

Solder Paste Printing – the Most Critical Part of Building a Quality PCB Explained

If someone were to ask what the most important part of their favorite electronic device is, they are likely to point to the printed circuit board (PCB) responsible for making the device work before moving on to other components. From large motherboards to tiny chips, most of the technology that retail consumers take for granted comes down to a properly installed and calibrated PCB.

While there are a great deal of steps in the PCB assembly process, and each is extreme importance, one stands out among all the others as being the most critical for ensuring lasting success. Incorporating surface-mounted devices (SMDs) onto PCBs is the most critical part of a building a quality electronics product, and for good reason.

Attaching Devices to PCBs – Solder Paste Printing

One of the most popular and reliable ways to attach SMDs to boards is using a process called solder paste printing. Essentially, a screen printing device uses a stencil of the finished boards components to apply solder paste to the board with extreme precision.

Solder paste is exactly what it sounds like – a [blended mix](#) of flux medium and graded particles of solder powder. It serves as the attachment medium between the device's interconnection sites and the PCB itself.

After a PCB has solder paste accurately layered over it, PCB assemblers can *reflow* the solder paste – melt it at a specific temperature – to form a tight, conductive bond.

In many ways, this process is similar to screen printing graphics or designs onto textiles. The main difference is the level of precision required – both the amount and consistency of the solder paste must adhere to strictly observed standards in order to create a durable bond.

Facing Challenges with Solder Paste Printing PCBs

There are many challenges to consistently solder paste printing PCBs. Less-than-professional PCB assembly companies will often introduce errors during this delicate process – especially when rushing to complete large orders with tight deadlines.

In order for the process to be successful, PCB assemblers must respect several principles:

- **Create a Perfect Stencil:** The accuracy of the process depends on the accuracy of the stencil. A shoddy, imprecise, or poorly printed stencil will never produce reliable results.
- **Program the Screen Printer Properly:** Solder paste screen printing devices need exact measurements and calibration to perform their work with precision. Expert screen printing programming is a valuable skill in the PCB industry.
- **Apply the Right Amount of Solder Paste:** Too much solder paste will create an imperfect outline, while a lack of solder paste will prevent the SMDs from sticking.
- **Choose the Right Process:** There are two main methods to achieving a good print – mesh screen printing and metal screen printing. While the former is [the cheapest method](#), it does not work on small SMDs that require precision attachment.

By paying close attention the solder paste printing process, you can accurately gauge just how capable a PCB assembly company truly is. Think of this process as the tell-tale sign of a professional approach.