SMT SENSOR PCB MOUNTING GUIDELINES

Technical Note

The purpose of this Technical Note is to assist the end-user with mounting Honeywell SMT (Surface Mount Technology) sensors to PCBs (Printed Circuit Boards).

1.0 SMT PRINTED CIRCUIT BOARD CONSIDERATIONS

Regardless of which type of solder is used (lead-free or Sn/Pb), ensure that the PCB surface finish and material meet the lead-free application requirements due to their higher reflow temperature and lead-free solder compatibility.

1.1 PCB SURFACE FINISH

Selection of a suitable surface finish depends on the end-user's PCB design, assembly process, handling/storage, and cost requirements. The most common surface finishes compatible with a lead-free SMT process are:

- Organic Solderability Preservatives (OSP)
- Electroless Ni/Immersion Au (ENIG)
- Immersion Ag
- Immersion Au

1.2 PCB MATERIALS

Due to lead-free solder's higher reflow temperature requirement, use a PCB laminate material with a $T_g \geq 170°C$ [338°F].

1.3 PCB PREPARATION BEFORE SOLDER PASTE PRINTING

Bake PCBs at elevated temperatures within eight hours of use. This step reduces excessive moisture from the PCB. (Moisture in the PCB, under solder resist layers, trapped within layers, etc., may lead to excessive solder defects.) A bake time of four hours minimum at 65°C [149°F] is generally adequate.

NOTICE

Process debug and final disposition of the SMT process is the responsibility of the end user.

2.0 SOLDER PASTE PRINTING PROCESS

2.1 SOLDER PASTE AND FLUX MATERIALS

2.11 SnAgCu Lead-Free Solder Alloy

Typical lead-free SnAgCu solder has a melting temperature of 217°C to 221°C [423°F to 430°F] for solder reflow applications. This alloy is widely accepted in the semiconductor industry due to its low cost, relatively low melting temperature, and good thermal fatigue resistance. The reliability of SnAgCu solder alloys (see Table 1) and their physical properties is almost as effective as the current lead-containing solders.

TABLE 1. SnAgCu FAMILY OF LEAD-FREE SOLDER ALLOYS

METAL	PROPORTION
Sn	95.5% to 96.5%
Ag	3.0% to 4.0%
Cu	0.5% to 0.7%

Honeywell

2.12 Flux Considerations

- "No-clean" flux must be used for soldering.
- If a water soluble (WS) flux is used, use deionized (DI) water to clean the flux residue from the PCB assembly. Ensure adherence to the following precautions:
  - Water or cleaning solution does not enter the sensor.
  - The sensor is not immersed in liquid.
  - Both pressure ports, or the pressure port and the reference hole, are covered if a wash process is used.

NOTICE

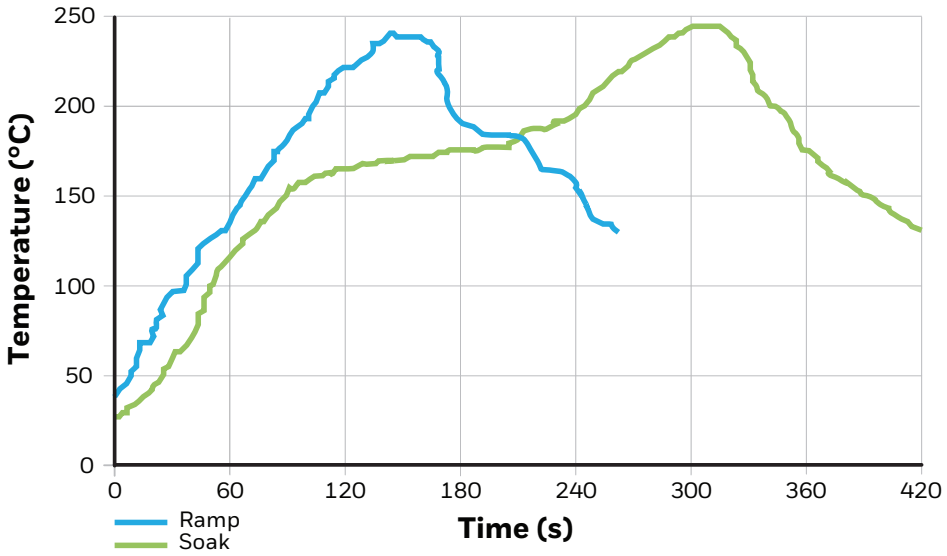
Honeywell is not responsible for potential and foreign material contamination found in the device cavities due improper cleaning processes.

2.2 LEAD-FREE SOLDER PASTE REFLOW PROFILES

Ensure that the solder paste reflow profile follows the solder paste manufacturer's recommendations and that the sensor's housing does not exceed its specified maximum reflow temperature. Figure 1 shows two typical lead-free solder paste reflow profiles:

- **Ramp:** Suited for use in most applications for enhanced solder performance.
- **Soak:** Suited for use in applications where the PCB has a large thermal mass or a large temperature delta (ΔT).

