

DESIGN FOR TESTABILITY

TOOLING HOLES

- Standard PCB 0.125" (+0.003" – 0.000")
- Accuracy between tooling hole centers 0.002"
- Tooling hole placement from board edges 0.200"
- Tooling holes to be non-plated
- Locate tooling holes as far apart from each other as possible
- No components or test pads on bottom side within 0.125" of tooling holes
- No components or test pads on bottom side within 0.125" of edge of PCB
- Minimum of 2 tooling holes diagonal of one another on PCB

TEST PROBES, TEST PAD, VIA'S

- | <u>Test Probe
Diameter</u> | <u>Test Pad Size
Ideal to Minimum</u> |
|--------------------------------|---|
| • 0.050" (50 mil) | 0.040" to 0.030" square on 0.050" centers |
| • 0.075" (75 mil) | 0.060" to 0.040" square on 0.075" centers |
| • 0.100" (100 mil) | 0.060" to 0.040" square on 0.100" centers |
- Test pad sizes smaller than 0.030" are nailable but additional fixturing costs may be incurred.
 - Test pad must be placed 0.200" from bottom side components exceeding 0.250" height.
 - Solder mask, ink or silk screen must not cover test pad.
 - Test pad should be square so it is distinguishable from vias.
 - Test pad should have a conductive, non-oxidizing surface.
 - All non-test pad vias must be filled (tented) to prevent leakage during ICT.
 - Use bottom side through-hole component pins as test pads whenever available.
 - Board origin to test pad center +0.002" max.
 - Multiple test pads for PCBA power and ground are required and should be spaced evenly throughout the PCB.
 - Test pads should be evenly distributed over the PCB. Spring force from large concentrations of test probes can cause warpage and fixture vacuum cycle problems.
 - General rule of thumb is provide one ground for every 28 nodes

GENERAL INFORMATION

- Ideal design defined as: All test pads to 100% of signal nets accessible from bottom side of PCB, pad size: 0.040" to 0.060" square at 0.100" spacing, no open vias.
- Test pads, vias and through-hole components are typical fixture nail targets. Probing SMT component legs is not recommended.
- PCB design should allow 100% signal net access, with 100% of test pads on bottom side of PCB. Probing both sides of a PCB will significantly increase fixturing costs.
- Any bottom-side component, which exceeds 0.250" in height, will require additional ICT fixture bottom plate milling and vacuum damming costs.