

H The difference between THT and SMT

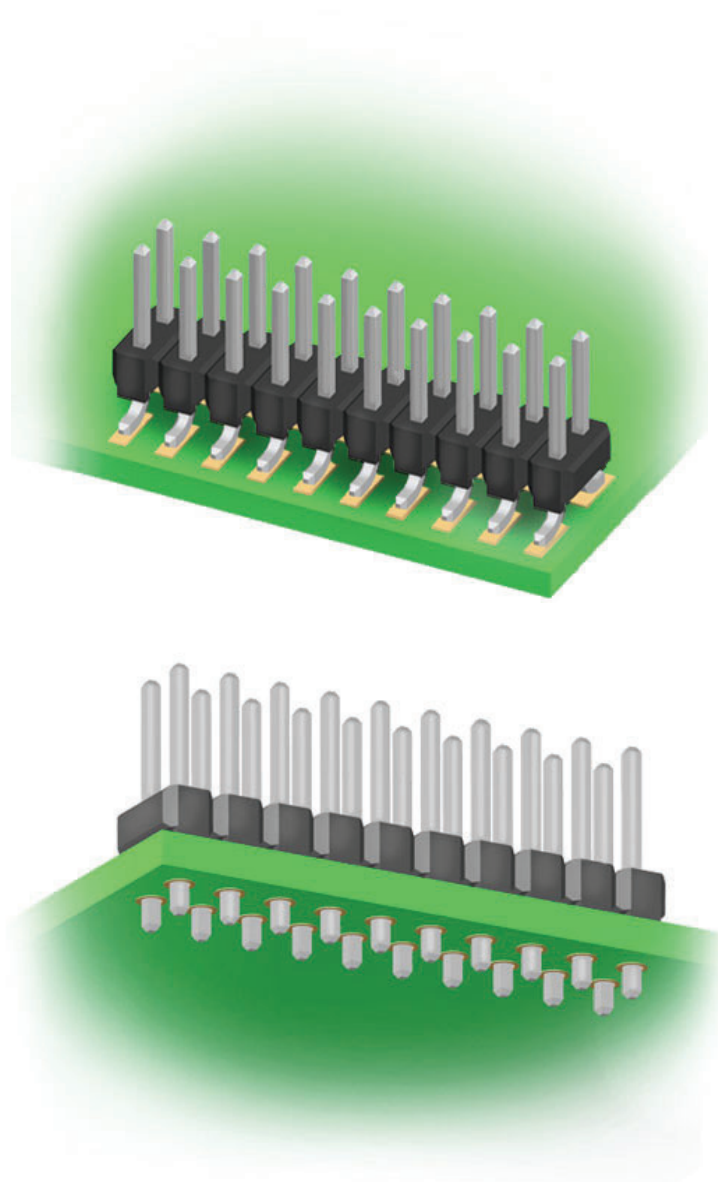
Traditionally, through hole technology (THT) was used to construct the vast majority of PCBs. Over recent years, however, the use of surface mounted technology (SMT) has grown in popularity and is increasingly used in place of through hole technology.

What is Through Hole Technology?

Through hole technology involves inserting components with tails or leads into holes drilled in the PCB. These components will be described as throughboard components. The leads can then be soldered onto pads or lands on the underside of the board, usually with a wave solder process (but also by hand).

A recent development of this process has been to change from a plain drilled hole to a plated through hole. Solder paste is applied within the hole, and the lead is pushed through this paste. The whole PCB is then heated to reflow this solder paste – this is called pin-in-paste soldering. This development allows for boards that are a mixture of through hole and surface mount, as both component types can be soldered in a single process.

Through hole mounting is extremely reliable as it creates strong mechanical bonds, and is a well-established process. The number of variables that may cause solder issues is less than with surface mount, and are generally well researched and understood. However, due to the additional drilling and possible requirement to print on both sides of the PCB, the process can make the bare board more expensive. It can also be more difficult to automate the placement of the components on the PCB, as many throughboard components are packaged loose or other bulk methods.



What is Surface Mounted Technology and how does it work?

Surface Mount Technology is a method that is used to assemble electronic circuits, where the components are mounted or placed directly onto the top surface of printed circuit boards.

Surface Mount devices (SMD) will have flat co-planar tails or leads that allow the component to rest on a flat exposed track on the PCB. No holes in the PCB are needed, and solder paste is applied through a stencil to cover the exposed areas. The components are then placed (often by machine) into the solder paste, and then the PCB is heated to reflow the paste.

Without the need for holes, SMT components can sometimes be smaller than through hole ones, as they use smaller leads or

contact pads instead of leads. This can allow the PCBs to be smaller and more compact, with a higher circuit density – or at least a cheaper design without holes and only circuit printing on one side of the board.

Although the PCBs for surface mounted devices tends to be less expensive than through hole mounting, the capital investment needed for machinery is often higher. Also, the level of design, production, skill, and technology required for SMT is typically more advanced than that needed for through hole mounting. However, this is then offset by the higher throughput capable with a fully automated setup, and investment is paid back through faster production.

Conclusion

Surface Mount technology is likely to be the system of choice for any manufacturer operating with high volumes of product – lower volume manufacturers can choose the most suitable system for their specific PCB requirements. Some devices may only be available in one style or the other, so mixed boards may be an inevitable outcome, and PCB designers need to consider this in their layouts.

Choose between THT and SMT using the Product filters on Hi-Rel, Durable Industrial and PCB Connectors.

