

H 5 Considerations for PCB Layouts

Designing compact and efficient printed circuit boards (PCBs) is at the heart of the modern electronics industry. Our homes and lives are filled with electronic devices, from smart fridges and wearable devices to cars and passenger transport. As each year passes, consumers are expecting more functionality, and it is the job of PCB designers to meet these expectations.

Just as consumers demand higher performance, there is also a desire for devices to be smaller and more convenient. As PCB-mounting components become more compact, designers are faced with new challenges. In order to help you, Harwin has highlighted 5 considerations that will help you avoid potential problems that might arise.

1 - Choose the Right Solder Mask Clearance

We need to use solder mask to protect our PCB from corrosion and contamination. In theory, we should use solder mask in all areas where we are not expecting solder to be placed. In practice, this means preventing solder mask from spreading into unwanted places.

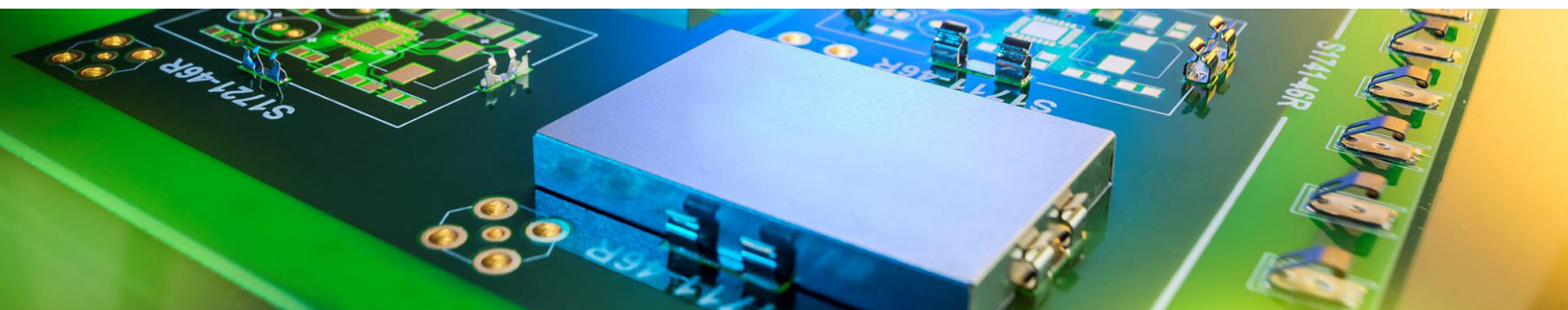
Solder mask is a viscous material that can continue to spread after application. If solder mask is allowed to contaminate the solder pads, it can prevent proper solder joints being formed. Therefore, designs should leave a small gap between the outer edge of the solder pads and the solder mask. This is known as the Solder Mask Clearance. This clearance can vary depending on how the mask is applied and is best defined after discussion with the PCB manufacturer.

2 - Solder Mask Webs, Solder Mask Dams

Solder mask width becomes critical when we consider the space between solder pads for components such as connectors. The distance between the terminals of a component is called the pitch. As electronic components have become smaller, so too has the distance between their terminals. It is common to find connectors with pitches of 1.00mm and below (and other components at even smaller pitches), and this presents additional challenges for us as PCB designers.

When fixing fine-pitch connectors to a PCB, the distances between the solder pads need to be considered carefully. We have already mentioned that we need to allow clearance between the edge of the pad and the solder mask. If two component pads are sufficiently close together, allowing clearance on both sides may not provide enough surface area for the solder mask to adhere properly to the PCB between these pads. Under these conditions, thin slivers of solder mask can lift and contaminate the solder joint, affecting solder wetting and creating poor solder joints that may not be visible to inspection.

Therefore, when designing the solder pads for fine-pitch connectors, it is important that we discuss the issue with our PCB manufacturer. They may also raise this as an issue on any PCB when they have completed their Design for Manufacture (DFM) review. The manufacturer can advise options on how to proceed, which may include removing the solder mask web between pads.



3 - Is Through-Hole the solution for you?

The advent of Surface Mount Technology (SMT) was one of the greatest innovations in PCB design. Surface mounting components has made manufacturing quicker and easier to automate, increasing factory throughput and capacity. However, for some applications, surface mounting might not be the right solution.

