ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES

IPC-S-816

SMT Process Guideline and Checklist

IPC-S-816

A standard developed by IPC

Original Publication
July 1993

2215 Sanders Road, Northbrook, IL 60062-6135
Tel. 847.509.9700 Fax 847.509.9798
www.ipc.org

The Principles of Standardization

In May 1995 the IPC's Technical Activities Executive Committee adopted Principles of Standardization as a guiding principle of IPC's standardization efforts.

Standards Should:
- Show relationship to Design for Manufacturability (DFM) and Design for the Environment (DFE)
- Minimize time to market
- Contain simple (simplified) language
- Just include spec information
- Focus on end product performance
- Include a feedback system on use and problems for future improvement

Standards Should Not:
- Inhibit innovation
- Increase time-to-market
- Keep people out
- Increase cycle time
- Tell you how to make something
- Contain anything that cannot be defended with data

Notice

IPC Standards and Publications are designed to serve the public interest through eliminating misunderstandings between manufacturers and purchasers, facilitating interchangeability and improvement of products, and assisting the purchaser in selecting and obtaining with minimum delay the proper product for his particular need. Existence of such Standards and Publications shall not in any respect preclude any member or nonmember of IPC from manufacturing or selling products not conforming to such Standards and Publication, nor shall the existence of such Standards and Publications preclude their voluntary use by those other than IPC members, whether the standard is to be used either domestically or internationally.

Recommended Standards and Publications are adopted by IPC without regard to whether their adoption may involve patents on articles, materials, or processes. By such action, IPC does not assume any liability to any patent owner, nor do they assume any obligation whatever to parties adopting the Recommended Standard or Publication. Users are also wholly responsible for protecting themselves against all claims of liabilities for patent infringement.

IPC Position Statement on Specification Revision Change

It is the position of IPC's Technical Activities Executive Committee (TAEC) that the use and implementation of IPC publications is voluntary and is part of a relationship entered into by customer and supplier. When an IPC standard/guideline is updated and a new revision is published, it is the opinion of the TAEC that the use of the new revision as part of an existing relationship is not automatic unless required by the contract. The TAEC recommends the use of the lastest revision. Adopted October 6. 1998

Why is there a charge for this standard?

Your purchase of this document contributes to the ongoing development of new and updated industry standards. Standards allow manufacturers, customers, and suppliers to understand one another better. Standards allow manufacturers greater efficiencies when they can set up their processes to meet industry standards, allowing them to offer their customers lower costs.

IPC spends hundreds of thousands of dollars annually to support IPC's volunteers in the standards development process. There are many rounds of drafts sent out for review and the committees spend hundreds of hours in review and development. IPC's staff attends and participates in committee activities, typesets and circulates document drafts, and follows all necessary procedures to qualify for ANSI approval.

IPC's membership dues have been kept low in order to allow as many companies as possible to participate. Therefore, the standards revenue is necessary to complement dues revenue. The price schedule offers a 50% discount to IPC members. If your company buys IPC standards, why not take advantage of this and the many other benefits of IPC membership as well? For more information on membership in IPC, please visit www.ipc.org or call 847/790-5372.

Thank you for your continued support.

ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES

IPC-S-816

SMT Process Guideline and Checklist

Developed by the Soldering Surface Mount Devices Task Group of the Joining Processes Committee of IPC

Users of this standard are encouraged to participate in the development of future revisions.

Contact:

IPC
2215 Sanders Road
Northbrook, Illinois
60062-6135
Tel 847 509.9700
Fax 847 509.9798

Acknowledgment

Any Standard involving a complex technology draws material from a vast number of sources. While the principal members of the IPC Soldering Surface Mount Devices Task Group of the IPC Joining Processes Committee are shown below, it is not possible to include all of those who assisted in the evolution of this standard. To each of them, the members of the IPC extend their gratitude.

