PCB Land Pattern Design and Surface Mount Guidelines for HDA POL Modules

Introduction

Intersil's HDA POL Module Product family offers a relatively new packaging concept that is currently experiencing rapid growth. The Module Product family features the HDA (High Density Array) Package and encompasses lead pitches of 1.0mm and above. This offers a variety of benefits including reduced lead inductance and both perimeter I/O pins to ease PCB trace routing, and in board I/O pins for complex pinouts. Also, the exposed Au plated copper die-pad technology offers good thermal and electrical performance. These features make the HDA packaged POL module an ideal choice for many new applications where thermal and electrical performance are important.

There are general industry references, such as IPC-SM-782, for printed circuit board (PCB) land pattern design. But because the HDA package style is relatively new, such industry guidelines for it are still in development. This Tech Brief helps by providing general guidelines for use in developing land pattern layouts and solder mounting processes.

Module HDA Package Outline Drawings

Intersil’s individual product data sheets reference to the appropriate Intersil package outline drawings. These in turn reference compliance to any applicable industry standard outlines.

General Design Guidelines

The POL Module HDA die pad and perimeter I/O pads are fabricated from a planar copper lead-frame substrate. This is encapsulated in plastic with the bottom of the die pad and I/O pads exposed to create an “exposed-pad” package. Both the I/O pads and die pad should be soldered to the PCB.

The corresponding PCB lands need to be designed to fit well within the PCB assembly process capabilities, as well as promote good long term solder joint reliability. Note that soldering the exposed die pad “anchors” the package and provides important thermo-mechanical temp cycling stress benefits that improve the reliability of the I/O pad solder joints.

The PCB “thermal land” design for the exposed die pad should include thermal vias that drop down and connect to buried metal plane(s). This combination of vias for vertical heat escape and buried planes for heat spreading allows the HDA to achieve its full thermal potential.

Land Pattern Guidelines

Peripheral I/O Lands

The I/O land should match the package I/O pads (1:1) see Figure 2. Design dimension guidelines are shown in the individual package outline drawing.

PCB Thermal Land

The thermal land should match 1:1 with the package exposed die pad. See Figure 2.

FIGURE 1. POL MODULE HDA PACKAGES

It should be emphasized that these guidelines are general in nature and should only be considered a starting point in this effort. The user must apply their actual experiences and development efforts to optimize designs and processes for their manufacturing practices and the needs of varying end-use applications.
Thermal Vias

A grid of 1.0mm to 1.2mm pitch thermal vias, which drop down and connect to buried copper plane(s), should be placed under the thermal land. The vias should be about 0.3mm to 0.33mm in diameter with the barrel plated to about 1.0 ounce copper.

Although adding more vias (such as by decreasing via pitch) will improve thermal performance, diminishing returns will be seen as more and more vias are added. Therefore, simply use as many vias as practical for the thermal land size and your board design ground rules.

Solder Mask Design

Two types of land patterns are used for surface mount packages:


Better control of the copper etching process as compared to the solder masking process makes NSMD preferable. Also, the SMD pad definition can introduce stress concentrations near the solder mask overlap region that can result in solder joints cracking under extreme fatigue conditions. Using NSMD instead improves the reliability of solder joints as solder is allowed to “wrap around” the sides of the metal pads on the board.

For these reasons, NSMD is recommended for the I/O lands and generally for the thermal land; however SMD should be used on the thermal land when it is relatively large as discussed in the following.

For NSMD pads, the solder mask opening should be about 120µm to 150µm larger than the pad size, providing a 60µm to 75µm design clearance between the copper pad and solder mask. Rounded portions of package pads should have a matching rounded solder mask-opening shape especially at corner leads, to allow for enough solder mask web to prevent solder bridging.

Typically each pad on the PCB should have its own solder mask opening with a web of solder mask between adjacent pads.

However, for fused pads, space may not be available for solder mask web in between the pads. In that case, use one big opening designed around a whole strip of pads (for instance all the pads on one side of a package) with no solder mask in between the pads.

For package designs with exposed die pad sizes near the maximum available for that package, the gap between the thermal land and I/O pads may be small. In this case there may be more potential for solder bridging, so the thermal land should be solder mask defined (SMD). The mask opening should be approximately 100µm smaller than the thermal land on all four sides, which increases the solder mask web between the thermal and I/O lands.

Solder masking is also required for thermal vias to prevent solder wicking inside the vias, drawing solder away from the thermal land-to-die pad interface. The solder mask diameter should be about 100µm larger than the via diameter. The vias can be plugged or tented with solder mask, either from the bottom or top surface of the PCB. Tenting from the top is considered a better option as it results in smaller voids under the die pad. Solder masking of vias from the bottom side can result in increased outgassing during reflow, creating bigger voids around vias. However, note that small voids in this area are not unusual and will have little effect on thermal or electrical performance or on the reliability of the perimeter I/O pad solder joints.

