leading up to assembly will impact that process. As you read PCB Assembly for Smarties, keep in mind some of the tips and best practices put forth describe measures that take place during one of these earlier phases, well before the components are attached to the board.

The journey so far...

For Smarties was originally aimed at makers, electrical engineers, and even rocket scientists without ready access to a PCB designer. As the popularity of the series grew, we embraced a broader audience and expanded our topical focus.

PCB Design for Smarties—Aimed at technical professionals who become their own PCB designer by circumstance, we share our expertise to help you be more confident there will be smooth sailing from design through manufacturing. Download now »

PCB Prototyping for Smarties—The second installment of the series identifies the vital need for rapid prototyping. This valuable step in the process validates manufacturability of design, helps avoid expensive mistakes, and creates opportunities to improve or innovate. Get the paper »

PCB Manufacturing for Smarties—With an emphasis on DFM, we demonstrate best practices for choosing design tools, outline an effective process, and offer tips for ensuring smooth production. Read more »

The Importance of Being Prepared for Assembly

Pretty much every electronic device needs functional components for power regulation, I/O interfaces, and processing. When we connect these components with the wiring of your PCB, the result is a printed circuit assembly (PCA). At this stage of the game, the PCA needs to do its job for the end product.

In terms of time, money, and reputation, the cost of a do-over after assembly is more than at the design or prototyping phases—especially at higher volumes. A PCA with performance problems at this point could result in delayed order fulfillment, project cost overruns, or a competitor beating your product to market.

That's why choosing your assembler wisely is so important. A reliable assembly partner will help you avoid costly failures associated with slipshod production, as well as defective and improperly installed components.

The Basics

There are also steps you can take to ensure a successful assembly.

First and foremost, collaborate with your assembly partner early and often, the same as you do your PCB manufacturer. Your assembler can be a resource that offers tips for effective board design, provides advice on which materials to use, and gives you input on layout techniques that will ensure component fit and functionality in the PCA.

Also keep in mind these best practices, many of which we discussed previously in the **For Smarties series** of white papers:

- Identify budget limitations and lead times early, so neither your manufacturer or assembler are put in a bind later in the process.
- Make full use of all design tools at your disposal and consult your assembly or manufacturing support team when necessary.
- Don't skip the design rule check or DFM review.
- Be sure you've matched PCB and PCA materials to the environment in which the finished product will operate.
- Double and triple check your parts lists, being sure to label consistently and correctly throughout.

Weigh the risks of offshoring carefully

The allure of low-cost overseas assembly can be tempting, especially on a tight production budget. Before taking the swim across the pond, consider possible risk factors, specifically use of substandard or imitation parts and potential supply chain disruption.

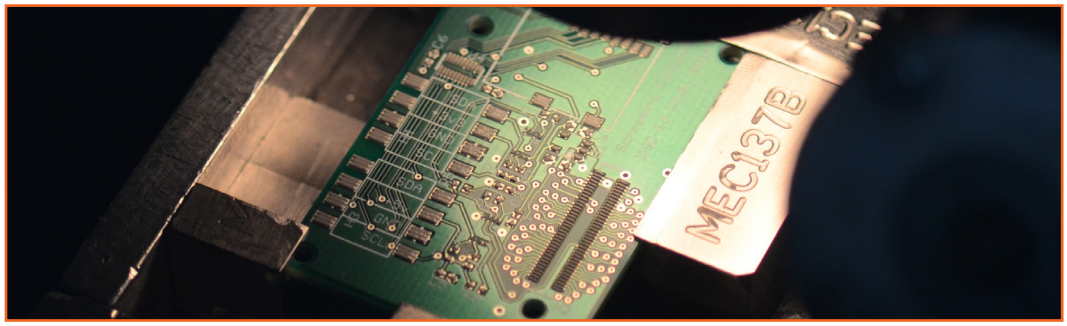
Parts not meeting specifications or industry standards can cause board malfunctions or failures—more than offsetting the perceived cost savings, and potentially harming customer relationships. And a lot can happen on a 5,000-mile transpacific journey, so carefully consider if the possible cost savings are worth risking a delayed product launch or customer delivery.

PCBs come in three basic forms: **single-sided, double-sided, and multi-layered**. Each has its own guidelines for component placement, but in general, parts should be placed on the top side of the board. To minimize trace lengths and help prevent short circuiting, put components that connect next to each other—including switch connectors, LED, mounting holes, and heat sinks. Integrated circuits should all be placed in one position, facing up, down, left, or right.