Joining Processes Committee

Chairman
David Schoenthaler
AT&T

Soldering Surface Mount Devices Task Group

Chairman
Les Hymes
Plexus

Technical Liaison of the IPC Board of Directors

Bonnie Fena
Hibbing Printed Circuits

Soldering Surface Mount Devices Task Group

Abbagnaro, L., Pace Inc.
Abel, C.W., E.I. DuPont de Nemours & Co.
Amick, P.J., McDonnell Douglas Elec. Sys. Co.
Aoki, E.M., Hewlett-Packard Laboratories
Aoki, H., Airex, Inc.
Astbury, A., Altron Incorporated
Baker, J., Robotic Process Systems Inc.
Bates, G., Sherwood Medical
Bellia, F., Raytheon Co.
Bernier, D.F., Litton Systems Inc.
Bora, S.T., Smiths Industries
Bradshaw, C., Avex Electronics Inc.
Brill, C., Amp Inc.
Brooks, C.P., Amp Inc.
Burg, J.S., 3M Company
Caci, S.F., Raytheon Co.
Carroll, T.A., Hughes Aircraft Co.
Cash, A.S., Northrop Corporation
Cook, D.A., Synoptics Communications Inc.
Couble, E.C., Shipley Co.
Currie, D., Teledyne Systems Company
Davison, R.C., Jet Propulsion Laboratory
Davy, J.G., Westinghouse Electric Corp.
Elliott, D.A., Electrovert Ltd.
Engelmaier, W., Engelmaier Associates Inc.
Evans, G.P., Indium Corp. of America
Evans, J.W., NASA HQ
Feldmesser, H.S., Johns Hopkins University
Flaten, A.D., AT&T Information Systems
Gamalski, J., Siemens AG
Gandhi, V., Teradyne Connections Systems
Gechter, J., Delco Systems Operations
Gonzalez, C.J., SCI Manufacturing
Gray, B.W., Bull HN Information Systems
Gundotra, V., Motorola Inc.
Hampshire, W.B, Tin Information Ctr.,
Hiett, C.E., Martin Marietta Astronautics
Hillman D.D., Rockwell International
Hinton, P.E., Hinton ‘‘PWB’’ Engineering
Holmes, R.R., AT&T Microelectronics
Hook, M., U.S. Navy
Hymes, L, Plexus Corp.
Inpyn, B., Pitney Bowes Inc.
Jawitz, M.W., Litton Guidance & Control System
Johnson, K.L, Hexacon Electric Co.
Jones, S.A., Wilcox Electric Inc.
Kasilag, M., Aerojet Electrosystems Co.
Kenealey, G.W., GWK Enterprises Inc.
Kenyon, Dr. W.G., E.I. DuPont de Nemours and Co.
Knowles, L.P., Librascope Corp.
Kraszewski, R., Litton Systems Division
Kumar, V., Martin Marietta Electronics
Kuo, S.Y., Boeing Aerospace & Electronics
Lambert, L.P., Digital Equipment Corp.
Le, Q.N., Boeing Aerospace & Electronics
Leibowitz, J.D., Shireline Composites Inc.
Lichtenberg, L.R., Motorola Inc.
Mackzum, S.C., Ericsson GE
Maguire, J.F., Boeing Aerospace & Electronics
McKenna, G.T., Symbol Technologies Inc.
Meeks, S., Lexmark International/ IBM Corp.
Moffitt, J.H., U.S. Navy
Munie, G.C., AT&T Bell Laboratories
Nickell, R.D., U.S. Navy
Nielsen, R.L., Eastman Kodak Co. KAD
Novick, D.T., Rockwell International
O’Halloran, Dr. H., Siemens Stromberg-Carlson Corp.
Officer, R.B., Lockheed Sanders Inc.

Parkinson, E.H., Digital Equipment Corp.

Payne, R., Sundstrand Data Control Inc.

Pond, R., Texas Instruments Inc.

Reed, J.R., Texas Instruments Inc.

Rehm, P.M., Intel Corp.

Reithinger, M., Siemens AG

Rietdorf, B.C, Magnavox Electronic Systems Co.

Robertson, D.E., Pace Inc.

Rosser, J.G., Hughes Aircraft Co.

Rotman, A.B., DCMDN - QTC (Dept of Defense)

Rubin, Dr. W., Multicore Solder Ltd.

Rudy, D., AT&T Bell Laboratories

Rumps, D.W., AT&T Technology Systems

Rusignuolo, N., Hexacon Electric Co.