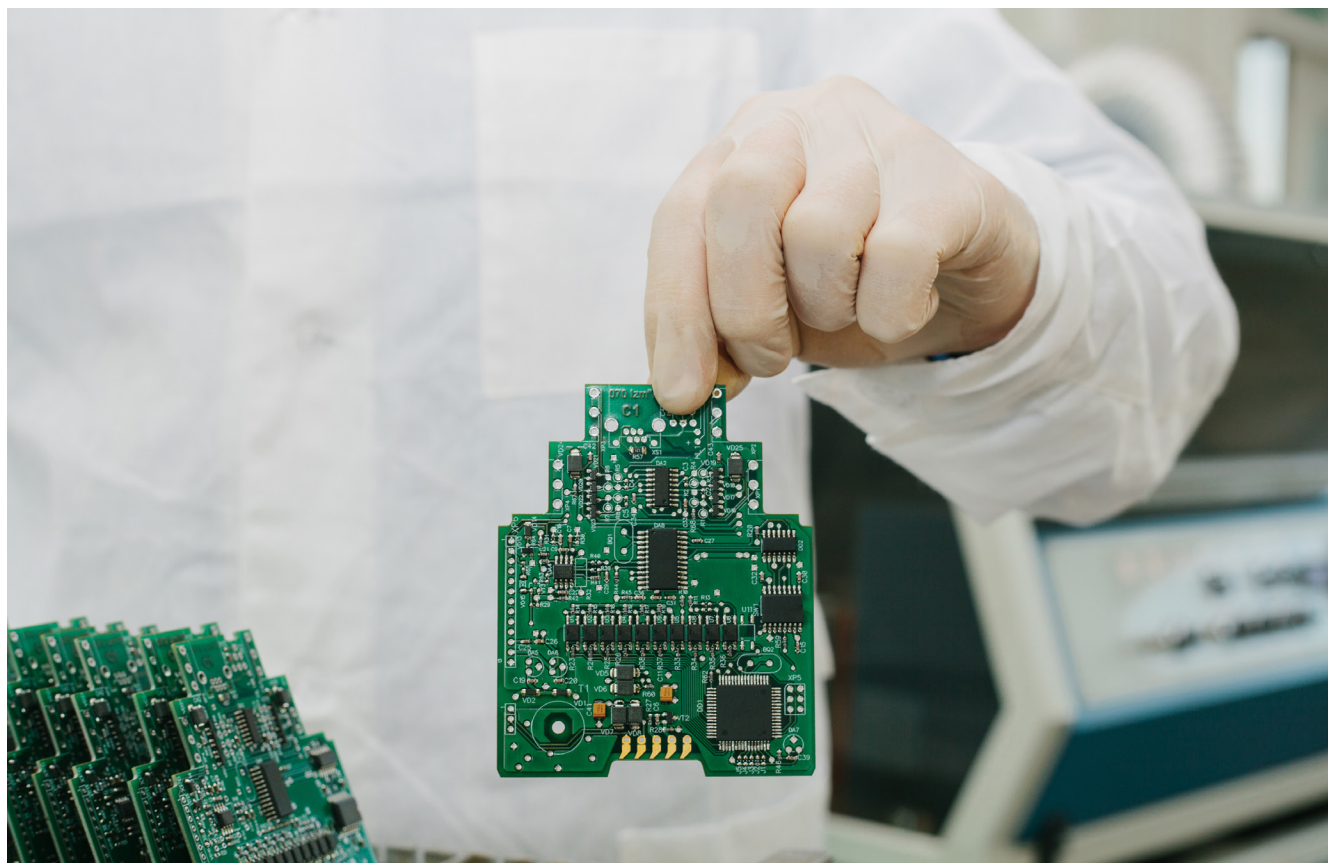
Selecting through-hole terminations may seem like a step backwards, but there are times when the physical benefits make them ideal. A copper pad for an SMT component can be tiny, and is fixed to the PCB over a small area. Electronic devices that are subjected to stresses during their life, be that vibration, shock or tension, need additional mechanical strength. In this case, a terminal passing through the PCB will provide robustness and stability that SMT techniques cannot.

4 - Don't Forget the Connector

It is important that we do not concentrate so completely on the solder pads that we forget to allow room for the connector. The placement of the connector on the PCB is crucial to allow space for the body of the connector to avoid conflicts with other components. This is especially important for modern, compact designs where board space is at a premium.

There are additional requirements when using right-angled connectors. Connectors that are designed for fixing to the edge of a PCB are often used in backplane or panel mount applications, and their placement is critical to the correct spacing of the boards themselves. When placing them in our designs, we must ensure that we pay close attention to the dimensions published by the manufacturer.

Datasheets and PCB schematics will provide this data, but the use of CAD models will save time and prevent errors. We should always check the manufacturer's website to make sure that we are using the most up to date information.



5 - Prototype versus Production

There comes a time when the design phase is over, and our products are ready to be released. Our concept has been proven, and our task becomes one of turning the design into a production-ready device. The design that was optimized for testing and prototyping might not be suitable for mass production.

We need to decide whether the design features that were necessary during testing (such as test points) are really needed when our device is released onto the market. It might not be possible to retain design tolerances used on our prototype PCB using cheaper, high-volume manufacturing techniques. Working with the production PCB manufacturer as the design comes closer to production is essential. This will help avoid delays and ensure our PCB is designed for high-volume manufacturing and its associated cost benefits.

Understanding and Planning are Key

The small size of modern electronic components makes PCB design challenging. However, understanding the potential pitfalls and planning our designs accordingly can prevent problems before they occur. By following the rules of PCB design and choosing partners who can provide all the necessary information, it is possible to create elegant and compact designs that will serve customers for years to come.

If you would like to talk to someone about Harwin's product, technical knowledge or documentation, contact one of our Experts for assistance.

**If you need more help designing your PCB layouts or choosing
our connectors, our Experts are waiting to advise you.**