Stencil Guidelines

Stencil Design for I/O Lands

Reflowed solder joints on the I/O lands should have about a 50µm to 75µm (2 mil to 3 mil) standoff height. The solder paste stencil design is the first step in developing such optimized reliable joints.

Stencil aperture size-to-land size should typically be a 1:1 ratio. For finer pitch parts, especially as tight as 0.4mm, the aperture width may need to be reduced slightly to help prevent solder bridging between adjacent I/O lands.
Stencil Design for Thermal Land

To reduce solder paste volume on the large thermal lands, it is recommended that an array of smaller apertures be used instead of one large aperture. The smaller apertures can be circular or square and of various dimensions and array sizes; the main goal being a dimensional combination that results in 50 to 80% solder paste coverage of the land area. This reduced coverage on the thermal land is important in achieving good solder joints and good temperature cycling related reliability at the perimeter I/O lands. Also, reducing the solder paste coverage area will reduce the amount of “Component Tilt”, which is the result of too much deposited solder paste on the large thermal land. The thermal land’s solder paste coverage should not be reduced too low (L.T. 50%) since this could potentially affect the "self-alignment" feature of the package during SMT reflow and could contribute to increased solder voiding, which could affect thermal performance.

Stencil Type and Thickness

A laser-cut, stainless steel stencil with electropolished trapezoidal walls is recommended. Electropolishing “smooths” aperture walls, resulting in reduced surface friction, good paste release and void reduction. Using a trapezoidal section aperture (TSA) promotes paste release and also forms “brick-like” paste deposits that assist in firm component placement.

Solder Paste and Reflow Profile

Due to the low mounted height of the HDA, “No Clean” Type 3 paste per ANSI/J-STD-005 is recommended. Nitrogen purge is also recommended during reflow.

A system board reflow profile depends on the thermal mass of the entire populated board, so it is not practical to define a specific soldering profile just for the HDA. The profiles given in Figure 3 and Tables 1, 2 and 3 are provided as guidelines to be customized for varying manufacturing practices and applications.

Solder Joint Criteria

HDA packages feature “Bottom Only” type terminations as described in Section 7.6.15 of the Joint Industry Standard IPC/EIA J-Std-001: “Requirements for Soldered Electrical and Electronic Assemblies”. The solder joint requirements for HDA packages are given in Table 4 and Figure 4. It is important to note that the bottom only HDA termination does NOT have ANY requirement for solder joint fillet heights of any kind. Only solder joint length, width and solder joint thickness are specified. X-ray inspection, similar to what is done on BGA assemblies, is the preferred inspection technique.
TABLE 1. PACKAGE PEAK REFLOW TEMPERATURES - Sn/Pb

<table>
<thead>
<tr>
<th>PACKAGE THICKNESS</th>
<th>VOLUME mm³ ≥350</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;2.5mm</td>
<td>225 +0°C/-5°C</td>
</tr>
<tr>
<td>≥2.5mm</td>
<td>225 +0°C/-5°C</td>
</tr>
</tbody>
</table>

TABLE 2. PACKAGE PEAK REFLOW TEMPERATURES - Pb-FREE

<table>
<thead>
<tr>
<th>PACKAGE THICKNESS</th>
<th>VOLUME mm³ 350 -2000</th>
<th>VOLUME mm³ &gt;2000</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.6mm to 2.5mm</td>
<td>250 +0°C (Note 1)</td>
<td>245 +0°C (Note 1)</td>
</tr>
<tr>
<td>≥2.5mm</td>
<td>245 +0°C (Note 1)</td>
<td>245 +0°C (Note 1)</td>
</tr>
</tbody>
</table>

NOTE:
1. Tolerance: The device manufacturer/supplier shall assure process compatibility up to and including the stated classification temperature (this means Peak reflow temperature +0°C. For example 260°C + 0°C) at the rated MSL level.

TABLE 3. REFLOW PROFILE PARAMETERS

<table>
<thead>
<tr>
<th>PROFILE PARAMETER</th>
<th>Sn/Pb</th>
<th>Pb-FREE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average Ramp-Up Rate (TsMAX toTp)</td>
<td>3°C/second maximum</td>
<td>3°C/second maximum</td>
</tr>
<tr>
<td>Preheat Temperature Min (TsMIN)</td>
<td>+100°C</td>
<td>+150°C</td>
</tr>
<tr>
<td>Temperature Max (TsMAX)</td>
<td>+150°C</td>
<td>+200°C</td>
</tr>
<tr>
<td>Time (tsMIN to tsMAX)</td>
<td>60 to 120 seconds</td>
<td>60 to 180 seconds</td>
</tr>
<tr>
<td>Time maintained above:</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Temperature (T_L)</td>
<td>+183°C</td>
<td>+217°C</td>
</tr>
<tr>
<td>Time (t_L)</td>
<td>60 to 150 seconds</td>
<td>60 to 150 seconds</td>
</tr>
<tr>
<td>-Peak/Classification Temperature (T_p)</td>
<td>See Table 1</td>
<td>See Table 2</td>
</tr>
<tr>
<td>Time within +5°C of Actual Peak Temperature (tp)</td>
<td>10 to 30 seconds</td>
<td>20 to 40 seconds</td>
</tr>
<tr>
<td>Ramp-Down Rate</td>
<td>+6°C/second maximum</td>
<td>+6°C/second maximum</td>
</tr>
<tr>
<td>Time +25°C to Peak Temperature</td>
<td>6 minutes maximum</td>
<td>8 minutes maximum</td>
</tr>
</tbody>
</table>