Russell, J.H., Defense General Supply Center

Scheiner, D., Kester Solder Division

Schneider, A., Alpha Metals Inc.

Schoenthaler, D., AT&T Bell Laboratories

Seltzer, M.L., Delco Systems Operations

Shah, K.K., Racal-Datacom

Siegel, E.S., Pace Inc.

Slezak, E., Litton Systems Inc.

Small, E., Multicore Solders

Smith, W.A., Hughes Missiles Systems Co.

Socolowski, N., Alpha Metals Inc.

Soper, W.A., TRW

Sovinsky, J.R., Indium Corp. of America

Stanik, E., Loral Defense Systems

Starosta, A., Eldec Corporation

Supp, M., Kester Solder Division

Svensson, J., Ericsson Telecom AB

Theroux, G., Honeywell, Inc.

Turbini, Dr. L.J., Georgia Institute of Technology

Underwood, H., U.S. Air Force

Vollmar, E., Methode Electronics Inc.

White, C.E., Indim Corp. of America

White, P.D., Tandem Computers Inc.

Won, Dr. T.S., Allied Signal Aerospace

Woodgate, R., Woodcorp. Inc.

Table of Contents

SMT Process Guideline and Checklist

1.0 SCOPE

This document is intended to provide guidelines and assistance in performing and troubleshooting the steps involved in the process of producing printed wiring assemblies incorporating surface mounting attachment of components.

Each section contains a list of problems often observed during a specific part of the surface mount assembly process. The list of observed symptoms is matched by a description of causes often associated with the symptoms. Suggestions for correction are also included. These solutions may be related to the equipment, materials, or design. Accordingly, some of these corrective measures cannot be implemented on the shop floor.

1.1 Environmental, Safety, and Industrial Hygiene Considerations The use of some of the materials referenced in this document may be hazardous. Precautions should be taken to safeguard personnel and the environment, as outlined in the appropriate data supplied with the materials used. In addition, equipment and procedures should adhere to applicable local and federal regulations.

1.2 Applicable Documents The following documents, of the issue currently in effect, are applicable to the extent specified herein.

1.2.1 IPC[1]

IPC-CH-65 Guidelines for Cleaning of Printed Boards and Assemblies

IPC-OI-645 Standard for Visual Optical Inspection Aids

IPC-R-700 Suggested Guidelines for Modification, Rework, and Repair of Printed Boards and Assemblies

IPC-SM-782 Surface Mount Land Patterns (configurations and Design Rules)

IPC-SM-786 Recommended Procedures for handling of Moisture Sensitive Plastic IC Packages

IPC-SM-840 Qualification and Performance of Permanent Polymer Coating (Solder Mask) For Printed Boards

1.2.2 Joint Industry Standards[1]

J-STD-002 Solderability Test for Component Leads, Termination, Lugs, Terminals and Wires

J-STD-003 Solderability Tests for Printed Boards

J-STD-004 Requirements for Soldering Fluxes

J-STD-005 General Requirements and Test Methods for Electronic Grade Solder Paste

2.0 HANDLING

Handling is one topic that is not confined to a specific area of the manufacturing operation. Handling problems are a ‘‘cradle to grave’’ issue for which no one will claim responsibility; yet the impact on the surface mount process yield is substantial. Every major step covered in this guideline, from adhesive application to cleaning of surface mount assemblies, relies on a comprehensive component and assembly handling strategy. Handling can’t be the concern of a single group in the manufacturing environment and must be an integral part of each process step in the surface mount assembly line. Although the general format of this document is problem-cause-corrective action, the following handling items need to be considered throughout the entire surface mount assembly process.

2.1 Electrostatic Discharge (ESD) Concerns The following are ESD concerns:

1) Has process machinery been properly grounded?
2) Have work stations and storage areas been properly grounded?
3) Are other environmental controls (i.e., ionizers, humidifiers, etc.) in place to reduce the occurrence of ESD?
4) Have operators and all other employees who may handle components or surface mount assemblies been properly trained to use ESD precautions such as wrist straps or other similar preventative measures?
5) Have the component packaging and assembly containers been reviewed for potential ESD problems?
6) Have ESD sensitive parts been identified and sufficiently protected?