FIGURE 1. TYPICAL LEAD-FREE SOLDER PASTE REFLOW PROFILES



2.3 LEAD-FREE REFLOW SOLDERING PROCESS

2.31 Forced Air Convection

Forced-air convection reflow soldering is the most common way to secure SMT sensors to PCBs. After applying the solder paste to a PCB and placing the sensors onto the paste, the reflow process consists of conveying the board through an oven with successive heating elements of varying temperatures. In the oven, each board typically goes through the following stages:

- Gradual preheating
- Brief duration at high soldering temperature
- Controlled cooling process

Critical parameters for effective soldering are:

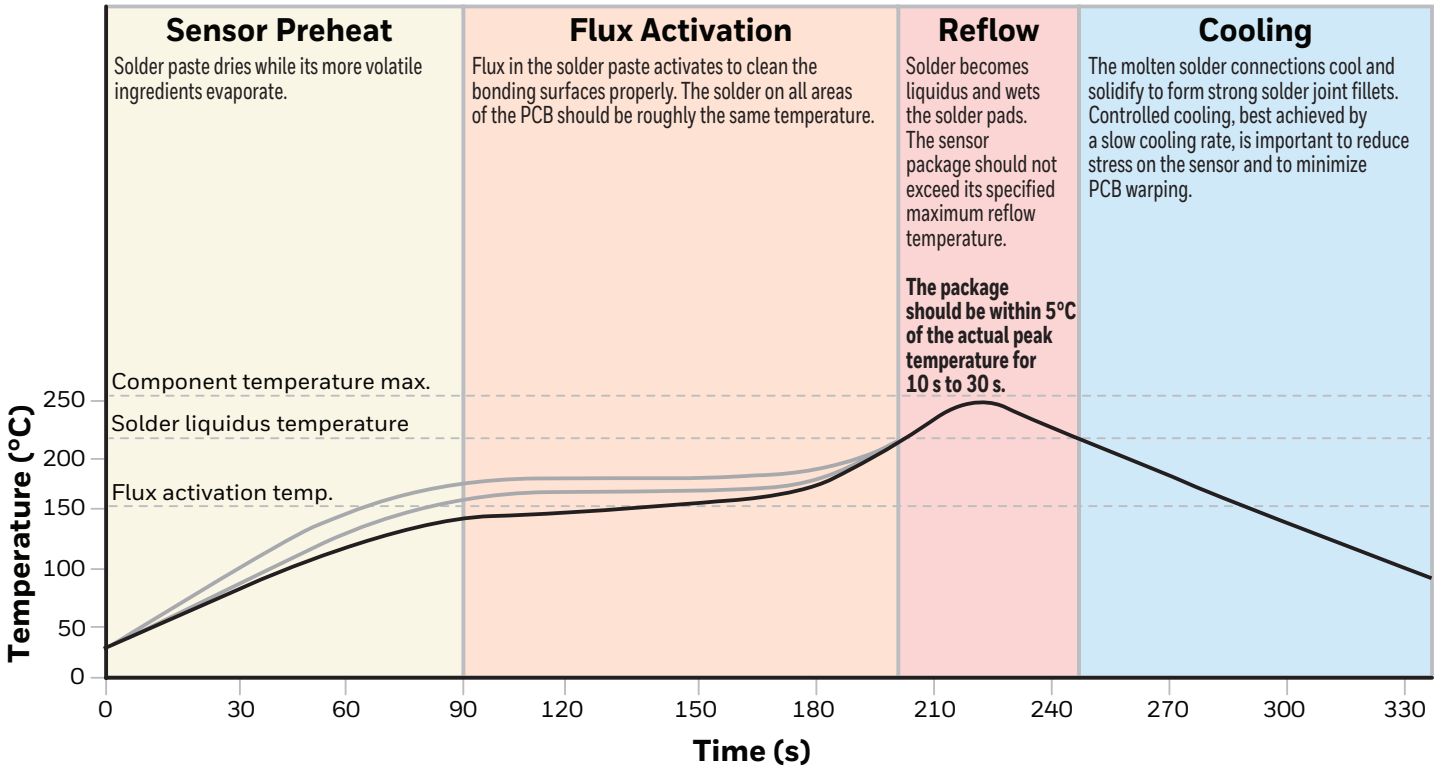
- Maximum temperature
- Heating rate
- The time a device spends at each temperature
- Controlled heating
- Controlled cooling

2.32 Reflow Profile Stages

Because different PCB designs accommodate different numbers and types of sensors, solder pastes, reflow ovens, and PCBs, no single temperature profile works for all possible combinations. However, following the proper guidelines and PCB-specific characterization leads to successful sensor mounting to the PCB. Figure 2 shows the four stages of a reflow profile:

For non-Honeywell sensors, compare the reflow profile temperatures to the sensor manufacturer recommendations to ensure that the maximum temperature limitations on all materials are not exceeded.

