Process Troubleshooting Guide

Selective Soldering Process Manual and Manufacturability Guideline

NOTICE

This is a Nordson SELECT publication that is protected by copyright. Original copyright date 2017. No part of this document may be photocopied, reproduced, or translated to another language without the prior written consent of Nordson SELECT. The information contained in this publication is subject to change without notice.

Originally published by and re-produced with permission of ITM Consulting, Springfield, TN, USA

Nordson SELECT

Table of Contents

Page

0.0 Introduction 4

1.0 Fundamentals of Through-Hole Soldering 4
- 1.1 Capillary Action 4
- 1.2 Through-Hole Vertical Fill 4
- 1.3 Solder Joint Formation 4
- 1.4 Vertical Force Model 5
- 1.5 Component Thermal Mass Differential 5
- 1.6 Process Parameters 5-6
- 1.7 Thermal Processing 6-7
- 1.8 Intermetallic Layer Formation 7

2.0 Solderability 8
- 2.1 Oxidation Layers 8
- 2.2 Surface Wetting 8
- 2.3 Zero Force Wetting Time 8
- 2.4 Bare Board Cleanliness 8
- 2.5 Post-Soldering Ionic Contamination Levels 8
- 2.6 Dross Abatement 9
- 2.7 Manual Soldering 9
- 2.8 Component Re-Tinning 9-10

3.0 Solder Alloys 10
- 3.1 Tin-Lead and Lead-Free Alloys 10
- 3.2 Surface Wetting Characteristics 10
- 3.3 Alloy Characteristics 10-11
- 3.4 Copper Dissolution 11
- 3.5 Melting Point and HMP Alloys 11-12
- 3.6 Solder Nozzle and Solder Pot Temperature Correlation 12

4.0 Flux Deposition and Flux Activation 12
- 4.1 Liquid Flux Chemistries 12-13
- 4.2 Thermal Aspects of Flux Activation 13
- 4.3 Preheat Temperature 14
- 4.4 Typical Thermal Profile 14
- 4.5 Time-Temperature Limitations 14
- 4.6 No-Clean Thermal Processing 14
- 4.7 Mitigation of Flux Residues 14-15

5.0 Thermal Profiling 15
- 5.1 Thermal Transfer Characteristics 15
- 5.2 Thermocouple Location Selection Criteria 15-16
- 5.3 Instrumentation Techniques 16
- 5.4 Preheat Methodologies 17

5.5 Preheat Selection 17
5.6 Sustained Preheat 17

6.0 Through-Hole Design Guidelines 17
6.1 Lead-to-Hole Aspect Ratio 17
6.2 Lead Projection 18
6.3 Lead Pitch 18
6.4 DFMA Guidelines 18-19
6.5 Adjacent Component Clearance 19-20
6.6 Critical Keep-Out Areas 20-21
6.7 Interlayer Construction 21
6.8 Ground Planes 21
6.9 Thermal Relief Design Rules 21-22

7.0 Quality Measurement 22
7.1 Solder Joint Inspection Criteria 22
7.2 Post-Soldering Inspection Protocols 22
7.3 Inspection Methodologies 22-23
7.4 Pareto Analysis of Defect Type 23-24
7.5 Defect Frequency and Location 24

8.0 Troubleshooting Guideline 25
8.1 Defect Condition and Root Cause Analysis 25-27
8.2 TH Solder Defect Cause and Effect Matrix 27
8.3 Defect Resolution 27-28
8.4 Prevention of Re-Occurrence 28-29

9.0 Process Optimization 29
9.1 Optimization of Process Parameters 29-30
9.2 DPMO and OFD Quality Measurement 30-31
9.3 DoE Methodologies 31-33
9.4 Validation Run 34
9.5 Defect Mapping Techniques 34

10.0 Preventative Maintenance 35
10.1 Maintenance Procedures, Practices and Frequency 35
10.2 Solder Nozzle Maintenance 35
10.3 Solder Alloy Contamination Levels 36

11.0 Appendix 37
Through-Hole Soldering Troubleshooting Guide .... Attachment

Nordson SELECT

Selective Soldering Process Manual and Manufacturability Guideline

0.0 Introduction

The purpose of this Selective Soldering Process Manual and Manufacturability Guideline document is to describe soldering variables to serve as guidance to enhance the flexibility, reliability and quality provided by selective soldering equipment. The guidelines expressed in this document are intended to be general in nature and are influenced by the design aspects of printed circuit boards, thermal mass of through-hole components, various flux chemistries, different solder alloys and individual selective soldering machine configurations.

Improving the solder quality and first pass yield of a given printed circuit board assembly entails a series of enhancements to the process, assembly, fabrication and design attributes affecting that particular board assembly. To this extent process materials, printed circuit boards and components must conform to industry standards and accepted best practices. Nordson SELECT is therefore not responsible for any non-conformance of these materials which are outside of their control.

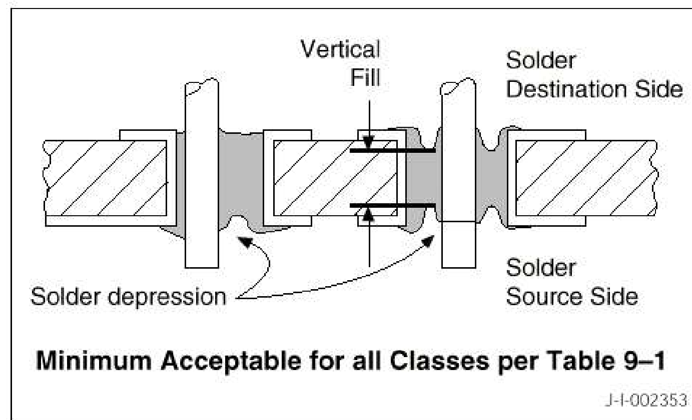
1.0 Fundamentals of Through-Hole Soldering

1.1 Capillary Action

Capillary action, or the pneumonia whereby liquidous solder rises in a column formed by a through-hole (TH) component and the inside walls of a plated through-hole (PTH), is directly dependent upon surface tension and wetting action of the liquidous solder as well as the wetting action of the plated surfaces.

1.2 Through-Hole Vertical Fill

Solder joints for through-hole (TH) components inserted into plated through-holes (PTH) must provide evidence of good wetting and must meet the requirements of a positive fillet on both the solder source side and the solder destination side as shown below.



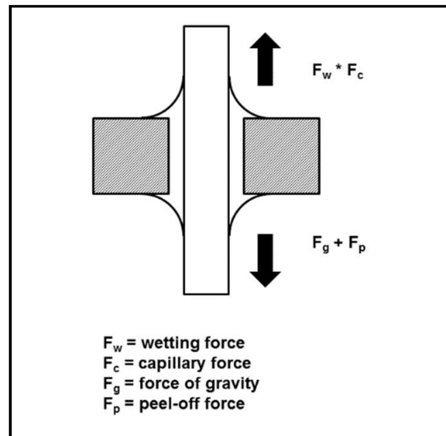
Through-Hole Vertical Fill

1.3 Solder Joint Formation

Wetting is the essential prerequisite for good soldering. Wetting means that a specific interaction has taken place between the liquidous solder and the solderable surfaces to be soldered. Wetting is possible only if the solder can come in immediate contact with the metallic surfaces of the through-hole (TH) component, plated through-holes (PTH) and circuit board pads. The extent to which liquidous solder will spread across a surface, or flow into a gap between two or more surfaces, depends upon its surface tension.

1.4 Vertical Force Model

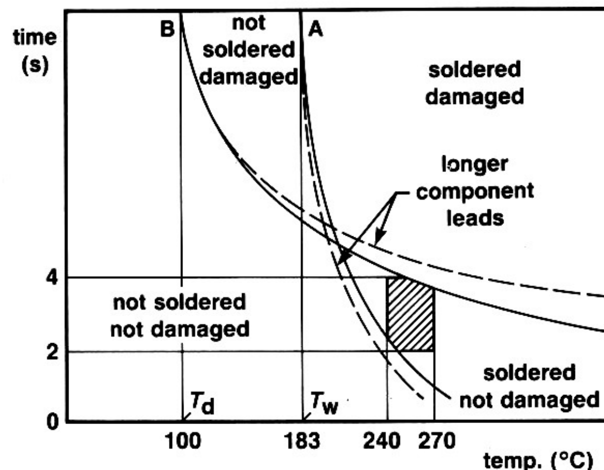
The ability of liquidous solder to fill a plated through-hole (PTH) and form a positive topside fillet on the solder destination side, is shown in the vertical fill force model below whereby a positive topside fillet will be formed when the product of the wetting force (F_w) times the capillary force (F_c) is greater than the force of gravity (F_g) plus the peel-off force (F_p) of the mechanical action of de-bridging the remaining solder on the solder source side.



Vertical Fill Force Model

1.5 Component Thermal Mass Differential

It is essential that all through-hole (TH) component leads, plated through-holes (PTH) and circuit board pads to be soldered must reach wetting temperature within a short period of time and in a uniform manner to promote uniform capillary action and assure vertical hole fill. The different thermal mass of various through-hole (TH) components, and/or the heat sinking effect of these components directly interconnected to a ground plane, will result in thermal mass differentials across the circuit board as shown below. Conversely temperature sensitive through-hole (TH) components can be damaged if their internal threshold temperature boundaries are exceeded during the soldering process.

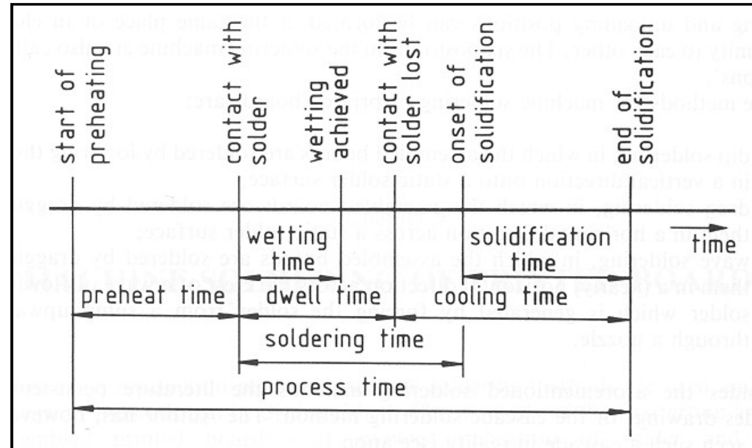


Effects of Thermal Mass Differential

1.6 Process Parameters

Selective soldering is a form of soldering that involves the application of heat and solder simultaneously to the through-hole (TH) components and the localized areas of the board to be soldered. Thermal considerations regarding the application of heat result in time-temperature boundaries consisting in an area of safe operation. These boundaries depend

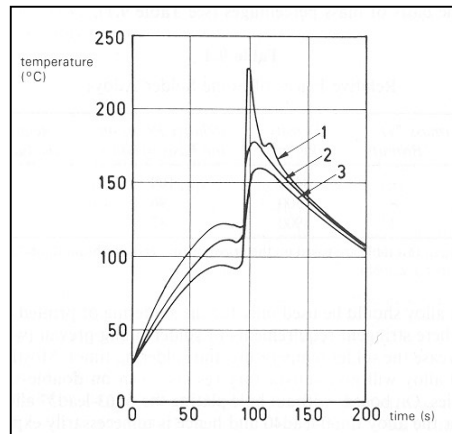
upon the particular product, but for most tin-lead (SnPb) liquidous solder processes the temperature of the solder bath can range between 280-300°C and the dwell time for liquidous solder contact is typically between 2-4 seconds. Process boundaries differ for lead-free solder alloys with most common tin-silver-copper (SAC305) can range between 290-310°C and the dwell times for liquidous solder contact 0.5-1.5 seconds longer. Regardless of the solder alloy, a typical flow diagram for liquidous flow soldering must take place as shown below.



Flow Diagram of Typical Liquidous Solder Process

1.7 Thermal Processing

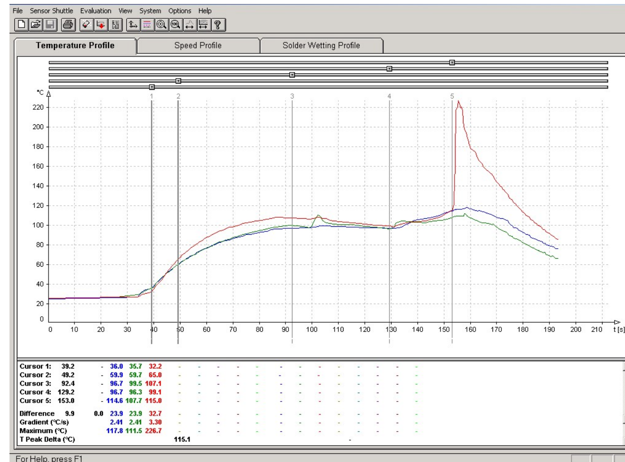
The preheat temperature, which is the temperature assumed by the circuit board at the end of the preheating period, is generally specified for the proper conditioning of a specific flux type. The dwell time, or solder contact time, and the soldering temperature determine the total heat flow to the soldering area. The time-temperature profile of the circuit board is affected by the thermal mass differential of the components and circuit board and the rate of heat dissipation. A typical flow solder thermal profile shows the temperature in a plated through-hole (PTH) (1), a topside trace (2) and on the topside of the circuit board (3) as shown below.



