18401 North 25th Avenue, Suite 130, Phoenix, AZ 85023
Phone: 602-843-8565 Fax: 602-843-8766 www.uni-fix.com

PCB DESIGN FOR TESTABILITY GUIDELINES
SMT PRODUCTS

Interfacing an SMT product for production test is a difficult task at best. Therefore, any assistance to aid in the testability of the product which may be incorporated into the PCB design will not only reduce the cost of both the test fixture and test fixture maintenance, but will also greatly improve the reliability and repeatability of the test operation.

Although the following information is primarily aimed at test fixturing for assembled SMT products, it will also prove useful in reducing the costs and repeatability problems associated with bare board test.

I. Design for Testability

a) Whenever possible, access to all SMD nets should be provided at dedicated test pads, preferably on the solder side of the unit under test (UUT), or at specified via holes. It is also possible to provide a "keyhole" type pad that allows access to the net on the end of the SMD pad. Keeping the test pads on the solder side of the UUT (or on the least populated side with the lowest component profile) allows conventional “single side access” test fixtures to be used. This reduces initial cost, as well as maintenance, and provides access for scratch probing during program debug. In addition, the reliability and repeatability of the test fixture is greatly enhanced. All test pads should be no closer than .075" from any device. This reduces the risk of the head of the test probe shorting to a component. If solder side components are greater than .325" tall and a standard vacuum test fixture is planned to be used, allow a minimum distance of .250" (from test pad to device) for milling "pockets" for the component in the test fixture's base plate.

b) Test pads should be maintained on minimum centers of .086". This allows the use of lower cost, more reliable .100" test probes. The incorporation of minimum .066" centers for test pads results in the use of .075" test probes. These test probes possess the same reliability and repeatability as the .100" probes but cost approximately 25% more per test point. Test pad spacing of minimum .046" centers forces the use of .050" probes. These probes are much more fragile than their .075" of .100" counterparts. Use of these probes effects the reliability of the fixture in terms of fixture life, and the size of the probe results in a lighter contact spring force which may effect contact repeatability, especially on PCBs using "no clean" flux processes. The cost factor of .050" test probes should also be considered. The initial price delta is approximately 60% greater than .100" probes and the life expectancy is approximately 50% of either the .100" or .075" test probes. Mixing of test probe center-to-center distances is acceptable.

c) To insure targeting accuracy during test, pad size should be a minimum of .035" (+.002"/-.000" diameter). With pads of this size, there is a total targeting tolerance of +/- .0165".

d) For tooling of the product, it is recommended that there be at least 3 non-plated tooling holes which are minimum of .125" (+.003"/-.000") in diameter. At least 2 of these holes should be placed on a diagonal at opposite corners of the board. The cross diagonal tolerance should be no more than .003". If the PCB is .087" thick or more, the tooling holes can be placed .125" from the board edge. If the PCB is less than .087" thick, these holes should be at least .200" in from any edge. The tooling hole size determines whether the test fixture's PCB alignment pins can be placed on the same platen as the spring probes. If the holes are too small, the alignment pins must be placed in the top moving platen on a vacuum actuated test fixture instead of the base plate which holds the spring probes. This means the product alignment gains an additional targeting tolerance of .003" in a standard vacuum test fixture. If the tooling holes are too small, a more costly mechanical or pneumatic actuated fixture may be required, depending upon the overall manufacturing tolerances of the PCB.

e) Front-to-Back layer registration should have a maximum tolerance of +/- .005".

f) Pad size tolerance should be +/-.001 and pad location tolerance should be +/- .002".

g) Fill all vias with solder or soldermask. For vacuum actuated fixtures, this allows a solid seal of the UUT. Soldermask on the top side of the PCB should not cover the vias used as test pads on the bottom side of the PCB. This allows for a more uniform fill of the solder in the vias designated as test pads. In addition, soldermask on the top side may "bleed" into the via. If the via is used for test access, the soldermask may "bleed" far enough into the hole to cause the probe tip to engage it and prevent the probe from contacting the sides of the plated hole.\

h) Resistive coatings that prevent solder from attaching to any pads, vias or holes used for test access should not be used. If the coating prevents solder adherence, it will most likely prevent good, repeatable probe contact.

i) Total fixture fabrication tolerances (including drill, load and probe radial play tolerances) will be .0135".

The total targeting tolerance of a PCB deigned with these criteria in mind will be .017". The fixture fabrication tolerances of .0135" (average fixture builder tolerance) added to the board tolerance is .0305". If the minimum pad size (including tolerance) is .035", the worst case target has the probe tip .0005" from the pad edge.

It should always be considered prudent to make the pad size at least .010" larger than the total fixture and board tolerance stack-up.

II. Fixturing Considerations

For a more reliable and repeatable test, the following test point locations should be considered in order of importance. Lower levels of test point recommendation increase the cost of the fixture (and the associated maintenance) and also reduce the repeatability, reliability and life expectancy of the test fixture.

a) Primary test point locations for SMT boards should be maintained as follows:
    1) Minimum .086" centers
    2) Solder side test pad or via
b) Secondary test point locations are as follows:
    1) Minimum .066" centers
    2) Unsoldered via
c) Last choice test point locations are as follows:
    1) Minimum .046" centers
    2) Top side access
    3) SMD lead

Remember, when probing SMD leads, if the solder junction is open, the probe can and will force contact to be made between the lead and the pad. This will mask opens and pass some boards that would fail if a separate test access location were available.

Be sure to supply test fixture drilling information from Gerber files and/or CAD ASCII database files. This eliminates the artwork tolerance found both in the film, typically +/- .001", and in operator error when digitizing from the film.