FIGURE 2. LEAD-FREE REFLOW PROFILE STAGES



2.33 Solder Paste-Related Defects

- **Poor solder wetting:** If poor solder wetting occurs across the entire PCB, implementing a hybrid profile may often address this issue. If poor solder wetting is related to the sensors, the root cause is generally a plating issue with either the sensor itself or the PCB pads. To enhance wetting, shorten the overall profile and increase the peak temperature.
- **Solder voids:** If voiding occurs, the root cause is generally flux related because solder flux loading may be as high as 50% of the solder. The currently used reflow profile does not adequately activate from the solder joint. Again, a hybrid profile often addresses this issue. To reduce voiding, increase the flux activation stage by increasing time and/or temperature. If the desired results are not meeting requirements, then consult the solder paste manufacturer's technical representative.

2.4 SOLDER PASTE STORAGE

- **Storage temperature:** Typically 0°C to 10°C [32°F to 50°F]. Store the cartridges with their tips down to prevent air pocket formation. Remove the solder paste from cold storage at a minimum of eight hours before use. Do not use forced heating methods to bring the solder paste up to temperature.
- **Shelf life:** Typically six months when stored tightly sealed in its original packaging at the proper storage temperature. Use air shipment to minimize the time solder containers are exposed to higher temperatures.

2.5 SOLDER PASTE STENCIL DESIGN

Solder paste application is the first step of the SMT process and stencil selection is very critical. Optimal stencil specifications are:

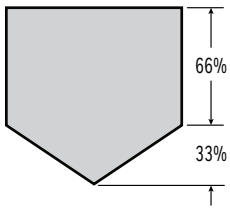
- **Material:** Stainless steel foil
- **Type:** Laser cut
- **Foil thickness:** 0,127 mm [0.005 in]
- **Stencil aperture reduction:** 10%

Ensure the stencil aperture geometry is optimized to reduce solder balls. Figure 3 shows two geometries that help control and mitigate the risk of SMT defects due to solder ball formation.

FIGURE 3. RECOMMENDED STENCIL APERTURE GEOMETRIES

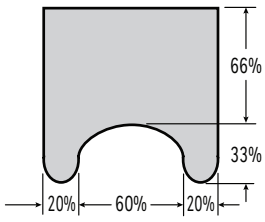
Home Plate

- Inside corners are cut back to limit paste volume.
- Printed at 1:1 with pad or at 10% reduction.



Radiused Inverted Home Plate (RIHP)

- Corners are rounded to eliminate tightness and protrusions.



2.6 SOLDER PASTE AND SQUEEGEE TYPES

Paste solder volume depends on the squeegee type used. For paste solder application, consider the following two squeegee types:

- Stainless steel
- Nylon

2.7 SOLDER PASTE MASK CONSIDERATIONS

- Ensure the solder mask is pulled away from the solder pad perimeter. The solder mask opening around the PCB pads may be as large as the spacing between the pads.
- Minimum solder mask width strongly depends on the PCB manufacturer's capabilities and the end-user's design guidelines. Consult with the PCB supplier about processing capabilities.
- Do not place PCB vias and traces near the package corners without using a solder mask. This step avoids potential shorting between exposed package PCB assembly features.

2.8 SOLDER PASTE PRINTING CONSIDERATIONS

- Follow the solder paste manufacturer's recommended guidelines to accommodate paste-specific characteristics.
- Solder paste is typically applied using an automated stencil printer capable of controlling the following key printing variables:
  - Squeegee pressure
  - Squeegee speed
  - Snap off distance
  - PCB separation distance
  - PCB separation speed
- Conduct post-print inspection and solder paste volume measurement. This step is critical for ensuring good print quality and uniform paste deposition.

