Introduction

Merifix test fixtures are designed to be compatible with a wide variety of boards, with no special requirements placed on the board design. But if your PCB is still at the design stage there are various things you can do that will make things easier.

Tooling Holes

Tooling holes are added to boards for a variety of reasons.

PCB fabricators use them for aligning the board during drilling and routing. Assemblers use them for alignment with stencils during solder paste printing, and for location on pick-and-place machines. They are used for alignment on electrical test fixtures. They may also be used as alignment holes in the final enclosure, often mating with small moulded plastic posts.

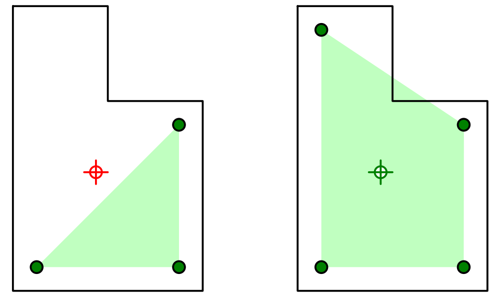
For some of these purposes tooling holes can be placed in the overall panel, but for activities that occur after de-panelling, or for boards that are not panelised, tooling holes are required within the board outline.

Tooling holes are often standard sizes, for compatibility with commonly used locating pins. Common imperial sizes are 0.062", 0.093" and 0.125". Common metric sizes are 1.0mm, 1.5mm, 2.0mm, 2.5mm and 3.0mm.

Tooling holes are always un-plated. This gives better accuracy since plating thickness is not well controlled.

For locating boards on Merifix test fixtures a good choice is four 0.062" (or 1.5mm) tooling holes positioned towards the outer corners of the board.

Putting the holes near the outer edges of the board helps ensure that the board will be stable when it is supported on the locating probes before the fixture closes. The centre of gravity of the board should be inside the polygon described by the support locations.

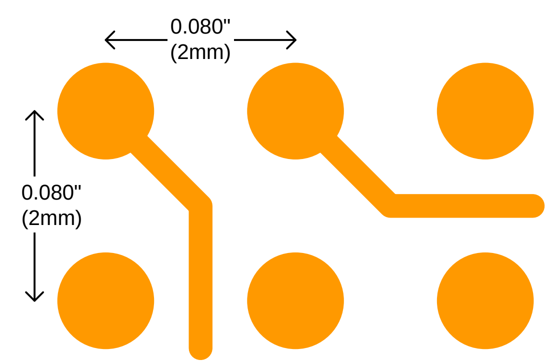


Keeping the largest possible distance between the holes maximises alignment accuracy.

Test Points

For compatibility with the popular and widely-available S-075 style probes, test points should be no closer than 0.075" centre-to-centre.

A straightforward approach is just to place test points on a 0.080" (or 2mm) grid. This has the advantage of requiring little thought and is easy to check.



A 0.080" grid is typically more convenient than a 0.075" grid for tracking reasons, as discussed below.

Where space is very tight you may need to place test points in arbitrary locations (not on any grid). This has the disadvantage that each location must be carefully checked for the proper clearance to all neighbouring locations, which can be a tedious and error prone task.

Through-Hole

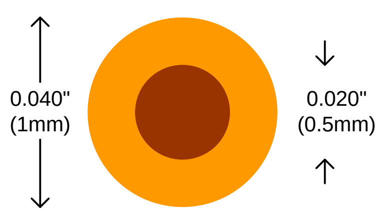
Through-hole test points are advantageous, even if you only intend to probe from one side, because the holes in the pads help the test probes to self-centre. This increases the effective pointing accuracy and results in a more reliable fixture.

A secondary benefit is that through-hole test points are accessible from both sides, which can help when debugging. For example, with the board on the fixture being probed from below it may still be possible to get a scope probe on to the test points from above.

A final advantage of through-hole test points is that the holes make it easy to drill the test probe locations in the fixture using the recommended method of using a bare board as a drilling guide.

The disadvantage of through-hole test points of course is that they take up space on both sides of the board.

If you have room on your board a good design for through-hole test points is a 0.040" (or 1mm) pad with a 0.020" (or 0.5mm) hole.



The hole can be a little smaller than suggested if desired - perhaps to make it the same size as via holes on the board. But don't make it too small. The smaller it is the less effective it will be at helping slightly misaligned probes to self-centre.

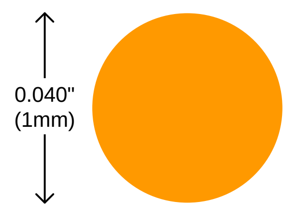
Typically the opening in the solder mask is made a little larger than the copper pad, either directly in the CAD files or by instruction on the fabrication drawing. This ensures that the effective pad area is not reduced if the solder mask is slightly misaligned.