2.2 Component Leads The following are concerns regarding component leads:

1) Has component lead coplanarity been addressed following manual or automatic lead forming operations?
2) Has component lead solderability concerns in relation to handling been reviewed and precautions taken where deemed necessary?

2.3 Storage The following are concerns regarding storage:

1. IPC, 2215 Sanders Road, Northbrook, IL 60062

1) Have the effects of the storage environment on components, materials or assemblies been reviewed within your manufacturing environment?

2) Does your handling system have a method of monitoring shelf life issues of stored components or assembly materials to prevent unacceptable material from being issued into the manufacturing environment?

3) Would the use of date codes on assembly materials and components be beneficial in ensuring timely usage?

2.4 Interim Handling The following are concerns regarding interim handling:

1) Would the use of dedicated assembly or component carriers prevent handling defects between process operations?

2) Have operators and assembly routers who handle components or surface mount assemblies between process operations been properly trained and made aware of their potential impact on the assembly process?

3.0 INCOMING INSPECTION OF MATERIALS AND COMPONENTS

Surface mount assembly complexity can range from just a few components to several hundred on one printed wiring board (PWB). Adhesives, solder, flux, printed circuit boards, heat sinks, and components are just some of the variety of ''building blocks'' used in surface mount technology (SMT) assembly. The breakdown of form, fit, or function of just one of these building blocks can result in a rework operation or even the rejection of a completed assembly. Incoming inspection of the materials and components which comprise an assembly is one method of reducing, or eliminating, the possibility of this situation occurring. Incoming inspection can accomplish two purposes:

1. *Conformance of Material to Specification Requirements* Surface mount assembly costs rise sharply when nonconforming materials find their way into the manufacturing process. Important key material characteristics can be identified and checked prior to the relase of the material to the manufacturing floor. Conformance testing can also be utilized for evaluating materials which can degrade over time, such as solderability or adhesive shelf life.

2. *Vendor Evaluation* When a particular material or component is procured from several different sources, incoming inspection can be used for evaluation comparisons. The comparison may reveal one particular vendor which supplies a material more consistently and with less variation between different material lots. This type of data can be shared with vendors to improve a particular assembly manufacturing process or be incorporated into a vendor rating system.

Incoming inspection of materials and components should have a defined purpose and be performed in conjunction with a good statistical process control plan. Emphasis on a well-planned incoming inspection system results in lower manufacturing costs through fewer surface mount assembly defects and a reduction in the number of surface mount assembly scrapped.

4.0 ADHESIVE APPLICATION

In surface mount technology the most prevalent use of adhesives is for component retention prior to, and during, bottom side wave soldering.

The most common method of adhesive application is automatic syringe dispensing. Hand syringe dispensing is also used for lower volumes and rework/repair operations. Pin transfer is another method often used in high volume, low mix operations.

Control of adhesive location and material properties for effective dispensing are the most common difficulties encountered.

This section presents a general listing of problems encountered in the adhesive application process, potential causes, and suggestions for corrective actions. These solutions may be related to equipment, materials, design, or prior processes. Accordingly some of the solutions cannot be immediately implemented on the shop floor.

4.1 Dispensing

4.1.1 Dispensing One Part Adhesive

4.1.1.1 Observed Condition: Stringing Most frequently related to both dispenser and adhesive.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Dispenser pressure too high	Reduce pressure
2. Dispensing period too long	Increase nozzle tip size
3. Nozzle too far from substrate	Reduce distance of nozzle to substrate
4. Nozzle drools	Use vacuum suck back
5. Adhesive on soldering surface	Change direction of nozzle travel
6. Tip length too long	Shorten tip
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Viscosity of adhesive too high	Lower viscosity Increase reservoir temperature
2. Viscosity of adhesive too low	Reduce reservoir temperature
3. Plastic flow of adhesive (dribble)	Choose more thixotropic material
4. Air in adhesive	Remove air by centrifuging