When you're done, overlay a copy of your layout on the PCB to ensure there is adequate space for each component. In this process, confirm that your signal traces are as short and direct as possible, using vias on multi-layer boards to move signals from layer to layer.

You also must choose between **hand and machine assembly**. If you're producing thousands of simple assemblies, the choice is obvious, but hand soldering the PCA makes sense in many cases. Makers and those developing a new product often choose the human touch. It depends on the complexity of the board, production volume requirements, and materials needs.

Once you move past the development phase or production volume increases, you'll likely want to transition from hand soldering to machine assembly.



Time out to discuss stencils

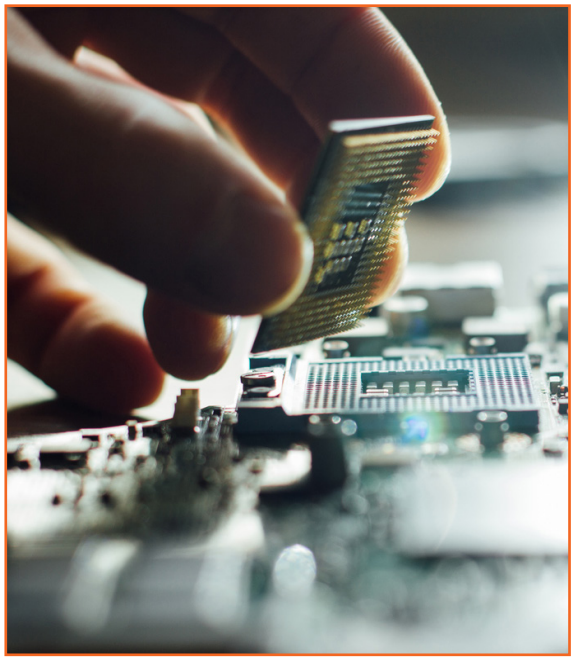
To ensure quality and save time, use a surface mount technology (SMT) stencil to transfer solder paste to the bare circuit board. That is the sole purpose of an SMT stencil. Using a stencil customized for your board is superior to relying on hand soldering methods. The stencil will help you avoid solder bridging, lifted components, excess solder, and solder balling—all of which can compromise the functionality and durability of your board.

You have two choices when it comes to stencils for your board: buy it or build it yourself. Choosing the latter requires a laser cutter so, if you don't have one of those handy, use a stencil manufacturer to create them for you. The benefits include:

- **Accuracy**
- **Speed**
- **Consistency**
- **Affordability**

Use of stencils will simplify and accelerate the assembly process cost-effectively.

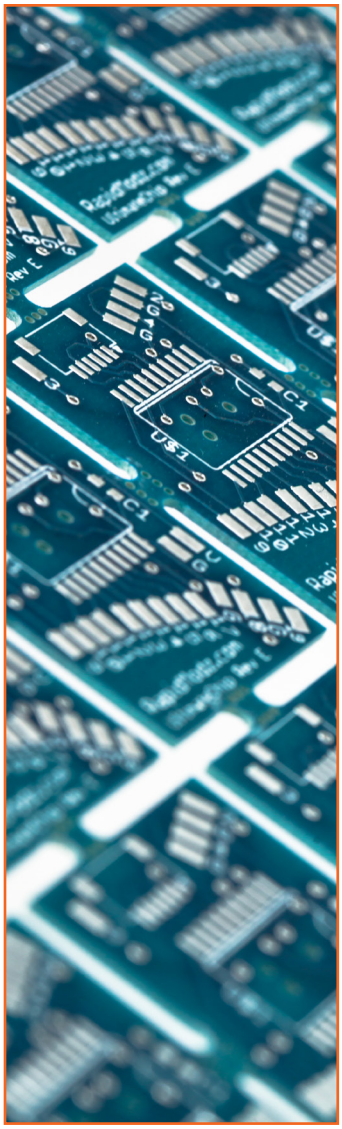
Five Pitfalls to Avoid in the Transition from Hand to Machine Assembly.



What works great when PCAs are soldered by hand may not work at all during the machine assembly process. Here are solutions to five common pitfalls that can happen when moving from hand to robotic assembly.

Protect your moisture-sensitive parts.

Though it may seem otherwise, plastic does indeed absorb moisture without being immersed in water. Plastic boards will absorb humidity if not protected from it. If plastic parts that have absorbed moisture are, for example, placed in a reflow oven, the H2O turns to steam. The expanding steam has the potential of splitting the board. If that damage isn't visible to the naked eye, the result will be an unreliable product in the field.