Time-Temperature Profile when in contact with Solder Nozzle

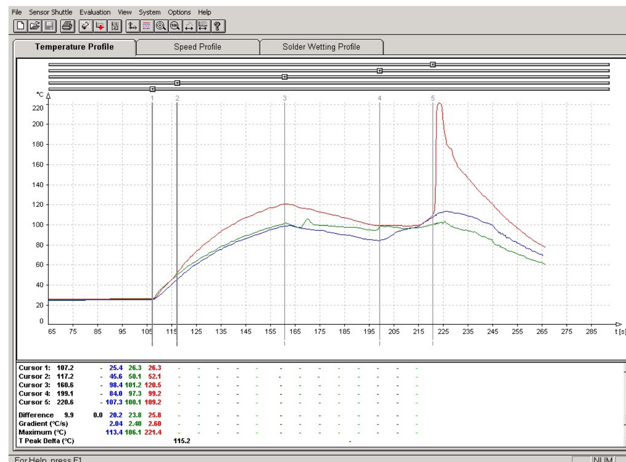
Flow solder process parameters are subject to a complex interaction of several factors, not only determined by the particular type of selective soldering machine, but also by the board to be soldered. Short wetting times are generally required to avoid component damage. Flux is used to raise the energy level and to promote wetting of the surfaces to be soldered. Assuring the presence of flux, correctly activated at the proper time and temperature and dried

to the correct viscosity so that it remains in contact with the solder surfaces until the assembly leaves the solder nozzle, is critical to good solder quality as shown below.



PCB Temperature during Typical Selective Soldering Thermal Profile (shown in Blue and Green)

Another important criterion is the dwell time of the board with the solder nozzle which is affected by the shape and contact length of the solder nozzle as shown below. Assuming that the circuit board is parallel to the solder nozzle, board-to-nozzle interaction has three distinct and interacting factors: dwell time, immersion depth and contact length. There is a direct interrelationship between immersion depth and contact length in that the contact length directly affects dwell time based on: dwell time = contact length ÷ traverse speed.



PCB Contact with Selective Soldering Nozzle (shown in Red)

1.8 Intermetallic Layer Formation

In order to form quality through-hole (TH) solder joints, the selective soldering process must: 1) raise the temperature of the base metals of the surfaces to be soldered to allow sufficient wetting, 2) provide adequate contact time for capillary action to occur, and 3) provide adequate thermal energy to create the formation of an intermetallic layer.

Proper intermetallic formation will result in a copper-tin (CuSn) intermetallic between the solder alloy and the copper base metal of the through-hole (TH) component lead as well as between the solder alloy and the copper base metal of the plated through-hole (PTH). An ideal intermetallic thickness is between 1.0-1.5 microns, however is should be noted that the thickness of the intermetallic layer increases with variations in thermal processing and cooling rates and when subjected to the effects of highly accelerated life (HAL) testing.

Nordson SELECT

2.0 Solderability

2.1 Oxidation Layers

Minimum standards for solderability of printed circuit boards (PCBs) and components must be met by vendors and maintained during in-house handling. Printed circuit boards and components should be kept in sealed packages to minimize contact with air and moisture during storage and should be used as soon as possible after receipt on a first-in, first-out (FIFO) basis to minimize the effects of oxidation upon the solderable surfaces.

2.2 Surface Wetting

Flow soldering can be defined as the formation of a metallurgical bond between two metal surfaces using a low melting point molten solder alloy that is quenched and solidified. The solder alloy must have a melting point below the melting point of the materials in the surfaces to be soldered. Solderability is the ability of these materials to be wetted by the solder and while wetting is taking place, the metallurgical bond known as an intermetallic compound is formed.

2.3 Zero Force Wetting Time

Through-hole (TH) flow soldering process parameters are subject to a complex interaction of several factors, determined more so by the board to be soldered and the solder alloy used then by a particular type of selective soldering machine. Short wetting times are generally preferred to minimize thermal damage to components. Unfortunately, some lead-free solder alloys exhibit elongated wetting times in comparison to more commonly used solder alloys based on standard laboratory testing as can be seen from the chart below.

Zero Force Wetting Time for Tin-Lead and Common Lead-Free Solder Alloys

Solder Temperature	Sn63Pb37 (183°C MP)	Sn96.5Ag3.0Cu0.5 (217-220°C MP)	Sn99.3.0Cu0.7 (227°C MP)
240°C	0.12 sec	0.72 sec	1.00 sec
250°C	0.11 sec	0.37 sec	0.87 sec
260°C	0.10 sec	0.22 sec	0.48 sec
270°C	0.08 sec	0.20 sec	0.32 sec

Selective soldering traverse speeds should be decreased to increase dwell time and contact with individual through-hole (TH) solder joints to compensate for this condition when using lead-free alloys with elongated wetting times.

2.4 Bare Board Cleanliness

Printed circuit boards (PCBs) as received from a board fabrication vendor should be tested for ionic and other contaminates that could adversely effect solderability. As a general rule, solderability testing should be performed to ensure that bare board cleanliness does not exceed 1.0µg/in^2 (0.15µg/cm^2) of sodium chloride (NaCl). At a bare minimum, one-time solderability testing should be conducted to establish a baseline consisting of: 1) PCBs as received from the board fabrication vendor, 2) PCBAs after double-sided SMT reflow, and 3) PCBAs after through-hole (TH) selective soldering.

2.5 Post-Soldering Ionic Contamination Levels

As a general guideline, the post-soldering cleanliness of printed circuit board assemblies (PCBAs) should not exceed 3.0-7.0µg/in^2 (0.45-1.1µg/cm^2) for sodium chloride (NaCl) as well as chlorides and bromides following all soldering processing.

2.6 Dross Abatement

Periodic cleaning of the solder pots along with routine dross removal is required to mitigate eliminate dross entrapment along with the accumulation of solder contaminates. Several dross abatement practices should be followed as shown below.

Recommended Dross Abatement Practices

Practice	Impact and Effect
Running Time	Reduce solder pot running time to minimize oxidation build-up
Wave Height	Reducing wave height to a minimum reduces dross generation
Nitrogen Inerting	Always maintain nitrogen curtain around solder nozzle and solder pot
Daily Dross Removal	Perform daily or once per shift surface dross removal of solder pots
Monthly Dross Removal	Perform monthly removal of solder pumps and clean impellers
Dross Inhibitor	The use of a dross inhibitor additive is not recommended

2.7 Manual Soldering

Very often hand soldering is performed on printed circuit board assemblies (PCBAs) after the selective soldering process to solder in-place topside through-hole (TH) components or other odd-form components. Whenever this is done a review should be made to ensure that these hand soldering operations are not in violation of the 'Seven Deadly Sins of Hand Soldering' – 1) excess solder tip pressure, 2) improper heat bridge, 3) incorrect tip size, 4) excessive temperature, 5) improper use of flux. 6) transfer soldering, and 7) unnecessary rework and repair as shown below.

The Seven Deadly Sins of Hand Soldering

Deficiency	Typical Causes and Corrective Action
Excess Solder Tip Pressure	Pressing harder on a solder pad does not transfer more heat – the additional pressure results in cracking and pitting of iron plating and hastens tip degradation
Improper Heat Bridge	Incorrectly applying cored solder wire to solder iron tip rather than solder pad and/or excessively fast hand soldering can result in cold solder joints
Incorrect Tip Size	Solder iron tips should be selected that are no larger than one-half the printed circuit board solder pad – different solder iron tip sizes should be used as required
Excessive Temperature	The use of excessive solder iron set point temperatures can result in de-wetting and thermal damage to circuit board pads and/or thermally sensitive components
Improper use of Flux	The use of additional liquid flux in no-clean hand soldering should be minimized since the solder iron heat source is highly localized and un-activated flux residues can result
Transfer Soldering	Incorrectly applying cored solder wire to the solder iron tip and excessively fast attempting to transfer the solder to several solder joints in a row can result in cold solder joints
Unnecessary Rework and Repair	Graphic-based workmanship standards should be used to diminish the subjective nature of manual solder joint inspection – operators should receive IPC certified hand soldering training

2.8 Component Re-Tinning

If through-hole (TH) component leads must be re-tinned prior to the selective soldering operation, it is recommended that the re-tinning process be carried out using a lead tinning machine or system utilizing controlled flux application, dynamic solder pot or dual pots, built-in dross skimmer, nitrogen inerting and defined process control. A defined process of this

type is suggested verses the use of manual dipping into static solder pots to reduce surface contamination, minimize non-wetting issues and enhancing solderability.

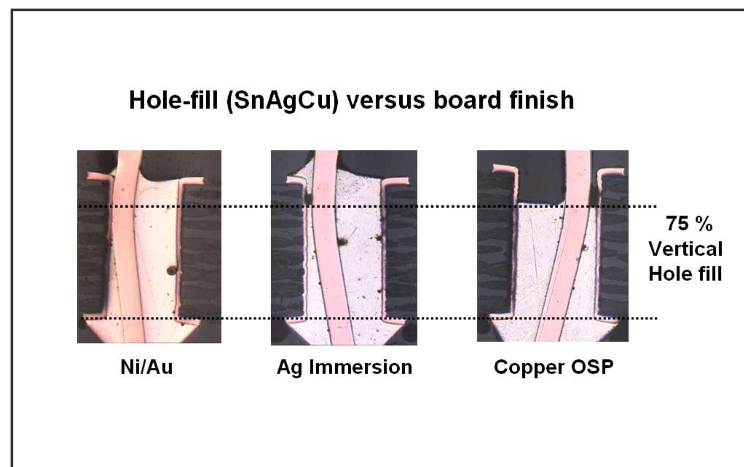
3.0 Solder Alloys

3.1 Tin-Lead and Lead-Free Alloys

The majority of lead-free solder alloys display decreased wetting characteristics and slower wetting times in comparison to tin-lead (SnPb) solders. Additionally, flow characteristics of lead-free alloys are generally more viscous. Lead-free alloys have melting points that are generally 30-40°C higher than SnPb solders requiring higher processing temperatures which places greater demand on flux performance. The use of a higher activity flux should be considered, however, the volume of flux applied to a board and the uniformity of deposition is essential. Increased preheat is generally required because of the higher melting point of lead-free alloys. Solder pot temperature and solder contact time must also be increased, but at the same time controlled to avoid thermal shocking, or excessive thermal differential upon coming in contact with the solder nozzle.

3.2 Surface Wetting Characteristics

The wetting ability of commonly used lead-free solder alloys such as Sn96.5Ag3.0Cu0.5 is greatly affected by the surface finish of the printed circuit board assembly (PCBA). An example of this is shown below where the ability of Sn96.5Ag3.0Cu0.5 solder alloy to fill a plated through-hole (PTH) is influenced by nickel-gold (NiAu), silver (Ag) immersion and organic surface protection (OSP) over bare cooper (Cu).



Effects of Various Board Finishes upon Through-Hole (TH) Wetting

3.3 Alloy Characteristics

Because of the increased tin content, lead-free alloys oxidize at a more rapid rate when the solder is in liquidous state as compared to tin-lead (SnPb) solder. Due to the higher tin content, tin oxide consisting of tin-oxygen (SnO) and (SnO^2), forms at a faster rate combined with higher processing temperatures to result in more oxidation and dross buildup.

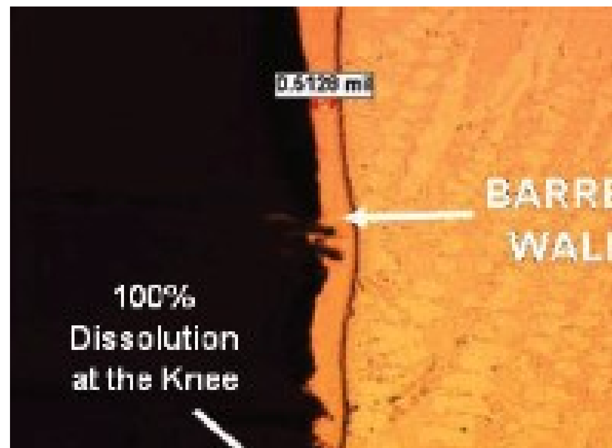
Characteristics of Common Lead-Free and Tin-Lead Solder Alloys

Solder Alloy	Melting Point	Solder Pot Temperature	Tin Content	Density
SnCu	227°C	300-330°C	99.3%	7.29
SnAg	221°C	290-320°C	96.5%	7.44
SnAgCu	217°C	290-320°C	96.5%	7.38
SnCuNi	227°C	290-320°C	99.4%	7.40
SnPb	183°C	250-270°C	63.0%	8.80

3.4 Copper Dissolution

The phenomenon of copper dissolution with lead-free through-hole (TH) soldering is a major quality concern that has undergone major investigation. The erosion of copper (Cu) from the printed circuit board (PCB) was present with conventional tin-lead (SnPb) solder but to a much lesser degree since the lead within the eutectic alloy functioned as an inhibitor. There is a much greater tendency for copper erosion with lead-free alloys since these alloys dissolve as much as two to four times the amount of copper as tin-lead solder. The reason comes down to the sluggish wetting behavior of lead-free solder alloys and the elongated time and temperature process window that is required for lead-free solder alloys because of the difference in their wetting and flow characteristics.

The presence of copper erosion can be seen across the entire exposed copper surfaces of a printed circuit board (PCB) but the effects of copper dissolution are most severe adjacent to the knee of the plated through-hole (PTH) and pad as shown below. It should ne noted that the effects of copper erosion are prevalent with all forms of lead-free flow soldering including wave soldering and selective soldering, but are much more pronounced during the post-wave or post-selective rework and repair process.