4.1.1.2 Observed Condition: Nozzle Clogging Most frequently related to down time, curing, and foreign matter.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Down time too long	Follow adhesive manufacturers' recommendation
2. Reaction with nozzle or tip	Anaerobic action—*do not use* copper or brass; use passivated stainless steel or plastic.
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Reaction by absorption of moisture (epoxy)	Keep nozzle tip closed during down time
2. Mixing of two different adhesives	Thoroughly clean system when changing adhesives

4.1.1.3 Observed Condition: Skipping and Insufficient Adhesive Most frequently related to clogged nozzle.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Clogged nozzle, bubbles, and foreign material	Remove cured adhesive, lumps, air
2. Worn or bent nozzle tip	Replace tip
3. Insufficient height	Dispense adhesive on top of original dot (double dispense)
4. Variation in dot size	Check dispenser output for consistency
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. PWB not flat	Obtain flat boards Check hold down
2. Viscosity too high for nozzle tip and pulse rate	Reduce viscosity Heat reservoir Select another adhesive
3. Insufficient height and/or excessive spread	Increase viscosity Select a more thixotropic adhesive Cool reservoir

4.1.1.4 Observed Condition: Excessive Adhesive Most frequently related to thixotropic properties of adhesive, temperature of reservoir, and equipment controls.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Low viscosity	Lower reservoir temperature
2. Dispenser pressure too high	Lower pressure
3. Tip size too large	Reduce tip size
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Low viscosity	Increase adhesive viscosity Select adhesive with higher thixotropic ratio
2. Excessive spread	Select another adhesive

4.1.1.5 Observed Condition: Insufficient Cure Most frequently related to insufficient curing energy leaving soft, gummy, or tacky adhesive.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Insufficient energy	Adjust cure time/temperature Check output of UV lamps
2. Too much energy which damages components	Reduce temperature Select another adhesive with different cure cycle
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Local heat sinks	Move components on PWB
2. Heat sensitive components	Move components to bottom side, IR heat top side

4.1.1.6 Observed Condition: Lost Components Most frequently related to insufficient cure, poor bond or adhesive, or non-wetted component.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Insufficient cure	Check time/temperature and energy level of cure cycle
2. Adhesive bond	Decrease time between placement and cure
3. Adhesive dot too low with insufficient component contact	Print second dot on top of original dot Increase tip size
4. Fast speed and stop in X-Y movement	Increase initial tackiness of adhesive Reprogram/reduce speed of placement
5. Poor placement of component	Realign equipment
6. Uncured adhesive (UV)	Place two dots either side of component
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Adhesive bond/strength	Check PWB and component cleanliness/bondability with adhesive
2. Adhesive dot too low with insufficient component contact	Use high dot profile adhesive and high viscosity for initial tack
3. Adhesive shrinkage during cure	Select another adhesive

4.1.1.7 Observed Condition: Adhesive Voids Most frequently related to the presence of moisture in adhesive or on surfaces.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
No information available	
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Moisture in adhesive	Check adhesive for excess moisture If excessive, cease use and determine and eliminate source of moisture
2. Absorbed moisture on bare PWB	Bake PWB before using adhesive Check solder mask for porosity
3. Absorbed moisture on surface of component	Bake components before assembly

4.1.2 Dispensing Two Part Adhesive Similar to 4.1.1 with a requirement that if recommended mixture ratio is not met, the cure will not be satisfactory with a potential loss of components.

4.1.3 Pin Dip Transfer

4.1.3.1 Observed Condition: Stringing Most frequently related to withdrawal rate and viscosity.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Rate of pin withdrawal	Correct speed of vertical travel
2. X-Y movement during withdrawal	Correct horizontal movement
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Viscosity too high	Reduce viscosity
2. Flow properties do not match pin travel	Select adhesive with proper properties
3. Viscosity too low	Increase viscosity so adhesive does not drip

4.1.3.2 Observed Condition: Skipping and Insufficient Adhesive Most frequently related to amount of adhesive in pins.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Pin size too small	Increase pin size
2. Pin shape incorrect	Select different pin shape
3. Pin does not wet	Clean pin and/or change material
4. Pin immersion incorrect	Check pin position and reservoir depth
5. Reservoir withdrawal rate incorrect	Check rate of withdrawal
6. Withdrawal from PWB incorrect	Reduce speed of withdrawal
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Insufficient adhesive on pin	Adjust viscosity and thixotrophy
2. Adhesive drops off during transfer	Increase viscosity