SMT

Single-sided (SMT) test points have the big advantage of only occupying space on the bottom side of the board. There is no pad using up space on the top side and no through-hole using up space on inner layers.

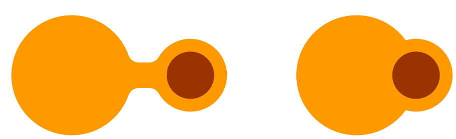
This advantage may not be as big as it seems if the components that the test points connect to are on the top side of the board, since the connecting tracks will require vias. Nevertheless, vias are usually smaller than test point holes and there is more flexibility in their location.

A good design for SMT test points that will be placed on a 0.080" (or 2mm) grid is a 0.040" (or 1mm) pad.



Again, the opening in the solder mask is typically made a little larger than the copper pad.

If you are going to use single-sided test points without holes, ensure that they really are without holes. Do not place vias so close that the via hole encroaches into the test point pad.



Acceptable Risky

This creates the danger of test probes slipping into the via hole and being bent off-centre, which will reduce pointing accuracy and can cause probes to jam.

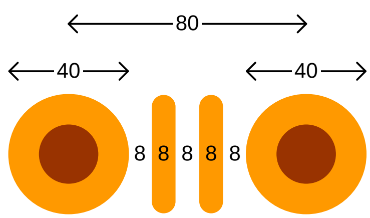
Silkscreen

There is no great advantage to having a silkscreened outline around the test points. It can make them easier to see. However, the silkscreened legend layer typically has the poorest registration accuracy of all layers of the board. There is a risk of the epoxy ink encroaching on to the test point pads if the outline is not much bigger than the pad and the registration is poor.

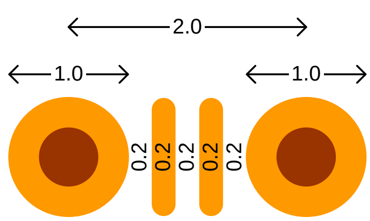
If you are going to add a silkscreened outline around the test points, ensure that the gap between the inner edge of the silkscreened line and the outer edge of the test point pad is comfortably greater than the worst-case silkscreen registration accuracy.

Tracking

The suggested 0.040" test points on a 0.080" grid allows two traces between test points using common 0.008" trace/space rules, routing on a 0.008" grid.



Similarly, 1mm test points on a 2mm grid allows two traces between test points using common 0.2mm trace/space rules, routing on a 0.1mm or 0.2mm grid.



If routing density between test points is not crucial the pads may be made a little larger if desired. This relaxes the pointing accuracy required of the test probes and can make for a more reliable fixture. Pads can be enlarged up to 0.056" or 1.4mm and still leave space to get a single trace between adjacent test points.

Locations

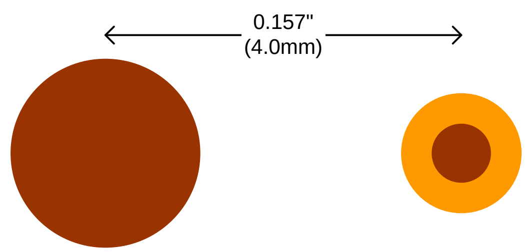
The locations of the test points will largely be dictated by the constraints of the circuit and the layout, but there are some general principles that will help minimise fixturing difficulty.

Try to spread the test points out over the surface of the board rather than concentrating them together in one small area. Lots of probes together will exert a large force concentrated in that spot, which can cause the board to bend.

Similarly, spreading the test point locations evenly across the board helps to prevent the board from twisting or tilting as the fixture is closed.

Keep the test points away from the extreme edges of the board.

Keep test points away from tooling holes that will be used for locating the board. For Merifix fixtures, a minimum centre-to-centre distance of 0.160" (4.0mm) is recommended.



If there are limited places on the top side of the board where hold-down posts can land (areas free of components), avoid placing test points on the bottom side directly under those locations. On the fixture you don't want hold-down posts directly opposite test probes, because the hold-down posts can hit the test probes if the fixture is closed with no board in place.

Summary

Compatibility with test fixtures will be maximised by following these straightforward guidelines when designing your PCB.

The most important recommendation is the one concerning the minimum test point spacing. If mechanical or electrical constraints mean you can't follow one or more of the other guidelines precisely, don't worry. Fixtures are flexible and most issues can be worked around. You should have little difficulty using Merifix or similar test fixtures.

Merifix Electronics
North Vancouver BC Canada.
www.merifix.com
sales@merifix.com