Cross Section of Copper Dissolution in Plated Through-Hole (PTH)

3.5 Melting Point and HMP Alloys

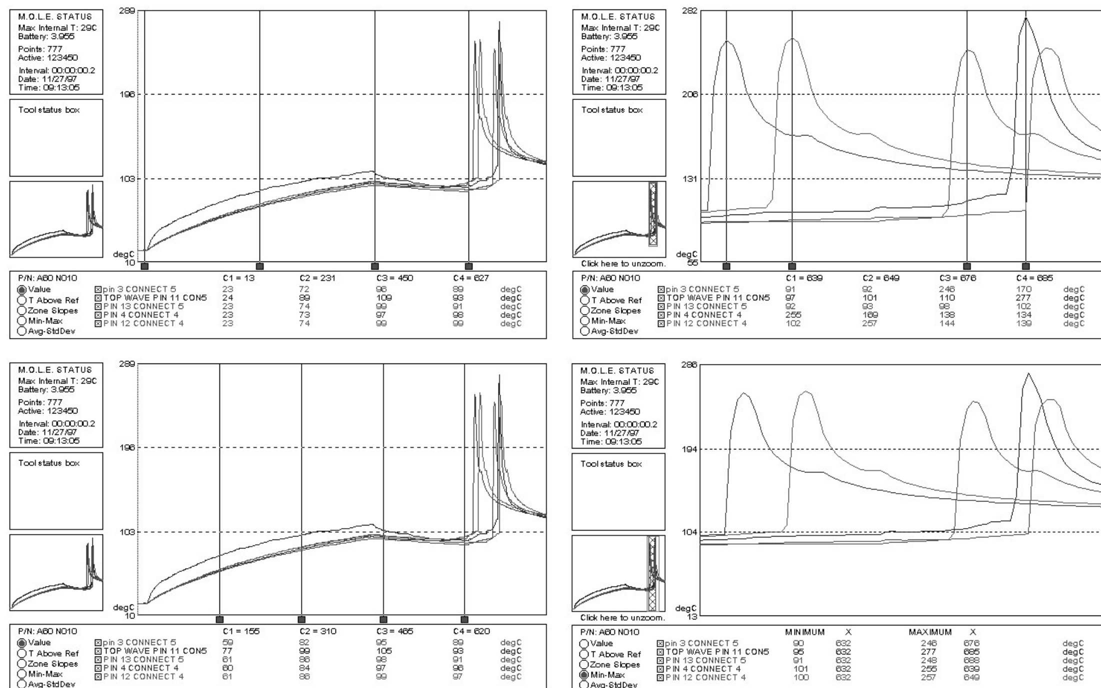
Some of the more commonly specified lead-free solder alloys are shown in the table below together with tin-lead solder. The selection of a particular lead-free solder alloy is often determined by the application as well as the physical properties of the alloy. Several other lead-free solders are also available as well as high melting point (HMP) solder alloys. HMP solder alloys such as Sn05Pb93.5Ag1.5, Sn95Sb05, Sn96Ag04 and Sn96.5Ag3.5, generally exhibit elongated wetting times as well as poor fluidity which requires single pass soldering of dual row connectors and other TH components in order to ensure adequate transfer of thermal energy into the solder interconnections.

Melting Range of Common Lead-Free and Tin-Lead Solder Alloys

Solder Alloy	Melting Range	Available in Wire	Available in Bar
Sn96.5Ag3.5	221°C	x	x
Sn96Ag04	221-229°C	x	x
Sn95Ag05	221-245°C	x	x
Sn99.3Cu0.7	227°C	x	x
Sn96.5Ag3.0Cu0.5	217-220°C	x	x
Sn96.5Ag3.8Cu0.7	217-221°C	x	x
Sn99.4Cu0.6Ni0.05	227°C	x	x
Sn63Pb37	183°C	x	x

3.6 Solder Nozzle and Solder Pot Temperature Correlation

The correlation between solder joint temperature (T_1) and solder pot temperature (T_2) for the selective soldering process should be monitored. The measurements shown below are an example that verifies that: a) proper flux activation has been achieved, b) the differential between the bottom-side temperature of the board prior to solder contact and the temperature during contact with liquidous solder does not exceed 100°C for tin-lead or does not exceed 140°C for lead-free to reduce thermal shocking of SMT or TH components, c) solder joint temperatures are within a range of 245-250°C for tin-lead or 255-260°C for lead-free to ensure optimum topside fillet formation, and d) individual solder joints within a specific component are within ± 1°C for consistent wetting.



Solder Joint Thermal Profiles during Selective Soldering Process for Tin-Lead Solder Alloy

4.0 Flux Deposition and Flux Activation

4.1 Liquid Flux Chemistries

The purpose of liquid flux during the selective soldering process is to remove surface oxides and provide an active base metal for the solder wetting to take place. Oxidation removal occurs through a chemical action which varies with the type of flux employed. Fluxes are

limited in their cleaning ability and are generally not capable of removing heavy or thick oxides, surface oils or organic coating as well as other types of surface contaminants. A separate pre-cleaning operation is imperative if such contaminants are present due to inadequate fabrication, storage or handling practices. In general, liquid fluxes can be classified as shown below.

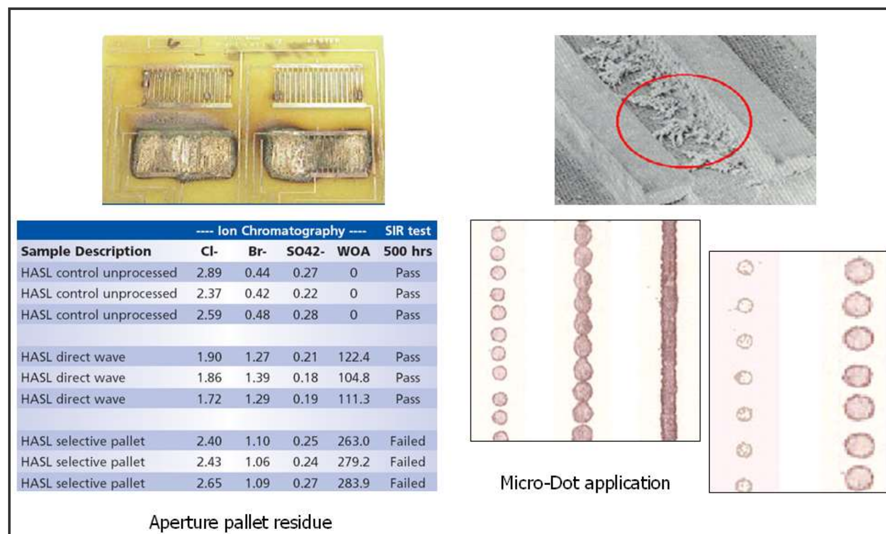
Common Liquid Flux Types and General Flux Characteristics

Flux Type	Activity Level	Residue	Comments
Rosin-Based Type R	Low	Non-corrosive	Rosin encapsulates residues rendering them benign
Mildly Activated RMA	Mild	Non-corrosive	Rosin encapsulates residues rendering them benign
Type RA	Medium	Residues are corrosive, removal is mandatory	Post-soldering cleaning is required to prevent corrosion
No-Clean, Low Solids	Medium	Non-corrosive	Process must be controlled to ensure benign residues
Organic Acid (OA)	Medium to high	Residues are corrosive, removal is mandatory	High activity level requires cleaning to prevent corrosion
Inorganic Acid	High	Residues are corrosive, removal is mandatory	Not generally used for electronic applications

4.2 Thermal Aspects of Flux Activation

In order to form solder interconnections, a liquid flux must be present and active to clean and protect the solderable surfaces when the printed circuit board assembly (PCBA) enters the selective soldering nozzle. Very high melting point solder alloys require robust fluxes that can survive the corresponding high preheat temperatures. High process temperatures can cause some flux residues to darken or become more difficult to remove after the selective soldering process is completed. Surface insulation resistance (SIR) testing should be performed on printed circuit board assemblies (PCBAs) that utilize no-clean fluxes to determine the relative activity of post-soldering residues.

A distinctive advantage of selective soldering over wave soldering using aperture pallets is that flux residues cannot become entrapped under openings in the aperture pallets resulting in high SIR readings as shown below.



	---- Ion Chromatography ----				SIR test
Sample Description	Cl-	Br-	SO42-	WOA	500 hrs
HASL control unprocessed	2.89	0.44	0.27	0	Pass
HASL control unprocessed	2.37	0.42	0.22	0	Pass
HASL control unprocessed	2.59	0.48	0.28	0	Pass
HASL direct wave	1.90	1.27	0.21	122.4	Pass
HASL direct wave	1.86	1.39	0.18	104.8	Pass
HASL direct wave	1.72	1.29	0.19	111.3	Pass
HASL selective pallet	2.40	1.10	0.25	263.0	Failed
HASL selective pallet	2.43	1.06	0.24	279.2	Failed
HASL selective pallet	2.65	1.09	0.27	283.9	Failed

Examples of Aperture Pallet and Selective Soldering Flux Deposition

4.3 Preheat Temperature

The preheat temperature, which is the temperature assumed by the circuit board at the end of the preheating period, is generally specified for the proper conditioning of a specific flux type. The dwell time, or solder contact time, and the soldering temperature determine the total heat flow to the soldering area. The time-temperature profile of the circuit board is affected by the thermal mass differential of the components and circuit board and the rate of heat dissipation.

4.4 Typical Thermal Profile

Liquidous flow soldering involves the application of heat and solder simultaneously to the through-hole (TH) components and the localized areas of the printed circuit board assembly (PCBA) to be soldered. Preheating of the printed circuit board assembly (PCBA) is required to ensure proper activation of the liquid flux chemistry. However thermal considerations regarding the application of heat result in time-temperature boundaries consisting in an area of safe operation must be maintained to avoid excessive overheating of temperature sensitive through-hole (TH) components.

4.5 Time-Temperature Limitations

It is essential that all through-hole (TH) component leads and printed circuit board (PCB) pads to be soldered reach wetting temperature within a short period of time and in a uniform manner to promote uniform capillary action and assure maximum vertical hole fill. Through-hole (TH) components of different thermal mass or the heat sinking effect of through-hole (TH) components directly interconnected to ground planes will result in mass differentials across the printed circuit board assembly (PCBA) and must be taken into consideration during the pre-heating cycle.

4.6 No-Clean Thermal Processing

Most no-clean fluxes have active ingredients that typically are mild organic acids in low concentration. In contrast to some high-solids rosin fluxes that have an 'activation temperature', above which the flux becomes active and below which the flux is inactive, the active ingredients in no-clean fluxes are active from the time of application until they are consumed by reaction or volatilized by heat.

Proper processing is the key to ensuring that the level of active residue is within acceptable standards. Even properly processed no-clean fluxes may leave an excessive amount of residue to be unsuitable for certain highly critical applications. Using no-clean fluxes with soldering processes that are not adequately controlled can result in flux residues that do not reliably meet required standards. Only a certain amount of active ingredient will be consumed in the reaction to remove the oxides from the metal surfaces, the bulk of the remaining active ingredients must therefore be 'burned off' by the heat during the soldering process. Therefore, it is important to carefully control both the amount of active ingredients applied and the heat during processing.

Controlling the amount of active ingredients applied requires monitoring the amount of flux being applied, the flux concentration and acid number, the uniformity of application, or deposition, as well as thermal processing characteristics including preheat temperature, solder pot temperature and dwell time.

4.7 Mitigation of Flux Residues

Since the preheat temperature should generally be lower than the thermal breakdown points of the common active ingredients in the no-clean flux, preheat temperature tends to be more important to solderability than it is to reduction of post-soldering ionic flux residues. However,

poor preheating can affect residue levels indirectly since a common first response to poor solderability is to increase the amount of flux, which can lead to residue problems.

During selective soldering, the heat from the solder pot is the highest temperature that will be seen by the board. Solder pot temperature has a greater effect than preheat temperature on reducing post-soldering residues. Since the solder is in intimate contact with the board when it passes over the solder nozzle, increasing dwell time not only allows the board to see the solder pot temperature for a longer period of time, but also helps to remove excessive flux through physical as well as thermal means.