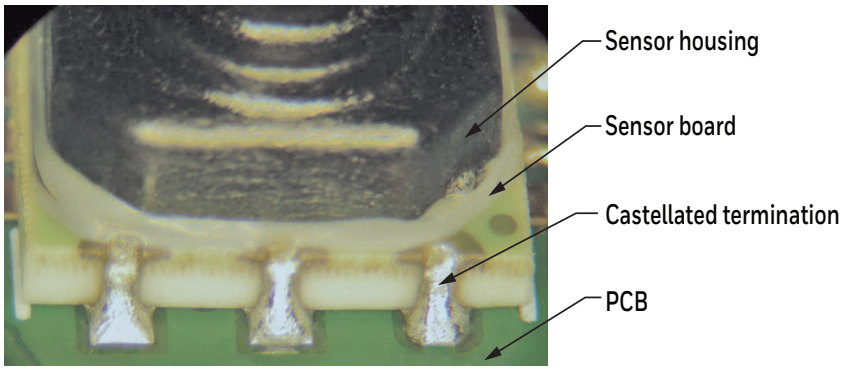
2.9 CASTELLATED TERMINATIONS

Castellated terminations, found on some Honeywell SMT leadless sensors, are plated through holes or vias located in the edges of the sensor substrate. Castellations are cut through to form a series of half holes which serve as pads intended to create a link between the sensor board and the PCB onto which the sensor is soldered. Solder connections between the sensor board and the PCB must meet the dimensional and solder fillet requirements defined in IPC-A-610 – Acceptability of Electronic Assemblies. The specific section of IPC-A-610 is Section 8.2.4 – Castellated Terminations. Figure 4 shows an example of a sensor using castellated terminations.

NOTICE

It is the responsibility of the solder paste manufacturer to develop the necessary process parameters to provide the required solder thickness and volume.

FIGURE 4. CASTELLATED TERMINATIONS



3.0 SMT COMPONENT PICK-AND-PLACE PROCESS

Current industry-standard, automated pick-and-place equipment should be able to support most lead-free solders. Ensure adherence to the following precautions:

- The pick-and-place machine is equipped with an optical recognition system (such as a vision system) for centering the sensor pads and the PCB assembly pads during the pick-and-place process.
- Conduct a placement accuracy study to determine compensation required to ensure proper pad-to-pad alignment.

4.0 SOLDER REFLOW PROCESS

Solder reflow process optimization is the most critical factor for ensuring successful, lead-free soldering that achieves high yield and long term solder joint reliability. Ensure that the development of an optimized thermal profile includes the following:

- Solder paste characteristics
- PCB size
- Sensor component density
- Larger/smaller component mix
- Sensor component peak temperature requirements

4.1 TEMPERATURE PROFILING

Perform temperature profiling on all new PCB designs by attaching thermocouples to the PCB assembly solder joints, on the top surface of the larger components, as well as at multiple locations on the PCB. This step ensures that all components are heated to a temperature above the minimum reflow temperatures and the smaller components do not exceed the maximum temperature limit.

4.2 REFLOW PROFILE GUIDELINES

Ensure the solder reflow profile follows the solder paste manufacturer's recommendation and the general JEDEC/IPC standard J-STD-20 guidelines.

4.3 REFLOW OVEN CONSIDERATIONS

For lead-free assembly, ensure the reflow oven is equipped with multiple heating zones and a gaseous nitrogen atmosphere. Ovens with multiple heating zones offer greater flexibility to optimize the reflow profile. A gaseous nitrogen atmosphere has been shown to improve lead solder wettability and to reduce the temperature gradient across the PCB assembly. This atmosphere type may also enhance the solder joint appearance by reducing oxidation effects. The vast majority of assemblers seek a solder paste that can be reflowed in air, so many lead-free solder paste chemistries are being developed with this in mind.

4.4 SENSOR REFLOW CONSIDERATIONS

For optimum performance, ensure adherence to the following precautions:

- The sensor is exposed to one reflow cycle only.
- The sensor is positioned with the port oriented upward, NOT downward.
- The sensor port is covered while being mounted to the PCB assembly.

4.5 POST REFLOW VISUAL INSPECTION

Lead-free solder joints are not as shiny as Sn/Pb solder joints. In addition, the solder fillet profile is generally not as great as with Sn/Pb solder joints. Honeywell recommends the end-user to:

- Train quality inspectors to distinguish the quality of lead-free solder joints after reflow.
- Train and certify all operators to follow IPC-610 (Acceptability of Electronic Assemblies).

4.6 SOLDER PAD VOIDS

Solder pad voiding is unavoidable and may be mitigated though several techniques of which fine tuning of the reflow profile is best. Defining allowable voiding is a very subjective process. Honeywell does not define the maximum allowable voiding of SMT solder pads.

NOTICE

It is the responsibility of the end-user to determine the allowable voiding through environmental testing.