4.1.3.3 Observed Condition: Excessive Adhesive Most frequently related to flow properties of adhesive and variations in temperature of the reservoir.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Pin size too large	Reduce pin size
2. Too much adhesive on pin	Reduce depth in reservoir
3. Adhesive reservoir/pin plate not coplanar	Level adhesive reservoir
4. Pins have become rough	Replace pins
5. Build up of adhesive on pin	Clean pins
6. Decreased distance between pin and PWB	Correct spacing
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Viscosity increased	Adjust viscosity
2. Wrong pins used	Replace pins

5.0 SOLDER PASTE APPLICATION

Several solder paste application techniques are used to transfer solder paste to areas being joined. The most commonly used methods include: stencil printing, screen printing, and syringe dispensing. Less common techniques, such as direct pin transfer and wheel transfer, are used for some special applications. With the trend toward use of components having closer and closer lead spacing, error-free application of solder paste is more difficult and the direct relationship between poor quality application and final yield is more apparent.

Poor application can cause five common defects: insufficient solder, excess solder, shorts, opens, and solderballs. Each of these defects can be the result of one or more of the following symptoms: insufficient solder paste deposited, excess solder paste deposited, wet bridging, or misplaced solder paste. Each of these symptoms can be the result of a variety of causes.

5.1 Stencil Printing

5.1.1 Observed Condition: Insufficient Solder Paste Deposited In order for enough solder paste to be deposited in the area to be soldered, the volume of the stencil opening should be approximately equal to the desired amount of wet solder paste.

A. Equipment/Process Related	
Potential Cause	*Corrective Active*
1. Stencil too thin	Increase stencil thickness

General Guideline of Stencil Thickness for Various Applications	
Stencil	*Components*
8-20 mils	chip components only
8 mils	leaded with >31 mil pitch
6 mils	leaded with 20–25 mil pitch
<6 mils	finer lead spacing

Potential Cause	*Corrective Action*
2. Stencil opening too small	Increase stencil opening

The width of a stencil opening is usually defined as the smallest dimension in the X-Y plane. If the width of the opening is too small when compared to the thickness of the stencil, it will be difficult for the solder paste to release from the stencil hole. Examine the smallest holes in the stencil after printing. If solder paste is hanging-up in these holes, but not in the larger ones, it is possible that the stencil is too thick or the openings are too narrow.

If it is absolutely necessary to use a thick stencil with narrow openings, then one can attempt to accelerate the release of the solder paste from the stencil by increasing the snap-off distance to 50 to 90 mils. With this method the solder paste has a better chance of pulling away from the stencil, but there will also be a greater chance of print misalignment.

Potential Cause	*Corrective Action*
3. Stencil defects	Check quality of stencil

In addition to design defects in stencils, various stencil manufacturing defects are also possible, such as incomplete etch, improper image placement, or rough surfaces inside the stencil holes. If too little solder paste is consistently observed on one part of the board, then examine the corresponding section of the stencil for possible defects.

Potential Cause	*Corrective Action*
4. Squeegee blade too soft	Soft squeegee blades pressure deform and scoop solder paste out of the large stencil holes, particularly when used with high squeegee pressure. Squeegee blades with a durometer hardness less than 80 are considered soft. Vendors usually manufacture blades in 5 to 10 durometer increments.
5. High squeegee travel speed	The squeegee travel speed may be too fast for the solder paste and stencil design used. If the squeegee speed is not slow enough the paste may not roll or flow evenly into the stencil openings. Normal squeegee travel speeds range from 0.5–2.0 in./second. The solder paste vendor should be consulted for the recommended range for a given formulation.
6. Snap-off distance too small	Some solder paste users attempt to print with thicker than recommended stencils having very narrow openings to compensate for common component problems, such as lead non-coplanarity. As a result, there is a greater tendency for the solder paste to adhere to the walls of the stencil hole. One way to compensate for this is to increase the starting distance between the stencil and circuit board and correspondingly increase the squeegee pressure to force the stencil to the circuit board and solder paste through the holes. The solder paste pulls more easily out of the stencil holes, because the stencil starts to lift from the board immediately after the squeegee passes. This technique is not normally recommended, because the incidence of print misalignment and smear is increased.