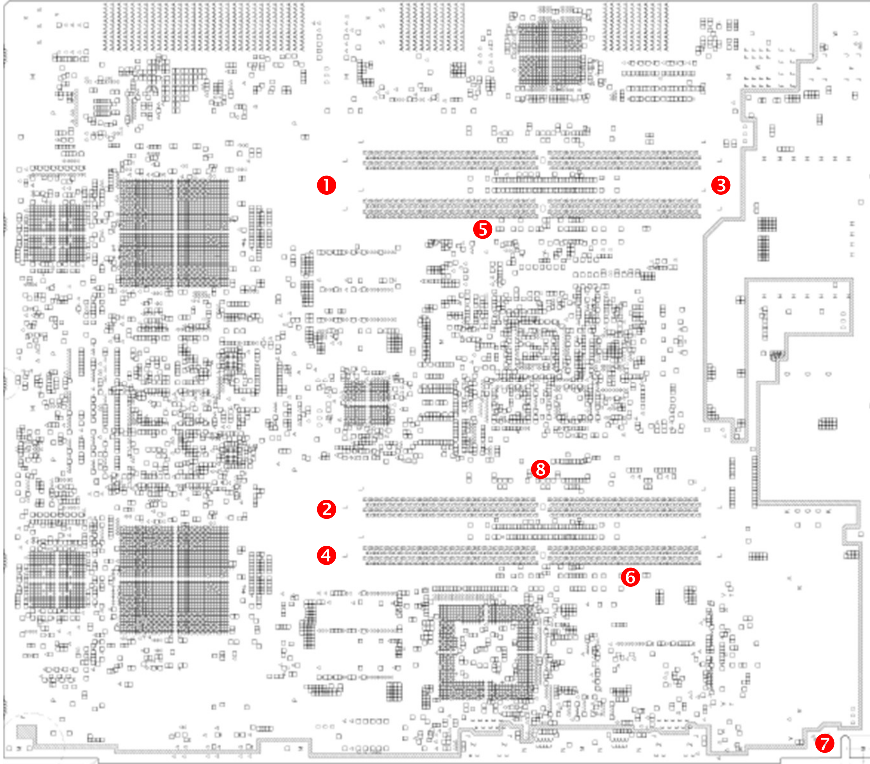
5.0 Thermal Profiling

5.1 Thermal Transfer Characteristics

The transfer of thermal energy into the solder joints of a printed circuit board assembly (PCBA) is facilitated by a combination of conduction, convection and radiation. Adequate contact between the bottom surface of the PCBA with liquidous solder for the recommended contact time and at the recommended solder temperature is essential to ensure board impingement and to facilitate the necessary heat transfer to form sufficient destination side solder joints. In addition to this conductive thermal energy component, supplemental energy in the form of either convection or radiation preheat is required to properly condition the liquid flux and raise the temperature of the PCBA prior to contact with liquidous solder.

5.2 Thermocouple Location Selection Criteria

When selecting locations to mount thermocouples onto a printed circuit board assembly (PCBA) attention should be taken to select locations that will: 1) monitor the flux activation window during preheat, 2) monitor flux survivability during the soldering process, 3) monitor solder joint formation temperature of critical through-hole (TH) components, 4) establish the solder nozzle temperature at the point of board impingement, and 5) monitor the temperature of critical interlayer or critical through-hole (TH) components were defects may be occurring. An example of typical thermocouple locations is shown below.



Position of thermocouple locations used to monitor board temperature, solder joint temperature and solder nozzle temperature

Typical Thermocouple Location Selection Criterion

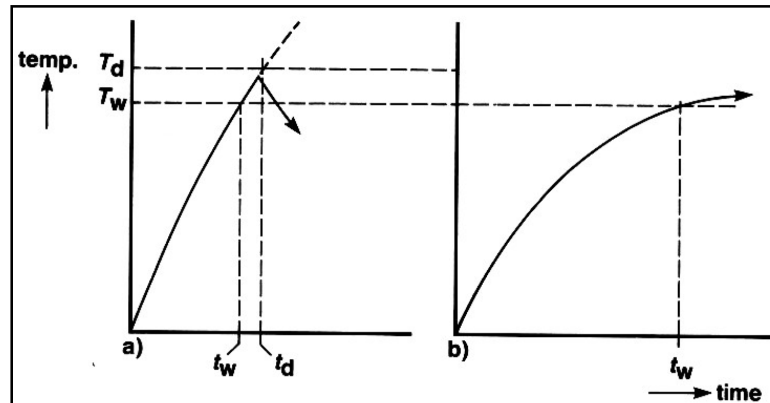
Locator	Thermocouple Location	Purpose
1	Top PCB surface adjacent to critical component	Monitor flux activation window
2	Top PCB surface adjacent to critical component	Monitor flux activation window
3	Bottom PCB surface adjacent to critical component	Monitor flux survivability
4	Bottom PCB surface adjacent to critical component	Monitor flux survivability
5	Pin 156 of critical component	Monitor solder joint formation temperature
6	Pin 216 of critical component	Monitor solder joint formation temperature
7	Solder immersion thru vacant Q1 hole	Monitor solder nozzle temperature
8	Drilled down to layer 20 adjacent to critical component	Monitor critical interlayer temperature

5.3 Instrumentation Techniques

In order to ensure that complete transfer of thermal energy is being monitored and controlled during the selective soldering process, the following instrumentation techniques should be used: 1) a previously soldered printed circuit board assembly (PCBA) fully populated with SMT and through-hole (TH) components should be used, 2) select critical points for thermocouple attachment and points at through-hole (TH) solder joints soldered during the selective soldering process, 3) remove as much existing solder at these points as possible using solder wick, 4) use 30 AWG type K thermocouples, 5) use liquid flux and high temperature (Sn95/Pb5) solder or conductive epoxy to mount thermocouples, 6) use Kapton tape for stress relief on thermocouple wires, 7) do not use Kapton or metal tape to attach thermocouples, and 8) do not use temperature labels to profile a PCBA.

5.4 Preheat Methodologies

All preheat methods available for selective soldering can be categorized as either an excessive heating method or an asymptotic heating method. An excessive heating method in one in which the heat source is at a much higher temperature than the melting point of the solder and can potentially result in component damage or incomplete solder joints if the excessive heat is applied for too short of a time duration and/or at a thermally unsafe gradient. An asymptotic heating method is one in which the heat source is slightly higher than the solder melting point. The asymptotic heating method minimizes thermal shocking and results in an equilibrium heating of components with a greater mass differential or for those through-hole (TH) components tied directly to ground planes as shown below.



Excessive and Asymptotic Heating Methods

5.5 Preheat Selection

Several different preheat methods are available for selective soldering including latent heating, infrared (IR) heating, convection heating and combination IR/convection heating. Latent heating is the heat radiated from the solder pot and solder nozzle. Medium wave length IR heating has a rapid response time and is flexible enough for most types of fluxes. Convection heating exhibits a lower gradient and heat transfer rate while exhibiting more uniformity. Combination IR/convection heating methods are beneficial for high thermal mass applications with either high thermal mass through-hole (TH) components, multi-layer printed circuit board assemblies (PCBAs) and/or assemblies with heavy ground planes.

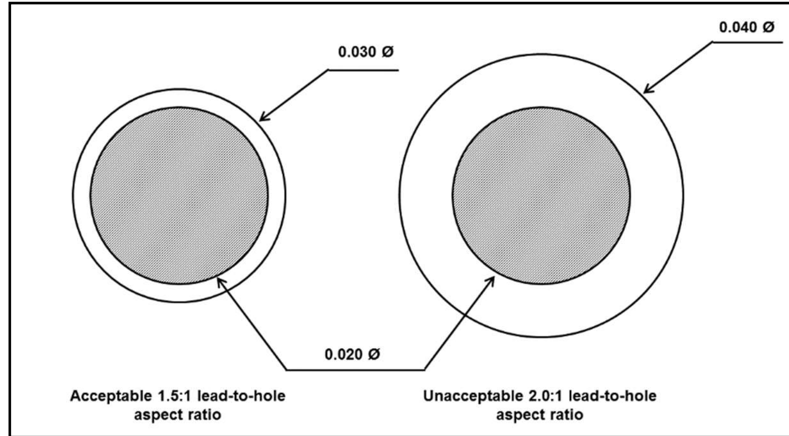
5.6 Sustained Preheat

For printed circuit board assemblies (PCBAs) containing a large number of through-hole (TH) components requiring an elongated cycle time, high thermal mass TH components, large multi-layer PCBAs and/or assemblies with heavy ground planes, it is beneficial to utilize sustained preheat during the soldering cycle. This is accomplished via a topside convection or IR heater with closed-loop control mounted over the solder pot, or nitrogen heating surrounding the solder nozzle, to maintain topside board temperature throughout the entire soldering cycle. It should be noted that while the use of sustained preheat is advantageous for improving solder quality and plated through-hole (PTH) vertical fill, it does not alter the intermetallic thickness or intermetallic microstructure.

6.0 Through-Hole Design Guidelines

6.1 Lead-to-Hole Aspect Ratio

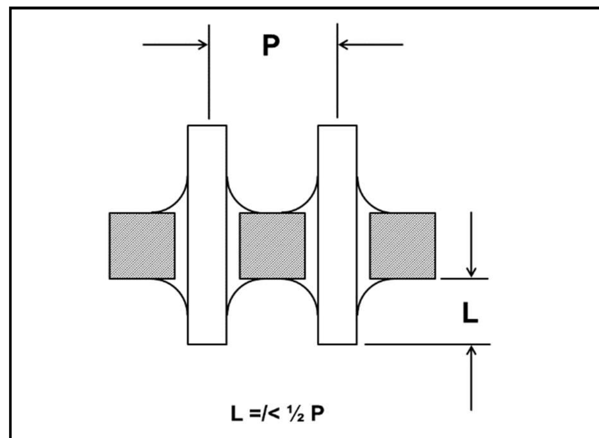
It is highly recommended that the lead-to-hole aspect ratio between the outside diameter of through-hole (TH) component leads and the inside diameter of plated through-hole (PTH) does not exceed a maximum aspect ratio of 1.5:1 which is considered as an industry standard. Therefore, if the component lead diameter is 0.020” (0.5mm) the diameter of the PTH should be 0.030” (0.75mm) to conform to industry standards for accepted best practice.



Examples of Acceptable and Unacceptable Lead-to-Hole Aspect Ratio

6.2 Lead Projection

It is imperative that proper consideration be given to lead projection, or the distance through-hole (TH) component leads project through the underside of the printed circuit board (PCB), to prevent bridging between adjacent through-hole (TH) solder joints. It is recommended that maximum lead projection exceed not more than one-half the pitch between leads, expressed as: L (lead projection) = ≤ ½ P (lead pitch).



Example of Recommended Lead Projection Design Rule

6.3 Lead Pitch

Selective soldering can easily solder through-hole (TH) components with a 0.100" lead pitch and in practice can successfully solder 0.050" (1.25mm) and 0.040" (1.0mm) lead pitch components providing reduced traverse solder speeds are used and attention is given to lead projection which is paramount for sub 100-mil (2.54mm) lead pitch components. In certain cases, selective soldering can be utilized for sub 40-mil (1.0mm) lead pitch applications providing that special attention is given to lead projection, solder nozzle design and special de-bridging techniques are employed.

6.4 DFMA Guidelines

It is recommended that design for manufacturability and assembly (DFMA) guidelines are adhered to in order to ensure a smooth transition of new printed circuit board (PCB) designs into the production process. With respect to selective soldering it is recommended that attention be given to adjacent component clearance, lead-to-hole aspect ratio, maximum lead projection and minimum lead pitch. Other aspects of the recommended design for manufacturability and assembly (DFMA) guidelines are listed below.

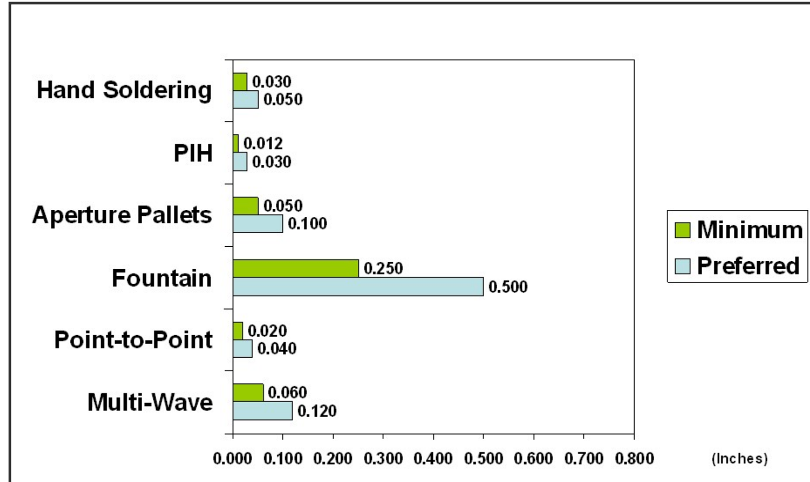
Nordson SELECT

Recommended Design for Manufacturability and Assembly (DFMA) Guidelines

Design Rule Summary	Preferred (inch/mm)	Minimum (inch/mm)
Internal Trace Width	0.008/0.20	≥ 0.004/0.10
External Trace Width	0.008/0.20	≥ 0.005/0.12
Signal Trace Width	0.008/0.20	≥ 0.004/0.10
Power Trace Width	0.025/0.63	≥ 0.005/0.12
Space Between Traces	0.008/0.20	0.005/0.12
Space Between Covered Pad and Trace	0.008/0.20	0.005/0.12
Space Between Uncovered Pad and Trace	0.008/0.20	≥ 0.007/0.18
Space Between Uncovered Pad and Covered Pad	0.025/0.63	0.020/0.51
Space Between Two Through-Hole Pads	0.050/1.27	0.030/0.76
Space Between Two Uncovered Pads	0.050/1.27	0.012/0.30
Space Between TH Pad and SMT Pad	0.050/1.27	0.030/0.76
Space Between Two SMT Components	0.050/1.27	0.020/0.51
Space Between TH Pad and Via Pad	0.050/1.27	0.030/0.76
Space Between Two TH Components	0.125/3.18	0.100/2.54
Space Between Two SMT Pads	0.050/1.27	0.025/0.63
Solder Mask to Uncovered Pad	0.010/0.25	0.007/0.18
Space Between SMT Pad and Via Pad	0.050/1.27	0.030/0.76
Space Between Ground and Power Pads	0.100/2.54	0.050/1.27
Edge of PCB to Edge of Trace	0.100/2.54	0.035/0.89
Space Between Two Via Pads	0.030/0.76	0.015/0.38
Pad for Plated Through-Hole Via	0.040/1.02	0.025/0.63
Finished Hole Size for Plated Through-Hole Via	0.025/0.63	0.013/0.33
Edge of SMT Component to Edge of PCB	0.200/5.08	0.050/1.27
Edge of Through-Hole Component to Edge of PCB	0.200/5.08	0.050/1.27
Edge of SMT Component to Edge of Through-Hole Component	0.050/1.27	0.020/0.51
Width of Solder Mask Between Openings	0.010/0.25	0.007/0.18
Space Between Edge of SMT Component and Edge of TH Pad	0.100/2.54	0.050/1.27

6.5 Adjacent Component Clearance

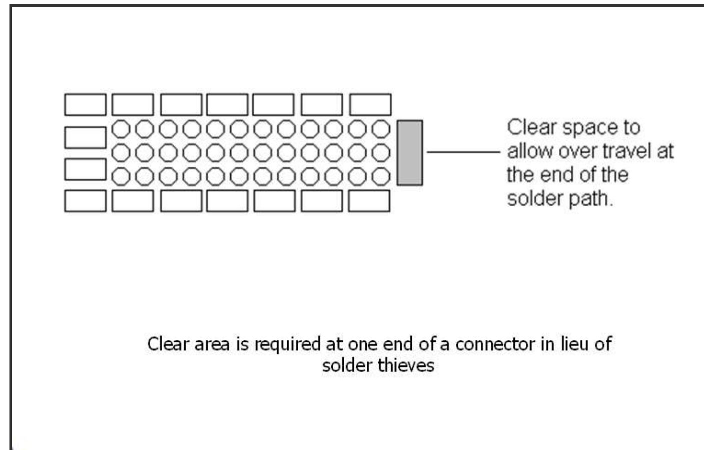
In comparison to other soldering methods, the adjacent component clearance for selective soldering is equal to or less than that of the most common through-hole (TH) soldering methods as shown below.



