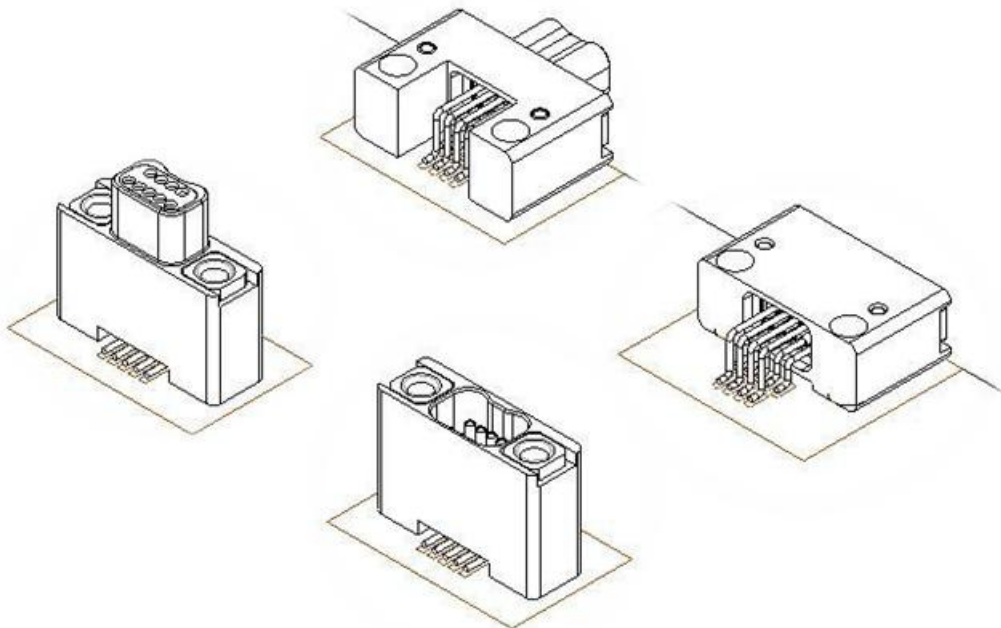




Purpose:

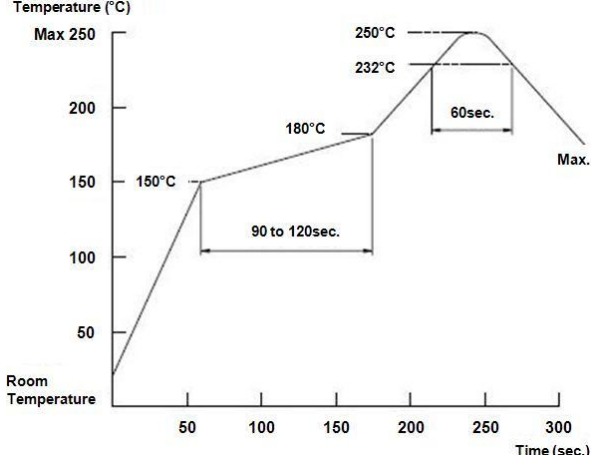
The purpose of this technical bulletin is to provide information regarding the proper procedure for processing AirBorn's Nano "surface mount" connectors to printed circuit boards.



Images are for reference only.



Usage Recommendations:

<p>1. Reflow solder temperature profile</p> <p>Solder composition (tin/lead): Paste, 63%SN/37%PB</p> <p>Solder composition (lead free): Paste, 96.5%Sn/3.0%Ag/0.5%Cu</p>	 <p>(Typical solder profile depicted. See solder supplier for recommended temperature profile.)</p> <p>Note 1: Up to 2 cycles of Reflow soldering are possible under the same conditions, provided that there is a return to normal temperature between the first and second cycle.</p> <p>Note 2: The temperature profile indicates the board surface temperature at the point of contacts with the connector terminals.</p> <p>Note 3: Care must be taken in selecting a temperature profile to prevent damage to other components on the PCB.</p>
<p>2. Recommended stencil thickness and open area ratio</p>	<p>Thickness: 0.006 inch Open area ratio: 94% recommended 100% acceptable</p>
<p>3. Precautions</p>	<ul style="list-style-type: none"> *Mating and un-mating of the connectors when not soldered on the boards is not recommended as this may cause damage of the solder terminals, contacts or insulators. *When mating/un-mating follow proper mating sequence taking care NOT to damage connectors. See AirBorn ESGN6006. * Slight discoloration of the insulating materials will not affect form, fit or function of the connectors.