Potential Cause	Corrective Action
7. Improper squeegee pressure	If squeegee pressure is too light, solder paste may not be effectively forced to the bottom of the stencil hole and will not transfer to the pad. If the pressure is too strong it is possible that the squeegee blade will deform and scoop solder paste out of the larger stencil openings. To properly adjust the squeegee, first decrease pressure until a thin layer of solder paste is left on the stencil after making a pass. Pressure should then be increased slowly until all of the paste is cleaned off the stencil after each pass.

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Too little tackiness	The solder paste may not be tacky enough for the given print parameters. In order for solder paste to roll in front of the squeegee blade, instead of slide, it must have sufficient tackiness to adhere to the stencil material. A solder paste which does not roll has difficulty filling all of the holes. Solder paste tackiness is most easily controlled by the vendor when developing the formulation, but may also be enhanced by increasing the flux percent or by raising the solder paste temperature.
2. Too much tackiness	It is also possible for a solder paste to be too tacky for a given application. If the solder paste is so tacky that it hangs up in the stencil holes without fully transferring to the pads, then it is too tacky.
3. Large or irregular solder particles	Particles which are too large or irregular may have difficulty being forced into small stencil openings. If there is any doubt about the solder powder particle size or shape, it can be examined by spreading out a small quantity of solder paste under a 10X to 100X microscope.
4. Viscosity too high	The solder paste viscosity may be too high for the print parameter being used. It is difficult to force very viscous solder paste into narrow stencil holes unless the other paste characteristics are properly balanced. In order to maintain consistency, it is best for the solder paste vendor to make any permanent viscosity changes; however, as a temporary step, it may be lowered by increasing the solder paste temperature, adding thinner to increase the flux percent, stirring vigorously or increasing the squeegee speed.
5. Viscosity too low	Low viscosity solder pastes can be scooped out of large stencil holes, leaving a thinner deposit than on other pads. This normally happens when low viscosity solder paste is used in combination with soft squeegee blades and high squeegee pressure. Although the solder paste vendor can increase the solder paste viscosity, the best way to handle this problem is by using a harder squeegee blade.
6. Low percent metal	The percentage of metal in the solder paste may be lower than required to give a proper sized solder joint. Even though the stencil opening may be properly designed for printing, solder pastes having too low a percentage of metal may not give a sufficiently large joint after reflow. Solder paste vendors report the amount of metal in solder paste as a weight percent. Because of the large difference between the solder and flux, approximately 7:1, a small change in metal weight percent has a larger effect on the metal volume percent. Consult the solder paste vendor if it appears that too little solder is in the joint area, even though the recommended amount of solder paste is being deposited.

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
7. Bubbles in solder paste	Bubbles may surface in the solder paste as it rolls in front of the squeegee, either as the result of air entrapment or as an inherent characteristic of the flux thickening system. Bubbles can prevent the correct quantity of solder paste from being deposited in the stencil holes by displacing solder paste. The solder paste vendor should be consulted if bubbles continue to be observed after several squeeqee passes.
8. Low solder paste temperature	Solder paste used at lower than normal operating temperatures will have a higher than published viscosity. The lower viscosity may cause the solder paste to slide on the stencil and not be pushed through the holes. Consult the solder paste vendor for alternatives if the solder paste or environment cannot be warmed up.
9. High solder paste temperature	As temperature increases, the viscosity of solder paste will decrease. Lower viscosity solder paste has a tendency to be scavenged from the pads by soft squeegee blades.
10. Not enough solder paste on stencil	If too little solder paste is used, it is difficult to get proper roll in front of the squeegee and some holes may be skipped. A round bead of solder paste, approximately 1/2 inch to 1 inch in diameter, is normally recommended.

5.1.2 Observed Condition: Excess Solder Paste Deposited Excess solder after reflow or solder shorts caused by excess solder paste deposits and wet bridging.