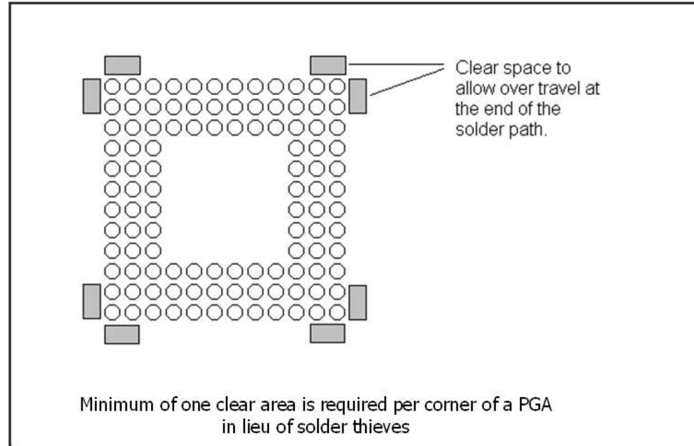
Comparison of Adjacent Component Clearance for Various Soldering Methods

6.6 Critical Keep-Out Areas

Under most conditions it is essential that the selective soldering nozzle be allowed to over-travel the last row of pins to prevent bridging of multi-row through-hole (TH) components such as connectors or pin grid arrays (PGA). In these cases, a keep-out area must be included during the printed circuit board (PCB) design phase to allow for this over-travel in lieu of solder thieves as shown below.



Typical Nozzle Over-Travel Clearance for Connector



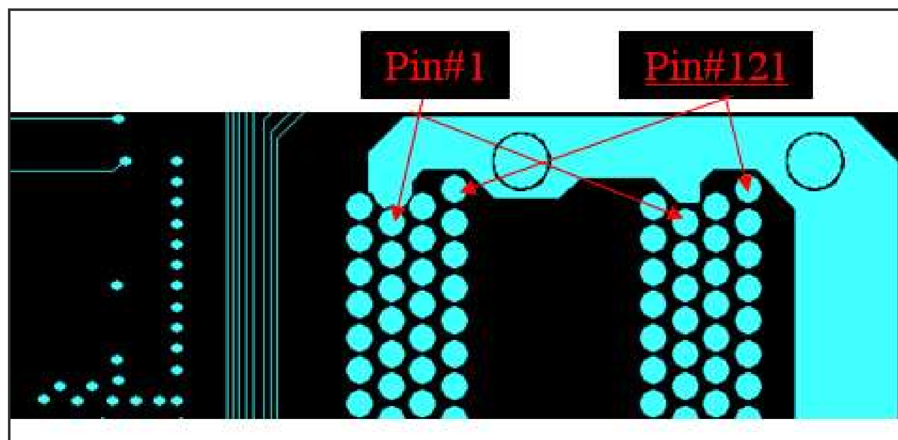
Typical Nozzle Over-Travel Clearance for Pin Grid Array (PGA)

6.7 Interlayer Construction

As part of a design for manufacturability and assembly (DFMA) review, it is recommended that the interlayer construction be reviewed to ensure that a limited number of layers be connected to any given plated through-hole (PTH) to ensure optimal vertical hole fill. Having an excessive number of signal and ground layers connected to a single plated through-hole (PTH) will draw thermal energy away from the barrel and will adversely affect capillary action and the corresponding vertical hole fill of the liquidous solder.

6.8 Ground Planes

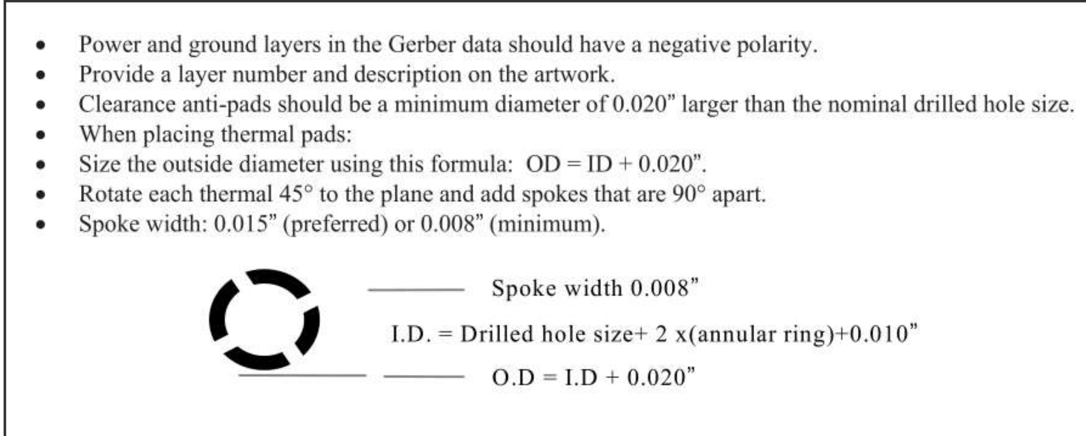
Likewise, every attempt should be made during the design process to avoid having ground planes directly connected to plated through-hole (PTH) as shown below which can result in poor vertical hole fill but rather having them defined with thermal relief design elements.



Example of Ground Plane Directly Connected to Through-Hole (TH) Leads

6.9 Thermal Relief Design Rules

When thermal relief design elements are incorporated into multiple layers of a multilayer printed circuit board (PCB), it is recommended to rotate the thermal relief by 45 degrees on alternate layers to minimize stress on the board in the Z-axis direction. Other recommended thermal relief design rules are shown below.



Recommended Thermal Relief Design Guidelines

7.0 Quality Measurement

7.1 Solder Joint Inspection Criteria

When assessing the solder joint quality of a printed circuit board assembly (PCBA) it is essential to determine the inspection criteria with respect to the applicable requirements of Class 1, Class 2 or Class 3 as shown below.

Typical Solder Joint Inspection Acceptance Criteria

Criteria	Class 1	Class 2	Class 3
A) Circumferential wetting on solder destination side of lead and barrel	Not specified	180°	270°
B) Vertical fill of solder	Not specified	75%	75%
C) Circumferential fillet and wetting on solder source side	270°	270°	330°
D) Percentage of original land area covered with wetted solder on solder destination side	0	0	0
E) Percentage of original land area covered with wetted solder on solder source side	75%	75%	75%

7.2 Post-Soldering Inspection Protocols

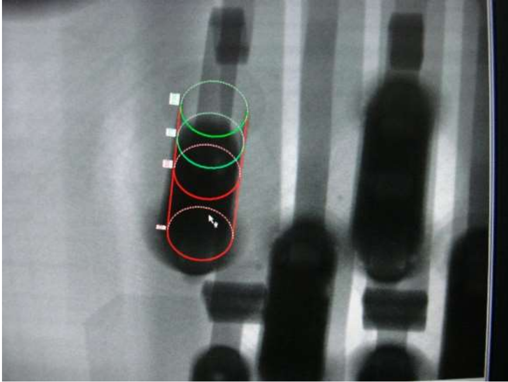
When monitoring the quality of through-hole (TH) solder joints within a printed circuit board assembly (PCBA) several elements of the solder joint formation can be measured including: 1) presence or absence of bridging on solder source side (microscope), 2) presence or absence of excessive solder on the solder side (microscope), 3) through-hole vertical hole fill (X-ray inspection or destructive cross-sectioning), 4) destination side wicking distance (endoscope-based microscope), and 5) destination side circumferential wetting angle (microscope).

For end-product applications where the subject printed circuit board assemblies (PCBAs) will be exposed to extreme temperature, humidity and or vibration, the intermetallic thickness and intermetallic microstructure of the through-hole (TH) solder joints may also be examined with scanning electron microscope (SEM) imaging.

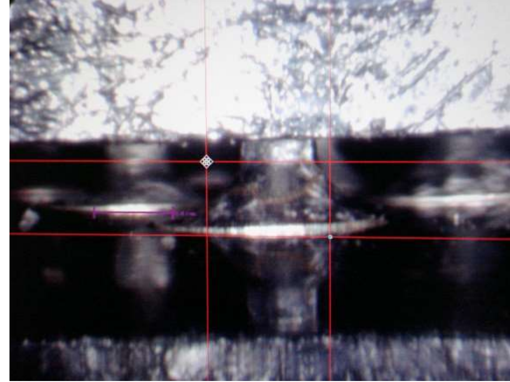
7.3 Inspection Methodologies

In the majority of cases the presence or absence of bridging and excessive solder can be detected by experienced inspectors using standard inspection microscopes. Through-hole vertical fill can be measured using potting and cross-sectioning, but this is a time consuming

and destructive inspection method. A much more effective method is to utilize a digital X-ray inspection system equipped with automatic measurement software capable of measuring the percentage of vertical hole fill. The measurement of destination side wicking distance can be carried out by using an endoscope-based microscope equipped with automatic measurement software as shown below.



Vertical hole fill percentage as measured with digital X-ray inspection system and auto measurement software

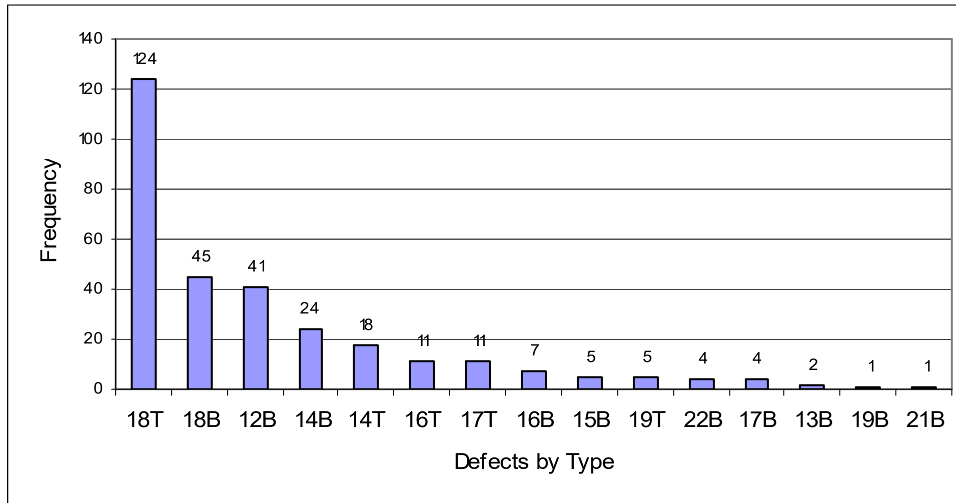


Destination side wicking distance as measured with endoscope-based microscope and auto measurement software

7.4 Pareto Analysis of Defect Type

An examination of through-hole (TH) solder defects must be made with the objective of improving and refining the selective soldering process including an analysis of defect reports for recent production batches. The defect data should be formatted into a Pareto analysis to determine which type of defects are occurring and the frequency of occurrence in order to determine the root cause and corresponding corrective actions as shown below.