5.0 SMT SENSOR SPECIAL HANDLING

5.1 PICK-AND-PLACE CONSIDERATIONS

Special handling must be considered when mounting an SMT sensor on a PCB assembly. Ensure adherence to the following precautions:

- Non-ported, cover-style sensors are picked up by the sides of the sensor cover.
- Ported, cover-style sensors are picked up by the port.
- Ported, cover-style sensors are not subjected to pick and place vacuum. The port is vented to atmosphere at all times.
- The vent hole on the bottom of gage sensors stays open and unobstructed. (An absolute pressure sensor does not require a vent hole.) This includes the use of ionically-clean adhesives where necessary (not recommended).
- Honeywell does not recommend the use of adhesives to secure the sensor; however, if the end-user does decide to use adhesives, ensure the vent hole at the bottom of gage sensors does not become blocked. Use ionically-clean adhesives that meet the following content guidelines:
  - Cl < 50 ppm
  - K < 50 ppm
  - Na < 20 ppm

5.2 SOLDERING PROCESSES THAT ARE NOT RECOMMENDED

- **Hand soldering:** Lead-free soldering requires an excessive amount of energy compared to lead containing solder alloys. The heat transfer to the solder joint is very critical and should not be attempted with a soldering iron. If the end-user does decide to use hand soldering, ensure the following:
  - Either hot air or a hot plate is used.
  - The sensor is hand soldered only one time. Sensor performance could degrade due to more than one de-soldering process and subsequent re-soldering of the sensor to the PCB assembly.
- **IR reflow:** Not recommended due to potential damage as result of radiation heat transfer.

5.3 SENSOR REWORK

Sensor rework is an unavoidable process defect as a result of SMT manufacturing. Honeywell recommends the end user to:

- Follow IPC-A-610 (Acceptability of Electronic Assemblies) guidelines when reworking a sensor assembly.
- Train and certify all rework operators to follow IPC rework standards. It is assumed that the end-user follows all good housekeeping practices and uses proper rework tools and personal protective equipment.

NOTICE

When rework is evident, Honeywell is not responsible for potential and foreign material contamination on the sensor and/or found in sensor cavities due to rework processes and workmanship.

5.4 PCB ASSEMBLY SPECIAL HANDLING

Packaging stresses associated with assembling a sensor to a next level PCB assembly should be addressed in the final assembly design. Ensure adherence to the following precautions:

- Use "keep out" areas at the end-of-line test probe locations. Avoid probe location around or directly on the opposite side of the sensor. Using test probes in the wrong locations may affect sensor output.
- Survey all end-of-line test systems to understand the maximum microstrain exerted on the sensor/PCB assembly. Honeywell recommends that strain on each assembly does not exceed 500 µstrain.
- Address sensor orientation sensitivity.
- Do not screw down or heat stake the PCB assembly near the sensor. Ensure screw and heat staking locations do not exert excessive force in, around, or directly on the opposite side of the sensor. Excessive screw or heat staking force may affect sensor output.
- Do not use ultrasonic cleaning. The frequencies used may damage wire bond interconnections.
- Consider using singulation processes such as PCB assembly depaneling/sawing with a sensor to understand the maximum strain exerted on the sensor/PCB assembly. Honeywell recommends that strain on each assembly does not exceed 500 µstrain.

Table 4 provides several diagnostic tools that may be used for process debug and final disposition of the SMT soldering process.

TABLE 4. SMT SOLDERING DIAGNOSTIC TOOLS

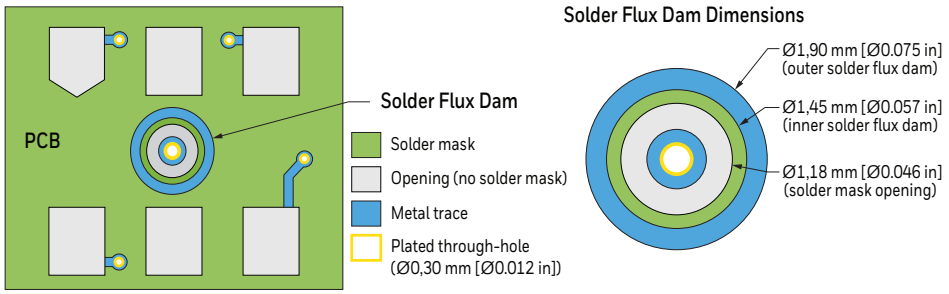
DIAGNOSTIC TOOL	PURPOSE
X-ray analysis	• Ensures proper solder spread under the sensor to the boundary of the pad • Addresses solder voiding
Solder joint micro-sectioning	Provides a method to inspect solder joint quality during process optimization; it is less suitable to production inspection due to process limitations
In-process 3D solder paste analysis	Monitors solder volume during the solder application process; 3D solder paste analysis real-time feedback may be attained using currently available equipment
Solder strength shear testing	Determines solder volume; analysis tools are available to provide feedback

5.5 PCB ASSEMBLY SPECIAL DESIGN FEATURE: SOLDER FLUX DAM

When mounting a sensor to a PCB assembly, consider using a solder flux dam to mitigate the risk of no-clean solder flux migration into the underside of the piezoresistive sense element. Figure 3 shows the basic feature geometry.

- If a different through-hole diameter is desired than the one shown, use the dimensions given in Figure 5 to proportion the new through-hole diameter.
- The recommended diameter for a gage sensor vent hole is 1,57 mm [0.062 in]. Ensure the PCB vent hole lines up directly below the sensor's vent hole and that it remains open and unobstructed.

