TITLE: 100-LEAD 12mm P=0.40mm QFN CAVITY PACKAGE

LEAD NUMBERING

www.MirrorSemi.com
Notes: (Unless Otherwise Specified).

1. DIMENSIONS ARE PRESENTED ONLY AS A GUIDELINE. DESIGNERS SHOULDN'T USE THEIR OWN KNOWLEDGE WHEN DESIGNING THE PCB. SURROUND EACH SIDE OF I/O PERIMETER PADS WITH 0.060~0.075 mm (2.4~3.0 mils) NSMD SOLDER MASK.

2. OPENING. Optionally OK to use rectangle (NSMD) mask opening around I/O pads.

3. Rounded PCB land pads reduce solder bridging. Pad chamfer angle may vary.

4. PCB lands should be 0.2 mm longer than the package I/O pads.

5. The width of perimeter PCB pads should match (1:1) the width of the package pads.

6. Refer to industry references such as IPC-SM-782 for PCB land pattern design.

7. Thermal ground pads may be changed to suit requirements of the designer.

8. Make copper thermal pad as large as possible. Drill multiple thermal vias 0.25~0.33 mm diameter using 0.8~1.2 mm pitch grid. Plate thermal via barrels with 1-ounce copper (18 µm). Tent (cover) thermal vias with solder mask 0.1 mm larger than the via diameter.

9. Stencil design may be changed to suit requirements of the designer.

10. Laser cut stencil 0.125 mm (5 mils) thick. Aperture size-to-land ratio of 1:1.

11. The solder paste opening in the thermal pad area should be a matrix array of smaller apertures instead of one large aperture to control paste amounts.

12. Apply 50% to 80% solder paste coverage in the pad area.

13. Recommended PCB layout:

   - Component pad
   - PCB pad
   - Component pad
   - Solder mask opening

14. PCB thermal pad design (optional, see note 7, 8)