Example of Through-Hole (TH) Solder Defects Sorted by Defect Type



Defect Codes:

18T – Non-wetting top
18B – Non-wetting bottom
12B – Lead too long bottom
14B – Fractured solder joint bottom
14T – Fractured solder joint top
16T – Excess solder top
17T – Pin holes top
16B – Excess solder bottom
15B – Cold solder bottom

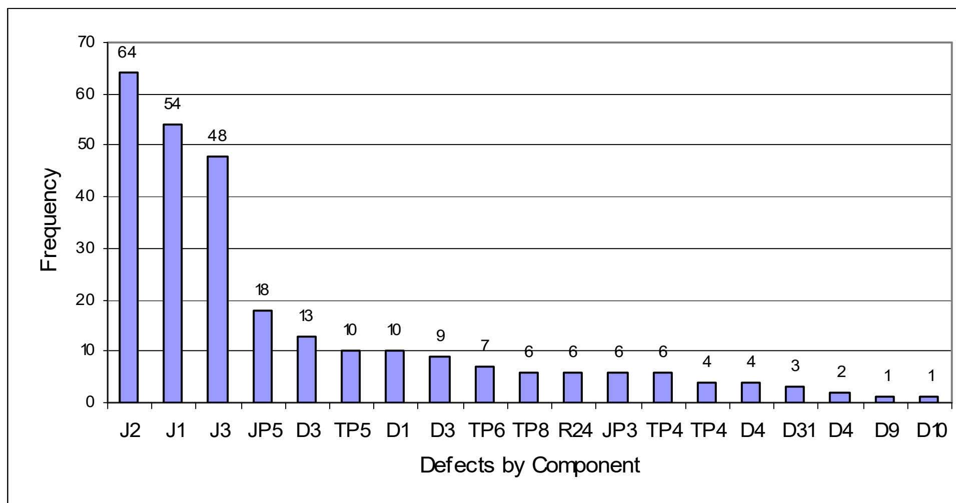
19T – Insufficient solder top
22B – Icicle bottom
17B – Pin holes bottom
13B – Lead not visible bottom
19B – Insufficient solder bottom
21B – Solder balls bottom

– Assembly related defects

7.5 Defect Frequency and Location

Further analysis of the solder defects must be made with respect to identifying the location and frequency of occurrence by through-hole (TH) component and component type. This information is helpful in determining potential component specific related root cause such as inadequate heat transfer due to component thermal mass, lead-to-hole design guideline violations or component solderability related issues as shown below.

Example of Through-Hole (TH) Solder Defects Sorted by Reference Designator

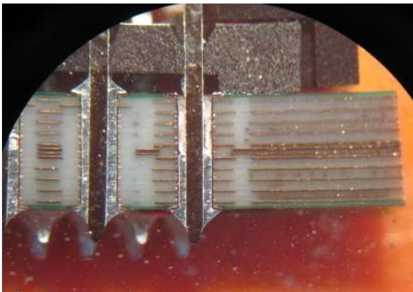


Note: Above data is based on ninety one (91) total boards soldered over three (3) months

8.0 Troubleshooting Guideline

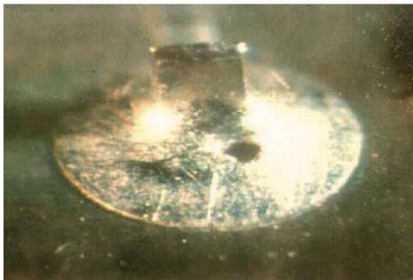
8.1 Defect Condition and Root Cause Analysis

Some of the more common process related solder defects include: 1) insufficient hole fill, 2) insufficient solder, 3) blow holes/voids, 4) excessive solder, 5) bridging, 6) solder balls, 7) poor hole filling, 8) pin-to-pin solder short, 9) adjacent solder short, and 10) sunken solder joints. Shown below are examples of these conditions as well as preventative actions for minimizing the corresponding defect.



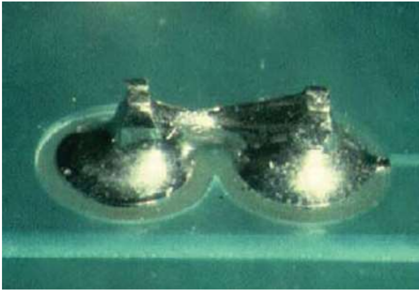
- Insufficient Hole Fill
  - Possible Causes: inadequate flux, topside PCB temperature too low, lead-to-hole ratio too small
  - Preventive Actions: verify flux deposition, verify preheat temperature, check wave height, check lead-to-hole aspect ratio, verify internal ground planes

- Insufficient Solder
  - Possible Causes: inadequate flux, dwell time too short, poor solderability, contaminated pads
  - Preventive Actions: verify flux deposition, reduce drag speed, verify preheat temperature, check component contamination, check board contamination



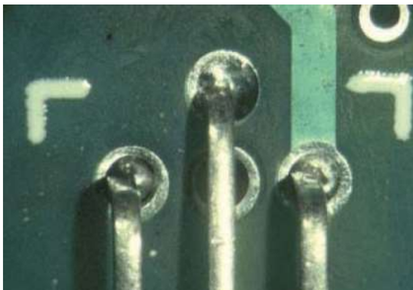
- Blow Holes/Voids
  - Possible Causes: PCB temperature too low, flux vehicle out-gassing, moisture in PCB, uncured solder mask
  - Preventive Actions: verify topside PCB temperature, verify flux deposition, check for moisture in laminate, check for defective or uncured mask material

- Excessive Solder
  - Possible Causes: lead length too long, solder speed too slow, excessive flux, solder temp. too low, contaminated solder
  - Preventive Actions: reduce lead projection, increase drag speed, verify flux deposition, increase solder temperature, check for solder contamination



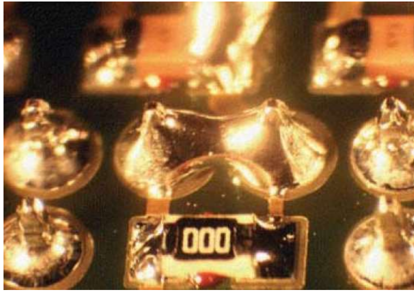
- Bridging
  - Possible Causes: lead length too long, inadequate flux, solder temperature too low, solder speed too fast
  - Preventive Actions: reduce lead projection, verify flux deposition, increase solder temperature, decrease drag speed, re-program peel-off movement

- Solder Balls
  - Possible Causes: excessive flux, topside PCB temperature too low, nitrogen level too high, solder mask porosity
  - Preventive Actions: reduce flux vehicles, verify topside PCB temperature, decrease nitrogen level, greater tendency with selective versus wave soldering

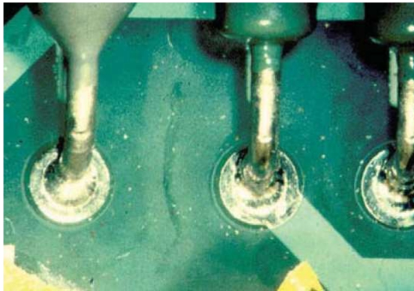


- Poor Hole Filling
  - Possible Causes: inadequate flux, topside PCB temperature too low, PCB contamination, poor solderability
  - Preventive Actions: Verify flux deposition, verify topside PCB temperature, check component contamination, check board contamination

- Pin-to-Pin Solder Short
  - Possible Causes: lead length too long, inadequate flux, lead pitch too small, solder temperature too low, solder speed too fast
  - Preventive Actions: reduce lead projection, verify flux deposition, increase solder temperature, decrease drag speed, re-program peel-off movement



- Adjacent Solder Short
  - Possible Causes: inadequate through-hole to adjacent SMT component clearance, lead length too long
  - Preventive Actions: implement keep-out design for manufacturability (DFM) design guidelines, re-program nozzle movement, re-program peel-off movement



- Sunken Solder Joint
  - Possible Causes: non-wetting or de-wetting, inadequate flux, topside PCB temperature too low, lead-to-hole ratio too large
  - Preventive Actions: check solderability, verify flux deposition, verify topside PCB temperature, check lead-to-hole aspect ratio, implement DFM design guidelines

8.2 TH Solder Defect Cause and Effect Matrix

An assembly process map of through-hole (TH) soldering must include all four aspects of the assembly operation that have a direct bearing on the quality of solder joint formation. Critical factors directly affecting solder joint quality include process parameters, printed circuit board assembly, board fabrication techniques, and board design.

Refer to the “Through-Hole Soldering Troubleshooting Guide” in Section 11.0 which provides a cross reference between common through-hole (TH) soldering problems and potential process, assembly, fabrication and design solutions.

8.3 Defect Resolution

The following measures are recommended as corrective actions to minimize through-hole (TH) bridging, shorts, the formation of solder balls, insufficient hole fill, and solder skips.

Corrective Actions to Minimize Bridging and Shorts

Recommended Action	Resulting Impact
Re-Establish Parallelism	Verify is board is seated properly in conveyor or gantry; verify if board is warped; verify if fixture is damaged
Control Board Impingement	Too deep of a solder immersion depth causes bridging; too shallow of a solder immersion depth causes insufficients
Optimize Dwell Time	Too long a dwell time causes bridging; shorten dwell time by increasing traverse speed
Improve Solder Conditions	Check dross removal and viscosity of solder by increasing temperature of solder pot

Corrective Actions to Minimize Solder Balls

Recommended Action	Resulting Impact
Increase Dwell Time	Solder balls are caused by too much flux solvent remaining on board causing micro-explosions; increase dwell time to re-liquefy solder or decrease flux deposition
Increase Preheat Temperature	Ensure flux solvent is completely vaporized and removed before soldering; too high of a preheat temperature can also burn away flux solids causing insufficients
Porosity of Solder Mask	Solder balls can be caused by a porous solder mask; check quality of solder mask in relation to IPC standards using microscope-based visual inspection

Corrective Actions to Minimize Insufficient Hole Fill and Skips

Recommended Action	Resulting Impact
Re-Establish Parallelism	Verify is board is seated properly in conveyor or gantry; verify if board is warped; verify if fixture is damaged
Control Board Impingement	Too shallow of a solder immersion depth causes insufficients; too deep of a solder immersion depth causes bridging
Optimize Dwell Time	Too short a dwell time causes skips; lengthen dwell time by decreasing traverse speed
Improve Solder Conditions	Check dross removal and viscosity of solder by increasing temperature of solder pot
Verify Fluxer Performance	Verify coverage, uniformity and amount of flux deposition by checking fluxer performance

8.4 Prevention of Re-Occurrence

The following rectification practices are put forth as suggested methods for the prevention of re-occurrence of conditions know to result in the generation of selective soldering defects.

Prevention of Re-Occurrence for Resolution of Defect Generation

Root Cause of Defect	Immediate Rectification
Flux Deposition	Through-hole (TH) flux penetration should be monitored using thermal facsimile paper mounted to the topside of a bare board or by using a Fluxometer or equivalent test instrument
Instrumentation Techniques	Proper instrumentation techniques and thermocouple attachment protocols should be used to ensure accurate measurement of thermal stress of a printed circuit board assembly (PCBA)
Thermal Profiling	The proper instrumentation of a fully populated PCBA should be carried out to accurately capture flux activation temperatures and verify the flux survivability process window
Solder Nozzle Condition	Solder nozzles should be properly cleaned, conditioned, re-tinned and maintained to deliver smooth and uniform solder flow around the entire outer surfaces of the nozzle with minimal turbulence
Board Impingement	Wave pressure and wave height consistency should be measured and controlled by using an automatic wave height detection system after proper nozzle conditioning to ensure uniform flow
Dwell Time	Solder dwell should be measured at the bottom surface of the printed circuit board (PCB) to ensure sustained thermal energy to render no-clean flux residues benign after selective soldering
PCB Design	Design practices should maintain a lead-to-hole ratio of 1.5:1, limited the maximum number of interlayer interconnections to a PTH and use thermal reliefs to optimize vertical hole fill
PCB Fabrication	Attention should be given to printed circuit board (PCB) fabrication practices to minimize the effects of excessive Z-axis thermal expansion (CTE_Z), pad lifting and the formation of solder voids
Defect Analysis	Ongoing monitoring and examination of solder defects should be carried out to detect trends and optimize process parameters to eliminate and/or minimize anomalies in solder joint quality
Process Optimization	A process characterization and design of experiments (DoE) should be carried out to ensure the selection of optimal process parameters and machine settings
Validation Run	A controlled lot should be carried out to determine the effects of the process optimization parameters and to further mitigate the frequency of defect occurrence
Component Re-Tinning	If component re-tinning is being carried out it should not be implemented using static solder pots, prone to dross and flux contaminants, or with manual process controls
Hand Soldering	If manual soldering is being carried out after the selective soldering process it should be verified that excessive liquid flux is not being used during assembly soldering or rework and repair
Post-Soldering Contamination	If no-clean flux is being used for reflow, selective and/or hand soldering, post-soldering board cleanliness should be monitored using a calibrated ionic test instrument
Solder Contamination	Analysis of the solder should be carried out every 8,000 boards or once every 3 months, whichever occurs first, to ensure contamination levels are within acceptable limits

9.0 Process Optimization

9.1 Optimization of Process Parameters

The underlying principle of conducting a process characterization is to ensure that the preheat and soldering temperatures are reduced to a minimum so that the printed circuit board assembly (PCBA) in question is subjected to minimal thermal stress during the selective soldering process. The process characterization work should be carried out in accordance

with a pre-defined selective soldering work plan and all quality measurement metrics should be carried out in accordance with a pre-defined inspection work plan.

In review, the objectives of a process optimization undertaking should be:

- Optimization of flux deposition and through-hole (TH) flux penetration
- Establish actual printed circuit board assembly (PCBA) temperature excursions using proper instrumentation and thermocouple attachment techniques
- Obtain flux activation temperature of printed circuit board assembly (PCBA) at lowest possible preheat settings
- Monitor flux activation window and flux survivability throughout the soldering operation
- Monitor solder joint formation temperature and solder nozzle temperature
- Monitoring of critical interlayer temperature throughout the soldering operation
- Minimize thermal shock to the printed circuit board assembly (PCBA) between preheat stage and soldering unit
- Confirmation of all applicable process parameters for use during a follow-on design of experiment (DoE)

As a general guideline, the following selective soldering process parameters can be used as a starting point when conducting a process characterization with the realization that printed circuit board assemblies (PCBAs) will vary drastically in their physical makeup.