If you are going to send your project off to be machine assembled, you can do two things with moisture sensitive parts:

Order the parts on a just-in-time (JIT) basis and keep the packages sealed.

If you have parts that have been exposed to the air, inform your assembly partner that this has happened and ask that they be baked prior to assembly in order to remove the moisture safely.

Don't skimp on solder mask.

Some board fabrication houses offer reduced prices if you order your boards without soldermask or silkscreen. That's not a problem when you're hand building—you can regulate the amount of solder by eyeball.

However, when a stencil is used to apply solder paste and the board is run through a reflow oven, the solder will spread back on the exposed copper traces. This may leave your parts without enough solder on the pins to create a reliable connection.

Solder mask may add a bit of cost up front but will increase reliability and reduce cost in the long run. Creative choice of solder mask color can also add some personality to your boards.

Silkscreen can improve accuracy.

Without silkscreen, machine assembly won't be as reliably accurate. Unfortunately, CAD files do not tell the assembly machines exactly where each part is supposed to go and what angle and orientation is required. Footprint errors are common, as are components with ambiguous marking. Clear silkscreen will help to ensure that any errors in the data are caught visually.

There is no need to fear surface mount.

One of the easiest ways to ensure that a board can be hand-built is to stick with thru-hole parts. But doing so puts many limits on your design, rules out a lot of new technologies, and are cost prohibitive in larger production runs.

Breakout boards can be used for small surface mount chips and pre-mounted on a PCB with hand solderable headers. These are available for a lot of new parts, but not all. They also take up a lot of extra real estate on the board and are costly. If you are hand building a prototype, or a small number of boards for your own use, breakout boards are fine.

For higher volume production runs where machine assembly is a must, re-layout your PC board to use the surface mount chip without the breakout board. Just don't forget the bypass capacitors or any other required support components. This layout tweak should be straightforward. Many breakout boards are open source, so you may be able to use its proven schematic and layout for that part of your design.

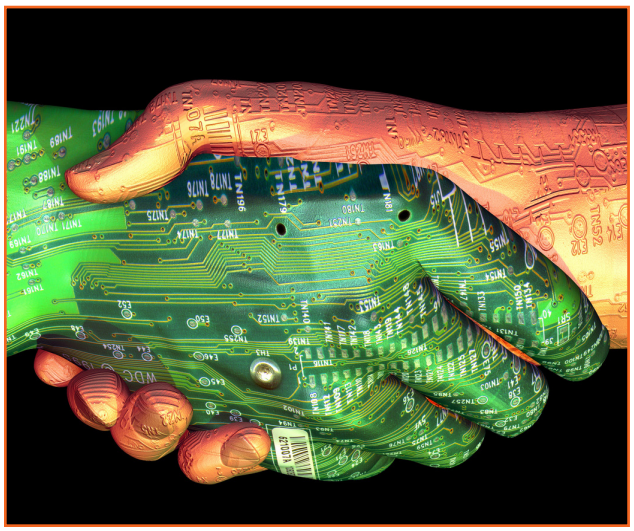
Do not use open vias in machine assembly.

Quad flat no-leads (QFNs) and ball grid arrays (BGAs) have pins/pads under the part but are often completely inaccessible. To deal with this, a common hand-soldering practice is to put large vias in the pad. Affix the part to the PCB with tape, turn the board over and using a small tipped soldering iron, stick solder through the via. This allows you to hand solder almost any leadless surface mount part.

This process is not possible in an automated assembly. The solder will flow down the via and end up on the back side of the board. You may end up with shorts on the back side, parts that fall off the front side, or parts that just don't connect with all of their pads.

If you have been using the open via hand solder technique, you'll need to re-layout your PC board without any open vias in the pads before sending it out for manufacture.

Conclusion



Now that you've read all four of our For Smarties papers, it's clear we believe open communication with production partners, as well as paying close attention to detail, are key to a smooth production process and a quality end result. If this is your first read in the series, we encourage you to read through **the whole series**. Each installment offers ways to avoid pitfalls and fix problems before they arise on the production line or after the PCB is in use.

To learn more about PCB assembly, visit **Screaming Circuits** and explore their **resource center**.

For more tips on PCB design and prototyping, check out our white papers **here**. And our customer success **stories** offer real-world case examples of solving problems before they occur.