SECTION A-A

Notes: (Unless Otherwise Specified)
1) BODY: PLASTIC, SEMICONDUCTOR GRADE
2) LEAD FRAME: COPPER, C-194F/H
3) LEAD FRAME PLATING: Ni, Pd, Au
4) FRAME THICKNESS: 0.203mm
5) DIE PAD: 3.54 X 3.54mm
6) JEDEC OUTLINE: MO-220

All dimensions in INCHES

TOLERANCES UNLESS NOTED

<table>
<thead>
<tr>
<th>TOLERANCE</th>
<th>UNLESS NOTED</th>
</tr>
</thead>
<tbody>
<tr>
<td>XX</td>
<td>± 0.05</td>
</tr>
<tr>
<td>XX</td>
<td>± 0.01</td>
</tr>
<tr>
<td>XXX</td>
<td>± 0.005</td>
</tr>
<tr>
<td>XXXX</td>
<td>± 0.0005</td>
</tr>
</tbody>
</table>

ALL DIMENSIONS IN INCHES

THIRD ANGLE PROJECTION

DO NOT SCALE DRAWING

SCALE 14:1

A

www.MirrorSemi.com
1. Dimensions are presented only as a guideline. Designers should use their own knowledge base when designing the PCB.

2. Surround each side of I/O perimeter pads with 0.060~0.075 mm (2.4~3.0mils) NSMD solder mask 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 0.2mm 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-6012M-1 for PCB land pattern design.

7. Thermal ground pads may be changed to suite requirements of the designer.
   A. Make copper thermal pad as large as possible.
   B. Drill multiple thermal vias 0.25~0.33mm diameter using 0.8~1.2mm pitch grid.
   C. Plate thermal via barrels with 1-ounce copper (18µm).
   D. Tent (cover) thermal vias with solder mask 0.1mm larger than the via diameter.

8. Stencil design may be changed to suite requirements of the designer.
   A. Laser cut stencil 0.125mm (5mil) thick. Aperture size-to-land ratio of 1:1.
   B. 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.
   C. Apply 50% to 80% solder paste coverage in the pad area.