General Selective Soldering Process Guidelines

Process Parameter	Recommended Initial Settings
Board Parallelism	Lack of board-to-nozzle parallelism is common cause of bridging, shorts, insufficient, skips and icicles; verify board warpage, proper seating in conveyor or gantry and fixture damage
Board Impingement	Inconsistent board impingement is common cause of bridging, shorts, insufficient, skips and icicles; suggested immersion depth with solder nozzle should be maintained at 0.038” to 0.048”
Dwell Time	Optimal dwell time should be 2.0-4.0 sec for tin-lead (SnPb) solder; optimal dwell time should be 0.5-1.5 sec longer for tin-silver-copper (SAC305) lead-free solder alloy
Topside Preheat Gradient	Suggested upper limit should be 2.0-2.8°C/sec or what is recommended by flux manufacturer; always follow thermal processing recommendations of flux manufacturer
Topside Board Temperature	Suggested topside temperature should be 88-110°C for alcohol-based flux or 100-130°C for water-based flux; always follow thermal processing recommendations of flux manufacturer
Maximum Topside Temperature	Do not exceed 183°C for tin-lead (SnPb) solder or 217°C for tin-silver-copper (SAC305) to prevent re-reflow of topside SMT components during selective soldering process
Maximum Board ΔT	Do not exceed 100°C for tin-lead (SnPb) or 140°C for tin-silver-copper (SAC305) ΔT between exit of preheat and entering solder nozzle to avoid thermal shocking and excessive warpage

9.2 DPMO and OFD Quality Measurement

An industry metric for quality measurement is expressed in defects per million opportunities (DPMO). It is highly recommended that defects per million opportunities be utilized as an ongoing measurement to establish a baseline of in-process quality as well as to monitor all follow-on improvements. Defects per million opportunities can be calculated as follows:

DPMO = (Number of Defects/OFD) x 1,000,000, where,
Opportunity for Defects (OFD) = Number of components + number of component I/O

World-class in-process quality for SMT assembly typically ranges between 50-200 DPMO while through-hole (TH) assembly is considerably higher at between 2,000-3,500 DPMO due to an increased number of material, component and assembly variables.

Example of DPMO Quality Measurement Data (over 4 Months of Production)

	Topside SMT	Bottom-Side SMT	Through-Hole
Number of Components	13	56	16
Number of Component I/O	29	60	68
Opportunity for Defects (OFD)	42	116	84
Sample Size (Boards)	13,879	15,679	16,339
OFD per Period	582,918	1,818,764	1,372,476
First Pass Yield	99.59-99.86%	98.66-99.69%	98.69-99.59%
Post-Wave Defects per Period	23	73	177
In-Process DPMO	39 DPMO	40 DPMO	1,289 DPMO

Note: Quantity of boards produced varies due to line balancing and production scheduling
Opportunity for Defects (OFD) = Number of components + number of component I/O
DPMO = (Number of Defects/OFD) x 1,000,000

9.3 DoE Methodologies

Once it has been confirmed that a selective soldering machine is in good working order and that initial process parameters have been established, a design of experiment (DoE) can be conducted in order to optimize all critical selective soldering process variables.

Based upon a correlation between the top and bottom machine pre-heater settings and the corresponding temperature of the topside and bottom-side of the printed circuit board (PCB) surfaces, a range of machine settings can be established for an excursion of target DoE process parameters. These relationships are expressed in the table below. It should be noted that variations in the temperature of the topside and bottom-side printed circuit board (PCB) surface should to be accomplished as part of the DoE by heating longer as opposed to raising pre-heat temperature thus emphasizing flux survivability.

Typical Design of Experiment Range of Target Process Parameters

Proposed DoE Settings	Minimum	Mid-Point	Maximum
Flux Value	35%	45%	2 x 35%
Flux Traverse Speed	5.0mm/sec	5.0mm/sec	5.0mm/sec
Top Convection Pre-Heat	90°C@70 sec	90°C@90 sec	100°C@90 sec
Bottom IR Pre-Heat	95°C@70 sec	95°C@90 sec	105°C@90 sec
Top Convection Pre-Heat	120°C	120°C	65°C
Solder Pot Temperature	280°C	290°C	300°C
Solder Traverse Speed	2.0mm/sec	2.5mm/sec	3.0mm/sec
Solder Nozzle Size	8/12	8/12	8/12
Target Top PCB Temperature	95°C	105°C	115°C
Target Bottom PCB Temperature	110°C	120°C	130°C

Based upon the above correlation, a range of experiments should be generated to define the matrix of machine settings to be used for the DoE in order to determine the optimized set of process parameters. It should be noted that each variation of a given process parameter should be conducted while other process settings were retained at their mid-point setting in order to assess the impact of each individual process parameter.

The range of experiments as defined prior to conducting the DoE is shown below.

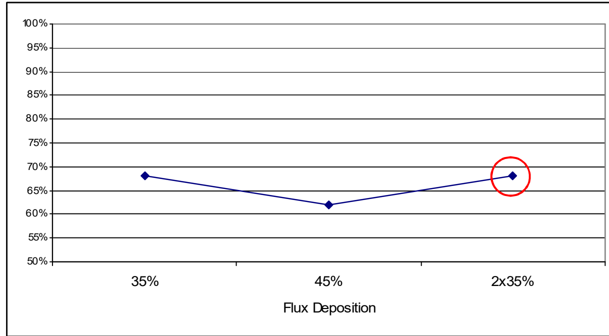
Nordson SELECT

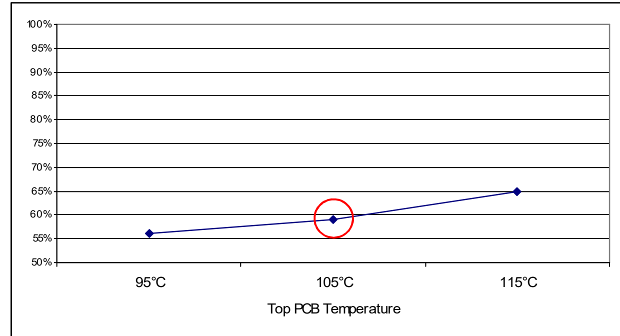
Typical Design of Experiment Matrix of Machine Settings

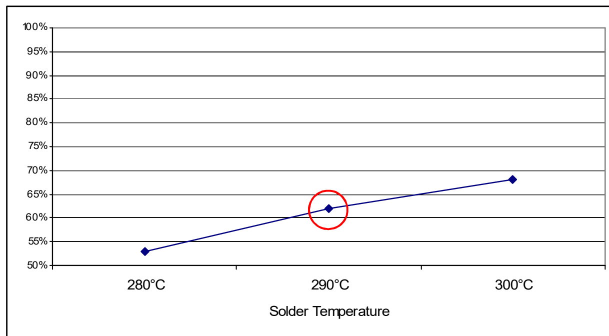
Run No.	Program No.	Experiment No.	Flux Deposition (%)	Top PCB Temperature	Solder Pot Temperature	Solder Speed (mm/sec)
1	1	1	35%	105°C	290°C	2.5
2	1	2	35%	105°C	290°C	2.5
3	2	3	45%	105°C	290°C	2.5
4	2	4	45%	105°C	290°C	2.5
5	3	5	2x35%	105°C	290°C	2.5
6	3	6	2x35%	105°C	290°C	2.5
7	4	15	45%	95°C	290°C	2.5
8	4	16	45%	95°C	290°C	2.5
9	5	17	45%	105°C	290°C	2.5
10	5	18	45%	105°C	290°C	2.5
11	6	19	45%	115°C	290°C	2.5
12	6	20	45%	115°C	290°C	2.5
13	7	23	45%	105°C	280°C	2.5
14	7	24	45%	105°C	280°C	2.5
15	8	13	45%	105°C	290°C	2.5
16	8	14	45%	105°C	290°C	2.5
17	9	21	45%	105°C	300°C	2.5
18	9	22	45%	105°C	300°C	2.5
19	10	7	45%	105°C	290°C	2.0
20	10	8	45%	105°C	290°C	2.0
21	11	9	45%	105°C	290°C	2.5
22	11	10	45%	105°C	290°C	2.5
23	12	11	45%	105°C	290°C	3.0
24	12	12	45%	105°C	290°C	3.0

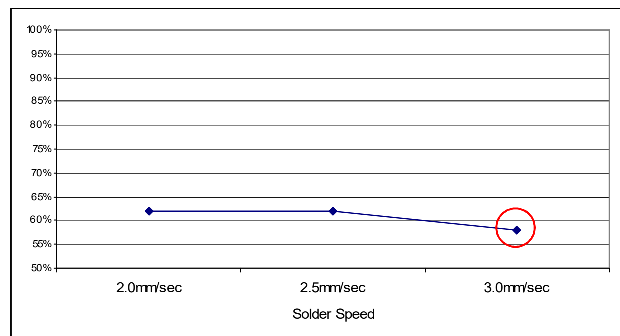
Measurement of discrimination metrics such as through-hole (TH) vertical hole fill as well as presence or absence of solder source side bridging and/or excessive solder should be used for determining the optimum DoE process parameters. Through-hole (TH) vertical hole fill is considered as a ‘positive’ indicator with a higher value being better while the presence or absence of solder source side bridging and/or excessive solder is considered as a ‘negative’ indicator with a lower value being better. Graphical representation of the DoE results is shown below.

Nordson SELECT

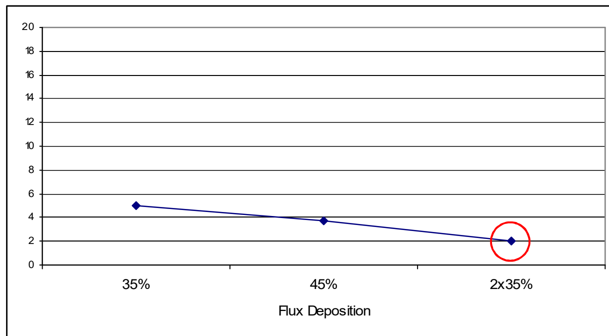


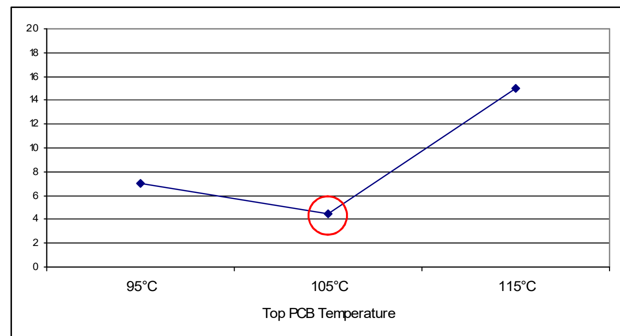


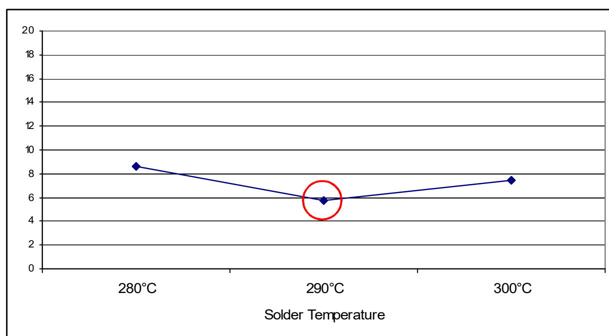


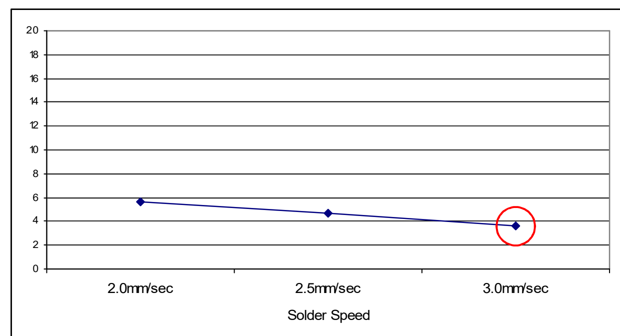


Average vertical hole fill percentage for pin 121 within critical components, as measured with digital X-ray inspection system for eight (8) total positions, sorted by variable process parameters; the higher the value the better









Frequency of solder source side bridging and/or excessive solder within critical components as measured with digital microscope for eight (8) positions, sorted by variable process parameters; the lower the value the better

In the above example, it was determined that 2x35% flux deposition, 105° topside printed circuit board (PCB) temperature, 290° solder pot temperature and 3.00mm/sec soldering speed produced the optimal results for both ‘positive’ and ‘negative’ indicators.