FIGURE 4. HDA TERMINATIONS
After PCB assembly, the package should be inspected in transmission x-ray for the presence of voids, solder balling or other defects underneath the package. Cross sectioning may also be required to determine the fillet shape and size and the joint standoff height. For rework of defects underneath the package, the whole package needs to be removed.

Removal and rework of HDAs should be done on a rework station with thermal profile control (see Figure 5). The following steps are provided as a guideline – a starting point in developing a successful rework process.

**Bake**

Before rework, bake the PCB assembly at +125°C for at least 24 hours to remove any residual moisture.

**Component Removal**

Ideally, the reflow profile for part removal should be similar to that of the component attachment. However, the time above liquidus can be reduced as long as the reflow is complete.

Typical rework stations will heat the board from the bottom side using convective heaters and from the top side with hot gas nozzle directing heat at the component to be removed (see Figure 6). An appropriate thermal profile must be developed to preheat, soak and reflow the module on the PCB.

### TABLE 4. DIMENSIONAL CRITERIA - HDA

<table>
<thead>
<tr>
<th>FEATURE</th>
<th>DIM.</th>
<th>CLASS 1</th>
<th>CLASS 2</th>
<th>CLASS 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum Side Overhang</td>
<td>A</td>
<td>50% (W), (Note 2)</td>
<td>25% (W), (Note 2)</td>
<td></td>
</tr>
<tr>
<td>Toe Overhang (outside edge of component termination)</td>
<td>B</td>
<td>Not Permitted</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Minimum End Joint Width</td>
<td>C</td>
<td>50% (W)</td>
<td>75% (W)</td>
<td></td>
</tr>
<tr>
<td>Minimum Side Joint Length</td>
<td>D</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Solder Fillet Thickness</td>
<td>G</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Minimum Toe (End) Fillet Height</td>
<td>F</td>
<td>(Notes 3, 6)</td>
<td>G + H (Notes 3, 6)</td>
<td></td>
</tr>
<tr>
<td>Termination Height</td>
<td>H</td>
<td>(Note 6)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Solder Coverage of Thermal Pad</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Land Width</td>
<td>P</td>
<td>(Note 3)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Termination Width</td>
<td>W</td>
<td>(Note 3)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**NOTES:**

2. Does not violate minimum electrical clearance.
3. Unspecified parameter or variable in size as determined by design.
4. Wetting is evident.
5. Not a visually inspectable attribute.
6. “H” = height of solderable surface of lead, if present. Some package configurations do not have a continuous solderable surface on the sides and do not require a toe (end) fillet.

**Rework Guidelines**

![FIGURE 5. REWORK STATION WITH THERMAL PROFILE CONTROL](image)

![FIGURE 6. TOP AND BOTTOM PREHEAT](image)
The circuit-board component being removed should reach at least +55°C ± 5°C using a temperature ramp of +1°C to +3°C/min before ramping to reflow temperature (see Figures 7 and 8).

Heating of adjacent components should be minimized. Use the lowest possible reflow peak temperature for component removal.

Once the joints have reflowed, the HDA is removed with a vacuum pick-up (see Figures 9 and 10).

Site Redress

Clean the site properly, removing residual solder with an appropriate vacuum nozzle and site repress process, which is typically provided with the rework station (see Figures 11 and 12).
Solder Paste Printing

Use a miniature stencil (see Figure 13) specific to the component and align/attach the stencil aperture under appropriate magnification ~50 to 100x (see Figure 14). Deposit the paste with a small squeegee blade (see Figure 15). The blade width should be the same as the package width to ensure single pass paste deposition, avoiding any overprinting.
Component Placement and Reflow

Carefully remove alignment securing device and place stencil and module into pick-up position (see Figure 16). The HDA has good self-centering abilities during solder reflow, so the placement of this package should be similar to that of BGA's. The placement machine should have fine adjustment capability in the x, y and rotational axes. Since the pads are on the underside of the package, use an optical system (provided with the rework station) that can overlay an image of the solder paste pattern to aid in component alignment (see Figure 17), which should be done at 50 to 100x magnification. Then, pick and place new component (see Figure 18). Reflow the PCB using the same profile as that developed for the initial attachment to complete the rework (see Figure 19).

FIGURE 16. PLACEMENT OF NEW DEVICE

FIGURE 17. OVERLAY IMAGE ALIGNMENT

FIGURE 18. COMPONENT REPLACEMENT

FIGURE 19. COMPLETED REWORK