FIGURE 5. SOLDER FLUX DAM GEOMETRY



6.0 TROUBLESHOOTING

See Appendices A through C for the following troubleshooting guidelines:

- **Table A:** Solder Paste Printing Process
- **Table B:** Solder Reflow Process
- **Table C:** Component Pick-and-Place Process

APPENDIX A: SOLDER PASTE PRINTING PROCESS TROUBLESHOOTING GUIDELINES

TABLE A. SOLDER PASTE PRINTING PROCESS TROUBLESHOOTING GUIDELINES

PROBLEM	PREVENTION	REMEDY
Excess solder paste	• Conduct operator training • Conduct stencil buy-off before production • Conduct first article solder paste height inspection after setup	• Use an Xbar-R chart to track solder paste height • Ensure a reflow oven thermal profile is part of the oven certification program • Reduce screen printer print gap • Reduce squeegee speed and/or force • Reduce stencil aperture by 10% • Investigate a different stencil aperture geometry such as the home plate design (See Section 3.5)
Solder balls	• Conduct operator training • Conduct stencil buy-off before production • Ensure proper solder paste storage temperature • Ensure proper solder paste thawing time • Ensure solder paste FIFO (First In/First Out) use • Conduct first article solder paste height inspection after setup	• Conduct 100% visual inspection • Measure solder paste height • Use an Xbar-R chart to track solder paste height • Reduce stencil aperture by 10% • Investigate a different stencil aperture geometry such as the home plate design (See Section 2.5)
Damaged stencil apertures	• Conduct operator training • Conduct stencil buy-off before production • Implement a periodic squeegee replacement program • Conduct first article solder paste height inspection after setup	• Conduct 100% visual inspection • Measure solder paste height • Use an Xbar-R chart to track solder paste height • Conduct stencil inspection at shift change
Pad-to-pad offset	• Conduct operator training • Conduct stencil buy-off before production • Conduct first article solder paste height inspection after setup	• Conduct 100% visual inspection • Supervision conducts first piece inspection and process sign-off • Readjust stencil position and pad registration • Check stencil tension • Conduct stencil inspection at shift change • Clean stencil every four hours
Solder smear	• Conduct operator training • Conduct stencil buy-off before production • Implement a periodic squeegee replacement program • Conduct first article solder paste height inspection after setup	• Conduct 100% visual inspection • Supervision conducts first piece inspection and process sign-off • Readjust stencil position and pad registration • Check stencil tension • Check stencil for excessive wear • Check squeegee for wear • Conduct stencil inspection at shift change • Clean stencil every four hours
Small areas of solder smaller than aperture/pad	• Conduct operator training • Conduct stencil buy-off before production • Conduct first article solder paste height inspection after setup • Implement periodic squeegee replacement program • Ensure stencil printer has real time squeegee pressure and speed control and feedback	• Conduct 100% visual inspection • Supervision conducts first piece inspection and process sign-off • Clean stencil every four hours • Add fresh solder paste to stencil • Adjust squeegee print speed

TABLE A. SOLDER PASTE PRINTING PROCESS TROUBLESHOOTING GUIDELINES (CONTINUED)		
PROBLEM	**PREVENTION**	**REMEDY**
Large areas of solder greater than aperture/pad	• Conduct operator training • Conduct stencil buy-off before production • Conduct first article solder paste height inspection after setup • Implement periodic squeegee replacement program	• Conduct 100% visual inspection • Supervision conducts first piece inspection and process sign-off • Reduce squeegee pressure • Ensure solder stencil printer has real time squeegee pressure feedback • Clean stencil every four hours • Inspect PCB for contamination • Inspect stencil at end of shift
Paste volume high/ solder deposition height too high	• Conduct operator training • Conduct stencil buy-off before production • Conduct first article solder paste height inspection after setup	• Conduct 100% visual inspection • Supervision conducts first piece inspection and process sign-off • Reduce squeegee speed • Reduce squeegee pressure • Reduce printer print gap • Clean stencil every four hours • Inspect stencil at end of shift
Solder slump	• Conduct operator training • Conduct stencil buy-off before production • Conduct first article solder paste height inspection after setup • Ensure proper solder paste storage temperature • Ensure proper solder paste thawing time • Ensure solder paste FIFO use • Ensure adherence to solder paste technical data sheet	• Conduct 100% visual inspection • Supervision conducts first piece inspection and process sign-off • Adjust squeegee speed • Reduce squeegee pressure • Ensure solder stencil printer has real time squeegee pressure and speed control feedback
Excess solder paste	• Conduct operator training • Conduct stencil buy-off before production • Conduct first article solder paste height inspection after setup • Ensure proper solder space storage temperature • Ensure proper solder paste thawing time • Ensure solder paste FIFO use	• Conduct 100% visual inspection • Supervision conducts first piece inspection and process sign-off • Conduct stencil inspection at shift change • Reduce stencil aperture size • Reduce stencil thickness
Excessive squeegee pressure/debris on PCB, damaged aperture, warped stencil	• Conduct operator training • Conduct stencil buy-off before production • Conduct first article solder paste height inspection after setup • Ensure periodic squeegee replacement	• Conduct 100% visual inspection • Supervision conducts first piece inspection and process sign-off • Reduce squeegee pressure • Ensure solder stencil printer has real time squeegee pressure feedback • Clean stencil and inspect PCB for contamination every four hours • Inspect stencil at end of shift