Nordson SELECT

9.4 Validation Run

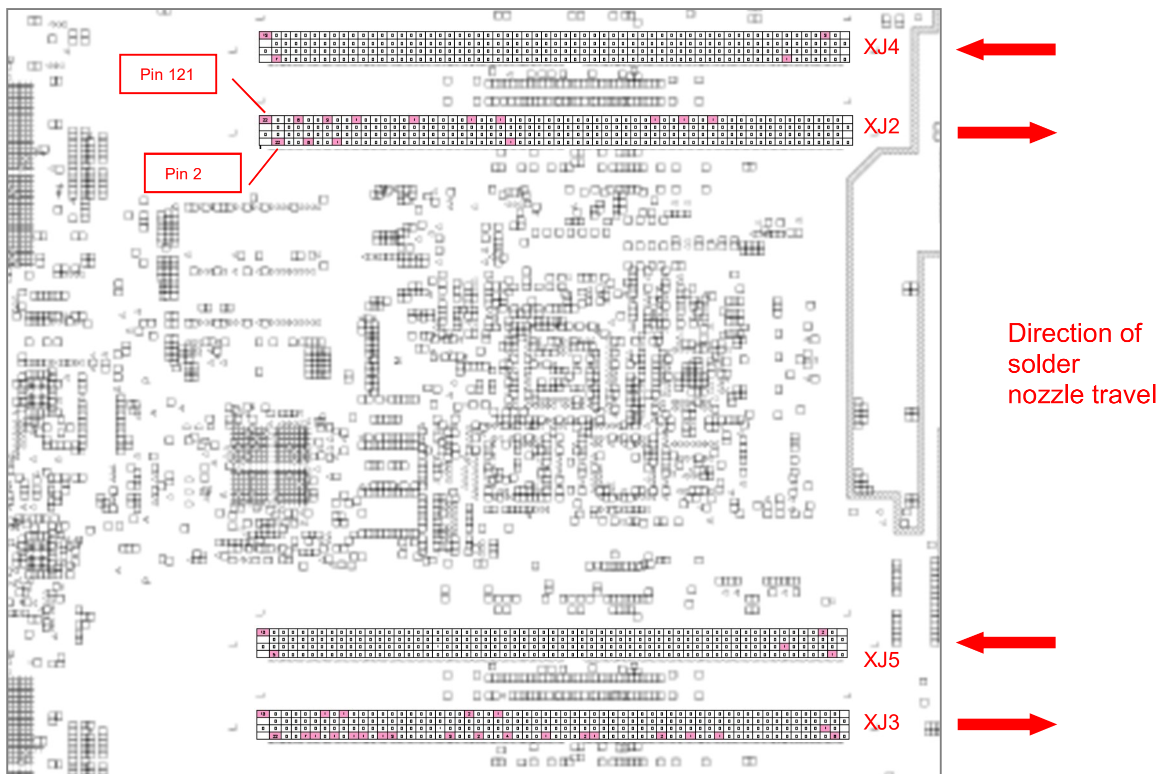
The purpose of a validation run is to confirm the optimal process parameters and machine settings developed as a result of the design of experiments (DoE) to determine the affects of these parameters upon the reduction or elimination of the related solder defects. Similar post-selective soldering inspection is carried out including measurement of bridging, excessive solder, destination side wicking height, vertical hole fill and solder voiding, etc. All in-process observations and anomalies should be noted and all solder defects should be observed and recorded to the pin level of each affected component to accurately denote the impact of follow-on corrective action.

Typically, fully populated and fully functional printed circuit board assemblies (PCBAs) are used for a validation run and are processed through all follow-on inspection and test operations such as in-circuit test (ICT), boundary scan, functional test, burn-in and thermal cycling, etc. to confirm optimal performance.

9.5 Defect Mapping Techniques

In cases where random anomalies and/or solder defects such as <75% vertical hole fill are detected on an infrequent basis, it is recommended that defect mapping be used to identify a pattern to the frequency of occurrence. An example of this defect mapping technique is show below.

Analysis of the defect map showed that the highest frequency of occurrence took place consistently on pins 2 and 121 due to these pins being connected to twelve (12) layers of a 22-layer multi-layer printed circuit board (PCB). It was also noted that random insufficient vertical hole fill was more prevalent on the outer row of pins on both connectors soldered during a second, or return pass of the solder nozzle. Further analysis confirmed this was due to the nitrogen shroud around the solder nozzle burning off some of the required flux solids on the adjacent connectors.



Defect mapping of <75% vertical hole fill observed during validation run plotted by frequency of occurrence and by pin number within critical components

10.0 Preventative Maintenance

10.1 Maintenance Procedures, Practices and Frequency

It is essential that adequate preventative maintenance be performed on a selective soldering machine in accordance with the equipment manufacturer's recommendations. This should include periodic cleaning of the drop-jet or spray flux head, removal of burned on excessive flux from bottom-side preheat modules, functional verification of thermocouples controlling preheat modules, frequent verification of solder nozzle condition and re-tinning as required, removal of dross from solder pots, and a periodic cleaning of the inside and outside of the overall machine. It is recommended that the equipment manufacturer's machine manual be consulted for specific preventative maintenance requirements to complement the general suggestions shown below.

Recommended Preventative Maintenance Schedule

Schedule	Frequency*	Preventative Maintenance Function
Daily	Every 8 hours of operation	Check solder nozzle condition
Daily	Every 8 hours of operation	De-dross top surface of solder pots
Daily	Every 8 hours of operation	Re-condition and re-tin solder nozzles
Monthly	Every 160 hours of operation	Remove pumps and clean impellers
Semi-Annually	Every 960 hours of operation	Clean solder pots
Semi-Annually	Every 960 hours of operation	Calibrate flux/solder pot X and Y axis
Semi-Annually	Every 960 hours of operation	Check flux/solder pot Z axis height
Semi-Annually	Every 960 hours of operation	Check solder contamination levels

* - Based on 40-hour work week, adjust accordingly for 2 or 3 shift operation, varies with type of solder alloy used

10.2 Solder Nozzle Maintenance

A critical area for a selective soldering machine is the preventative maintenance procedures and practices used for the various types of solder nozzles. Pre-tinning of wetted solder nozzles extends their working life and improves the flow characteristics of the molten solder as does periodic cleaning and re-tinning. Re-tinning of wetted solder nozzles should be carried out after first cleaning the nozzle with a highly active liquid flux followed by tinning the nozzle in a static solder pot filled with the same solder alloy as used in production.

Most selective soldering machines offer either wetted or non-wetted solder nozzles. Each type has advantages and disadvantages and each has different maintenance criteria. Wetted nozzles are generally made of cold-rolled steel and have the following advantages: 1) they can solder in any direction and at many angles, 2) can be as small as 1.8mm in diameter, 3) can solder extremely close to adjacent components, and 4) are lower in cost. The disadvantages of wetted nozzles are: 1) they may last 2 to 12 months depending upon the nozzle diameter, type of solder alloy and maintenance procedures employed, 2) require re-tinning maintenance after 2 to 8 hours of use, and 3) create slightly more dross due to the need to constantly run the solder pump to prevent the wetted nozzle from oxidizing.

Non-wetted nozzles offer directional flow of molten solder and are made of stainless steel with a coating to prevent corrosion by aggressive lead-free solders and have the following advantages: 1) they solder at a 7 degree angle similar to a traditional laminar wave to promote breakaway of excess molten solder, 2) can last up to 5 years or longer when using lead-free solder, 3) require no re-tinning and almost no maintenance, and 4) only run during the soldering cycle thus creating a lessor amount of dross. The disadvantages of non-wetted nozzles are: 1) they require more keep-out area between the side of the nozzle and adjacent components, 2) cannot solder backwards, 3) can only be as small as 4.0mm in size, and 4) are more costly than wetted solder nozzles.

10.3 Solder Alloy Contamination Levels

Since most lead-free alloys are lower in density than tin-lead (SnPb) solder, the copper-tin (CuSn) intermetallics that form will sink and disperse throughout the solder pot. Also, lead-free alloys dissolve copper at a faster rate than SnPb solder which results in a higher rate of copper build-up and contamination of the solder pot. Over prolonged periods of operation, the lead-free alloys begin to exhibit sluggish behavior caused by the build-up of CuSn intermetallic that forms at the bottom of the solder pot. Generally, this problem doesn't exist with standard SnPb solders because CuSn intermetallic floats and can be easily removed.

Tests have shown that some lead-free alloys cause corrosion to the base metals used for solder pots, impellers and nozzles due to the aggressive nature of tin at high temperatures. The surface of various base metals can exhibit pitting and start to dissolve after prolonged contact with lead-free alloys. The use of a higher grade stainless steel does retard this effect somewhat since these higher grades exhibit a reduced rate of iron (Fe) dissolution because due to a lower iron content. Moreover, unprotected cast iron solder pots are prone to high rates of iron dissolution due to the high iron content contained in cast iron.

Summary of Commonly Used Base Metals

Material	Advantages	Disadvantages	Estimated Longevity*
304 stainless steel	Inexpensive	No corrosion resistance	2-12 months
316 stainless steel	Inexpensive	No corrosion resistance	2-12 months
Cast iron	Inexpensive	No corrosion resistance	1-3 years
Surface coated stainless steel	Good resistance to corrosive effects of tin	Will degrade when coating is scratched	5-7 years
Surface coated cast iron	Good resistance to corrosive effects of tin	Will degrade when coating is scratched	3-5 years
Titanium	Excellent resistance to corrosive effects of tin	Can be expensive to fabricate	10 years

* - Estimated longevity under sustained production conditions in direct contact with molten lead-free solder

It is highly recommended that lead-free solder alloys be analyzed every 8,000 boards or once every three (3) months, whichever occurs first. A sample of the solder alloy should be sent to an outside laboratory for analysis of possible contaminants including copper (Cu) which should not exceed 1.0% by weight, iron (Fe) which should not exceed 0.02% by weight, or lead (Pb) which should not exceed 0.1% by weight as shown below.

Example of Lead-Free Solder Alloy Contamination Levels

Element	Upper Limit	Principle Cause	End Result
Copper Contamination (CuSn)	Tolerable up to 1.0%	Dissolution of Cu from circuit board surface finish	Will make solder flow sluggish and adversely affect performance
Iron Contamination ($FeSn_2$)	Maximum amount is 0.02%	Leaching of machine elements in contact with lead-free solder alloy	Can potentially make solder joints brittle
Lead Contamination (Pb)	Maximum amount is 0.1%	SnPb surface finishes and solder pot contamination	Can result in lead rich low melting point segments, cracking and other defects

* - Lead-free solder alloys should be changed whenever the contaminants reach 80% of the allowable upper limit

11.0 Appendix

- Through-Hole Soldering Troubleshooting Guide

Nordson SELECT

Through-Hole Soldering Troubleshooting Guide

Solutions		Insufficient Solder (Destination Side)	Insufficient Solder (Source Side)	De-Wetting or Non-Wetting	Solder Voids or Out-Gassing	Skipped or Omitted Solder	Excessive Solder (Destination Side)	Excessive Solder (Source Side)	Excessive Solder (Both Sides)	Icicles	Bridging	Solder Balls	Rough or Disturbed Solder	Grainy Solder	Cold Solder Joint	Disclosed Solder Joint	Flux Entrapment	Solder Mask Failure	Blistering	Measling	Lifted Components	Board Warpage
		Problems																				
Process	Board Not Seated	●	●	●	●	●	●	●	●	●			●								●	●
Process	Solder Angle Too Low							●	●	●	●											
Process	Solder Angle Too High	●	●		●																●	
Process	Conveyor Vibration									●			●	●	●						●	
Process	Solder Speed Too Low			●	●		●	●	●		●				●	●		●	●	●		●
Process	Solder Speed Too High	●	●	●	●	●		●		●	●	●	●	●	●	●	●					
Process	Excessive Flux	●		●	●			●	●			●					●					
Process	Inadequate Flux	●	●	●		●				●	●										●	
Process	Flux Application Uneven	●	●	●	●	●		●	●	●	●											
Process	Flux No Longer Active	●	●	●	●	●		●	●	●	●											
Process	Flux Contaminated	●	●	●	●	●		●	●	●	●	●				●	●					
Process	Preheat Temp Too Low	●	●	●	●					●	●	●		●	●	●	●					●
Process	Preheat Temp Too High	●	●			●				●	●					●		●	●	●		●
Process	Excessive Solder Dross												●	●								
Process	Solder Contaminated	●	●	●				●	●	●	●		●	●	●	●						
Process	Solder Wave Uneven	●	●		●	●	●	●	●	●	●	●	●								●	
Process	Wave Height Too Low	●	●	●	●	●				●	●		●									
Process	Wave Height Too High						●		●		●	●									●	
Process	Solder Temp Too Low	●		●	●		●	●		●	●		●	●	●							
Process	Solder Temp Too High	●	●		●		●		●			●			●	●		●	●	●		●
Assembly	Improper Board Handling	●	●	●	●	●				●	●										●	
Assembly	Comp Leads Too Short		●																			
Assembly	Comp Leads Too Long									●	●										●	
Assembly	Comp Contamination	●	●	●	●	●				●	●				●	●						
Assembly	Board Contamination	●	●	●	●	●				●	●	●		●	●	●		●				
Fabrication	Rough PTH	●			●							●					●					
Fabrication	Board Warped	●	●		●	●	●															●
Fabrication	Hole/Pad Misregistration	●	●		●	●																
Fabrication	Moisture in Laminate	●			●							●										
Fabrication	Defective Mask Material				●	●					●	●						●	●	●		
Fabrication	Misregistration of Mask	●	●		●	●																
Fabrication	Masking in Holes	●	●		●	●																
Fabrication	Board Contamination	●	●	●	●	●				●	●	●		●	●	●		●	●	●		
Fabrication	Board Oxidized	●	●	●	●	●				●	●	●										
Design	Weight Distribution	●	●			●	●	●														●
Design	Size of Board					●																●
Design	Component Orientation				●	●					●											
Design	Large Lead-to-Hole Ratio	●	●		●	●	●															
Design	Small Lead-to-Hole Ratio	●			●												●					
Design	Internal Ground Plane	●	●		●													●	●	●		●
Design	Plane on Solder Side	●	●					●		●	●							●	●	●		
Design	Plane on Comp Side	●	●				●															