PC Board

A. Material

PC board material shall be glass Epoxy (FR-4) with a Tg of 170°C or higher or Polyimide with a Tg of 200°C or higher.

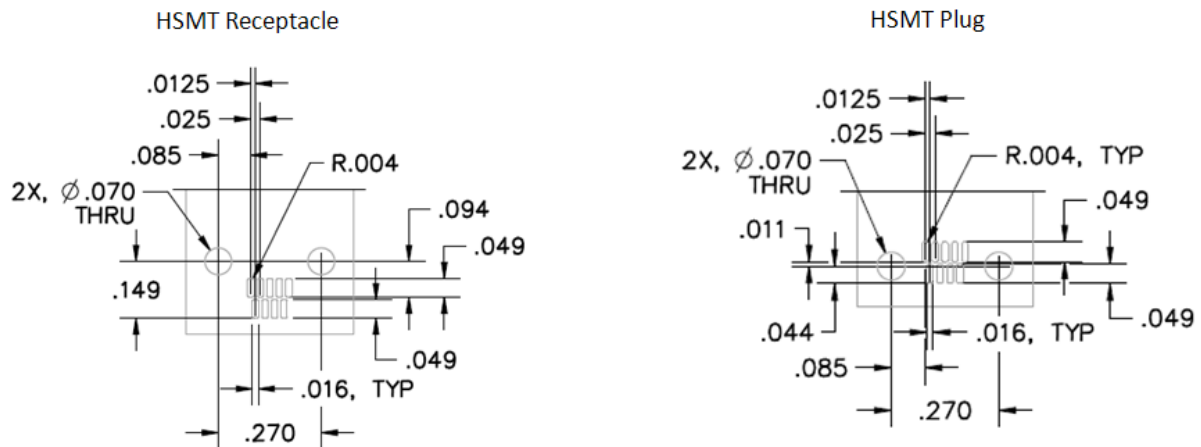
B. Tolerance

Maximum allowable bow of the pc board shall be 0.002 inch over the length of the connector.

C. Layout



Figure 1 represents typical pc board vertical and horizontal layouts for this product. For layout dimensions for specific product, obtain the appropriate layout from the AirBorn catalog. The layout is viewed from the connector side.



All dimensions are in inches and for reference only.
2 row product shown.

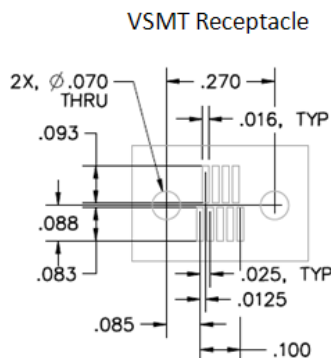


Figure 1

Special Handling

Connectors must be handled by the housing or by “pick & place” clip only and not by the solder leads.

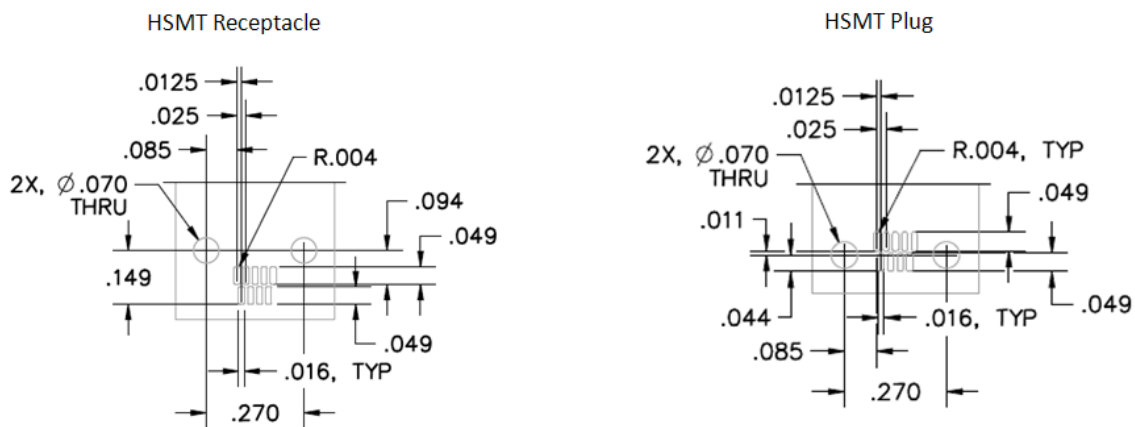
Stencil

AirBorn recommends using a solder stencil to minimize solder bridging between lands. Stencil aperture will be determined by the thickness of the stencil being used. Generally the stencils aperture will be smaller than layout dimensions to maintain a given volume of solder paste. Solder deposition should be within the pad area of the contact leads. Typical aperture openings are shown in Figure 2.

