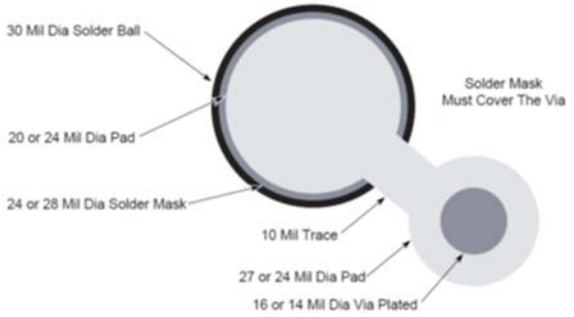
Design rules for PCB Assembly

1. Biggest board SMT machine can handle- 22x24 inches.
2. Closest spacing between 2 parts is 10mil (edge of part to edge of part).
3. SMT machine part placement accuracy is +/-001.
4. Closest part to board edge should be 10mil. This is more dictated by fabrication. It is recommended to add rails (at least on 2 sides). For non-square/rectangular board, array will be required in order for board to be ran on SMT machine.
5. Each board should have 3 fiducials (in 3 corners of the board). Fiducials should be .030 round exposed copper pad (they should not be cover by solder mask).
6. Parts rotated 45, 90, 180 degree are okay. Do not rotate parts on odd angles such as 15, 22, 32 degree.
7. Via-in-pad technology- vias should be plugged and plated over. Exposed vias in solder pad can lead to solder bridging or insufficient solder wetting.
8. All vias around BGA pads should be tented over or plugged with solder mask.



9. Solid ground pads (such as for QFN) should have a cross-hatch pattern in the solder paste data. Since this is a solid pad, the idea is not to have too much

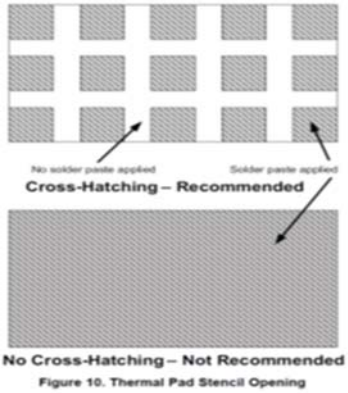
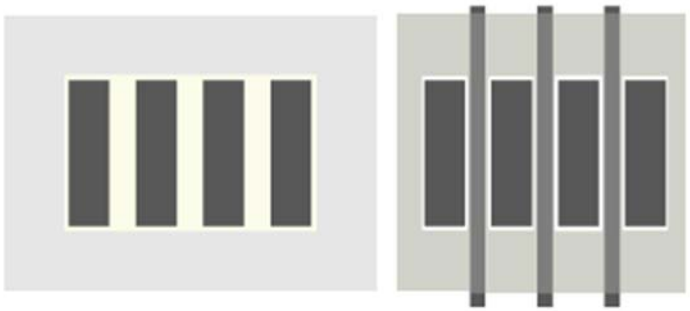


Figure 10. Thermal Pad Stencil Opening

10. Standard solder paste stencil is 4 or 5mil thickness. 3mil stencil is also available.
11. Smallest solder mask DAM between every SMT pad (4mil minimum). It is recommended that there be solder mask dam between every SMT pad. Gang masks are not recommended because they are susceptible to solder bridging.



12. Assembly Drawings (in PDF format) should have reference designators for each location. Pin 1 or polarity orientation should be clearly marked.(Required)
13. Parts sensitive to heat, water, air pressure, etc. should be clearly noted on the assembly drawing (these parts will be hand solder or install as a last operation before shipping).
14. For flying probe testing, there should be a test point for each net. Otherwise 100% flying probe testing cannot be preform (using a component pad as a test point is not reliable).
15. ODB++ (non-compressed) is (Preferred). ODB++ has everything embedded in it such as XY data, part orientation, etc.
16. XY data preferred format are EXCEL or TXT file. (Required)
17. CAD ASCII file or IPC356 Netlist (Preferred).
18. BOM (Bill of Material) preferred format is EXCEL. BOM should include reference designators, manufacturer part number, part description, and quantity per item. (Please also include all DO NOT LOAD items/parts on the BOM. It is good practice to have every location accounted for just in case there was a problem with exporting the BOM.) (Required)
19. Array solder paste is required if boards are in array (you can get this from the fab house). (Required)
20. SMT 1 or 2 sides are fine (SMT 2 sides will cost more to assemble for obvious reasons).
21. Thru parts will add assembly time and should be avoided for high volume production.