TABLE A. SOLDER PASTE PRINTING PROCESS TROUBLESHOOTING GUIDELINES (CONTINUED)		
PROBLEM	**PREVENTION**	**REMEDY**
Large variation in solder height	• Conduct operator training • Conduct stencil buy-off before production • Conduct first article solder paste height inspection after setup	• Conduct 100% visual inspection • Supervision conducts first piece inspection and process sign-off • Inspect stencil at end of shift • Measure solder paste height • Use an Xbar-R chart to track solder paste height • Adjust screen printer separation speed • Adjust squeegee speed. • Ensure solder stencil printer has real time squeegee pressure and speed control feedback
Low solder volume/ low solder height	• Conduct operator training • Conduct stencil buy-off before production • Conduct first article solder paste height inspection after setup • Implement periodic printer squeegee replacement program.	• Conduct 100% visual inspection • Supervision conducts first piece inspection and process sign-off • Inspect stencil at end of shift • Measure solder paste height • Use an Xbar-R chart to track solder paste height • Adjust screen printer separation speed • Adjust squeegee speed. • Increase stencil thickness. • Ensure solder stencil printer has real time squeegee pressure and speed control feedback
Ionic/non-ionic Contamination	• Conduct operator training • Ensure adherence to operator instructions • Ensure proper use of personal protective equipment • Ensure adherence to PCB specification for: - manufacturing - shipping - handling - cleanliness	• Ensure non-ionic cleanliness requirements of the PCB surface material is 10 µg/cm^2 as measured per IPC-TM-650 (2.3.38) • Ensure ionic cleanliness requirements are specified as <2.5 µg/in^2 NaCl per IPC TM-650 2.3.28 in the manufacturing specification
Solder flux not activated	• Conduct operator training • Ensure adherence to operator instructions • Ensure adherence to solder manufacturer technical data sheet	• Ensure a reflow oven thermal profile is part of the oven certification program • Ensure oven profile is above 150°C [302°F] for one to two minutes to completely activate solder flux
Component open circuit	• Conduct operator training • Ensure adherence to operator instructions • Ensure proper use of personal protective equipment • Ensure adherence to ANSI/ESD S20.20 (Electrostatic Discharge Certification)	• Supervision conducts first piece inspection and process sign-off • Implement a formal electrostatic discharge ESD management program to provide constant ESD monitoring and prevention; program should include ESD process surveys throughout the manufacturing environment

APPENDIX B: SOLDER REFLOW PROCESS TROUBLESHOOTING GUIDELINES

TABLE B. SOLDER REFLOW PROCESS TROUBLESHOOTING GUIDELINES

PROBLEM	PREVENTION	REMEDY
Cold solder joint	• Conduct operator training • Ensure adherence to operator instructions • Ensure adherence to manufacturer technical datasheet	• Supervision conducts first piece inspection and process sign-off • Ensure a reflow oven thermal profile is included in the oven certification program • Ensure solder exceeds liquidus temperature of 217°C to 221°C [423°F to 430°F]
Short circuit	• Conduct operator training • Ensure adherence to operator instructions • Ensure adherence to manufacturer technical datasheet	• Supervision conducts first piece inspection and process sign-off • Conduct In Circuit Testing (ICT) • Use an Xbar-R chart to track solder paste height • Ensure a reflow oven thermal profile is included in the oven certification program • Reduce stencil aperture by 10% • Investigate a different stencil aperture geometry such as the home plate design (see Section 2.5)
Ionic/non-ionic contamination	• Conduct operator training • Ensure adherence to operator instructions • Ensure proper use of personal protective equipment • Ensure adherence to PCB specification for: - manufacturing - shipping - handling - cleanliness	• Ensure non-ionic cleanliness requirements of the PCB surface material is 10 µg/cm^2 as measured per IPC-TM-650 (2.3.38) • Ensure ionic cleanliness requirements are specified as <2.5 µg/in^2 NaCl per IPC TM-650 2.3.28 in the manufacturing specification • Review PCB specification for: - shipping - handling - cleanliness
Component tombstone	• Conduct operator training • Ensure adherence to operator instructions • Ensure adherence to manufacturer technical datasheet	• Ensure a reflow oven thermal profile is included in the oven certification program • Slow down the oven profile ramp rate to reduce uneven thermal gradients • Conduct post reflow automated optical inspection
Component billboard	• Conduct operator training • Ensure adherence to operator instructions • Ensure adherence to manufacturer technical datasheet	• Ensure a reflow oven thermal profile is included in the oven certification program • Adjust component placement position • Conduct post reflow automated optical inspection
Solder flux not activated causing short circuit	• Conduct operator training • Ensure adherence to operator instructions • Ensure adherence to manufacturer technical datasheet	• Ensure a reflow oven thermal profile is included in the oven certification program • Ensure oven profile is above 150°C [302°F] for one to two minutes to completely activate solder flux
Component open circuit ESD	• Conduct operator training • Ensure adherence to operator instructions • Ensure adherence to manufacturer technical datasheet • Ensure proper use of personal protective equipment • Ensure adherence to ANSI/ESD S20.20 (Electrostatic Discharge Certification)	• Supervision conducts first piece inspection and process sign-off • Conduct In Circuit Testing (ICT) • Implement a formal electrostatic discharge ESD management program to provide constant ESD monitoring and prevention; program should include ESD process surveys throughout the manufacturing environment