NOTE



The recommended aperture width is 0.015 ± 0.001 in. It may be wider; however, care must be given to ensure against solder bridging during processing.



All dimensions are in inches and for reference only.

2 row product shown.

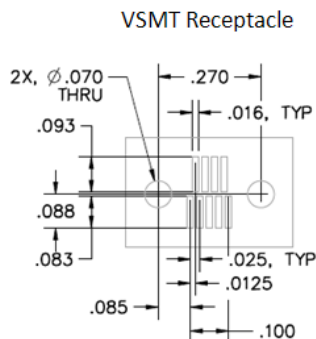


Figure 2

Solder Mask

See AirBorn catalog for recommended footprint, solder mask, and PCB layout.

Connector Placement

Machine placement of the connector is recommended due to the inherent difficulty of manually placing fine-pitch connectors. The top surface of the connector housing has a flat area to facilitate vacuum pick-up and handling. To avoid damage, the connectors should be picked up directly out of the embossed-tape packaging by the pick-up device. The placement machine is used to position the connectors to minimize the possibilities of damage that could result from improper handling.



Placement of the connectors may be done by hand; however, extreme caution must be used when handling connectors to prevent deformation and contamination of the solder leads.

Connector Alignment

Optimally, the connector contact solder leads should be centered on the pc board pads. However, misregistration is permissible for some performance classifications. See Figure 3.

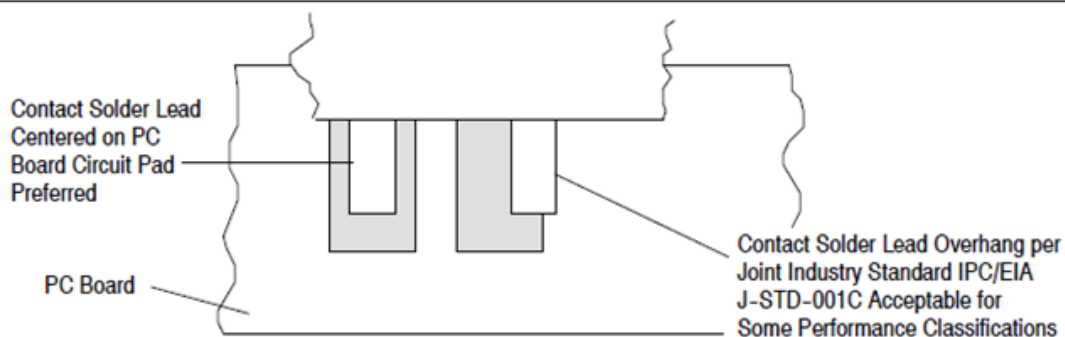


Figure 3

Connector Mounting Hardware

Connector hardware should be installed after the reflow solder process. Installation of hardware before the reflow solder process could negatively affect solder lead pad alignment.



Cleaning

After soldering, removal of fluxes, residues, and activators is necessary for best connector performance. Consult with the supplier of the solder and/or flux for recommended solvents. Cleaners must be free of dissolved flux and other contaminants. If cleaning is desired AirBorn recommends cleaning with the pc board on its edge. If using an aqueous cleaner, standard equipment such as a soak-tank or an automatic in-line machine should be used.



Even when using “no clean” solder paste, it is imperative that the contact interface be kept clean of flux and residue for optimal connector performance. Flux may migrate under certain conditions with elevated temperatures and, therefore, cleaning is necessary.

Drying

Air drying of cleaned connectors is recommended. Temperature for the connectors should not exceed -55° C to +125°C [-67°F to 257°F]. Degradation of the connectors could result from extreme temperatures.

Checking of Installed Connector

Solder joints should conform to the performance specification being designed to. The connector housing must visually seat on the pc board for proper solder terminal fillets. Solder terminal fillets should conform to the performance specifications being designed to.

SMT X-ray Results

SMT solder joint x-ray photos shown below depict typical results when the described process and materials depicted in this report are utilized. Results may vary depending on PCB design, solder paste and process equipment used.