APPENDIX C: COMPONENT PICK-AND-PLACE PROCESS TROUBLESHOOTING GUIDELINES

TABLE C. COMPONENT PICK-AND-PLACE PROCESS TROUBLESHOOTING GUIDELINES

PROBLEM	PREVENTION	REMEDY
Misplaced components	• Conduct operator training • Follow operator instructions • Ensure proper use of personal protective equipment	• Supervision conducts first piece inspection and process sign-off • Ensure pick-and-place machine is equipped with visual inspection to detect mis-picked components • Ensure pick-and-place machine aborts after three pick attempts
Component polarity incorrectly placed	• Conduct operator training • Ensure adherence to operator instructions • Ensure proper use of personal protective equipment • Ensure component polarity is controlled in tape and reel	• Supervision conducts first piece inspection and process sign-off • Ensure pick-and-place machine is equipped with visual inspection to detect mis-picked components • Ensure pick-and-place machine aborts after three pick attempts
Wrong component value	• Conduct operator training • Follow operator instructions • Ensure proper use of personal protective equipment • Ensure MRP (Material Requirements Planning) system/BOM (Bill of Materials) is linked with bar code reader on reel, and bar code on reel feeder	• Supervision conducts BOM walk, first piece inspection, and process sign-off • Ensure pick-and-place machine has programming aspects that dedicate feeders and components
Missing component	• Conduct operator training • Follow operator instructions • Ensure proper use of personal protective equipment • Clean vacuum tip periodically	• Supervision conducts first piece inspection and process sign-off • Implement a formal electrostatic discharge ESD management program to provide constant ESD monitoring and prevention; program should include ESD process surveys throughout the manufacturing environment

Reference: Shea, Chrys. "Stencil Design for Lead-Free SMT Assembly," Available: www.practicalcomponents.com. [Accessed Sept. 29, 2017].

WARRANTY/REMEDY

Honeywell warrants goods of its manufacture as being free of defective materials and faulty workmanship during the applicable warranty period. Honeywell's standard product warranty applies unless agreed to otherwise by Honeywell in writing; please refer to your order acknowledgement or consult your local sales office for specific warranty details. If warranted goods are returned to Honeywell during the period of coverage, Honeywell will repair or replace, at its option, without charge those items that Honeywell, in its sole discretion, finds defective. **The foregoing is buyer's sole remedy and is in lieu of all other warranties, expressed or implied, including those of merchantability and fitness for a particular purpose. In no event shall Honeywell be liable for consequential, special, or indirect damages.**

While Honeywell may provide application assistance personally, through our literature and the Honeywell web site, it is buyer's sole responsibility to determine the suitability of the product in the application.

Specifications may change without notice. The information we supply is believed to be accurate and reliable as of this writing. However, Honeywell assumes no responsibility for its use.

FOR MORE INFORMATION

Honeywell Sensing and Internet of Things services its customers through a worldwide network of sales offices and distributors. For application assistance, current specifications, pricing or the nearest Authorized Distributor, visit sensing.honeywell.com or call:

USA/Canada	+1 302 613 4491
Latin America	+1 305 805 8188
Europe	+44 1344 238258
Japan	+81 (0) 3-6730-7152
Singapore	+65 6355 2828
Greater China	+86 4006396841

Honeywell
Sensing and Internet of Things
830 East Arapaho Road
Richardson, TX 75081
sensing.honeywell.com

008329-2-EN | 2 | 07/20

THE FUTURE IS WHAT WE MAKE IT

Honeywell