Although it is very difficult to create excessively heavy fillets by any means other than depositing too much solder paste, there are many reasons for the creation of solder shorts or bridges: improper reflow profile, excessive component placement pressure, component movement, improper solder paste, or incorrect solder paste application.

There are many signs of incorrectly applied solder paste, including: large deposits, wet bridges, smeared print, or misplaced print.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Damaged stencil	One very frequent cause of wet bridging is damage to the stencil. In order for solder paste to print cleanly, intimate contact between the stencil and circuit board must be maintained as the squeegee passes the opening. Frequently, the fragile stencil area between adjacent holes can be bent slightly. The bent area allows solder paste to pass underneath the stencil and form a wet bridge. If one consistently observes poor quality print in one area of the board, then one should look for damage in the corresponding area of the stencil.
2. Stencil defects	If the stencil is not properly imaged or etched, inconsistency will exist from opening to opening. Improper imaging or etching can cause some holes to be larger than others and stencil areas between holes may become very narrow. If this happens, one can expect a higher occurrence of wet bridges.
3. Inadequate stencil frame size	Because stencils are anchored within finite frames, the center of the stencil is capable of greater displacement from the frame plane than the stencil edges. When using small snap-off distances, the ability of the squeegee to press down on the stencil will not be affected. Inconsistent print thickness and wet bridging is possible when using large snap-off distances. If consistent wet bridges or smeared prints are found at the edges of the stencil near the frame, but not near the center, then consider decreasing the snap-off distance or using a larger frame size.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
4. Stencil not centered in frame	When using fixed rail in-line assembly systems, the stencil surface is not centered in the frame; instead, it is oriented relative to the fixed rail. Unequal stencil deflection is a common cause of inconsistent print because the edge of the stencil close to the frame will not come in contact with the board as easily as the center of the stencil. If an unbalanced stencil must be used, then one should use on-contact print with the corresponding stencil thicknesses and print parameters.
5. Improper tension of screen border	If the screen border around the stencil is loose or has been stretched in one direction, there will be more stencil movement in the X-Y plane. As the stencil moves in the X-Y plane, the likelihood of a smeared print increases. X-Y stencil movement increases as squeegee pressure is increased, because the amount of friction between the squeegee blade and stencil surface is increased. It is common to find X-Y movement when using thick stencils with high snap-off distances and greater than normal squeegee pressures. Poor circuit board design or quality can affect solder paste printing. It is very difficult to get a consistent print on an inconsistent surface.
6. Squeegee speed too fast	At high squeegee speeds, solder paste will shear thin and heat up. As this happens, the solder paste viscosity drops. If the drop is large enough, there will be a deterioration of print definition. This can be seen by the increased formation of wet bridges and poorly defined mounds of solder paste. If the print quality is good, but there is a gradual deterioration, the squeegee speed should be decreased. If this is not possible, then the solder paste manufacturer should be consulted for further recommendations.
7. Environment too warm	Not all solder paste users have the ability to control their working environment. Because solder paste generally thins as temperature increases, room environment plays an important role in printing quality. If the room temperature cannot be controlled, the solder paste vendor should be consulted.
8. Heat generated by screen printer	After prolonged periods of use, some screen printers generate heat which affects the solder paste and stencil. This heat generation reduces the solder paste viscosity and changes print quality. Consult the equipment manufacturer for recommendations or the solder paste vendor for a solder paste designed for higher operating environments.
9. Squeegee pressure too low	Unless sufficient squeegee pressure is applied, the stencil will not come into contact with the circuit board and solder paste will not be cleaned thoroughly from the stencil surface. With the stencil not in contact with the board, excess solder paste is extruded into an area between the stencil and the board. In addition, any added thickness of solder paste left directly above the stencil holes is deposited.
10. Squeegee blade angle too small	With flat blade squeegees, as the angle of the blade to stencil plane decreases, the amount of deformation of the blade increases. Excess solder paste will be deposited if the blade is not able to push the stencil into contact with the board or wipe the stencil clean.

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Incorrect solder particle size	As the size of the stencil openings decrease, the size of the solder particles in the paste should also be decreased. Solder particles having a diameter in the range of 45–75 microns are recommended for most applications where stencil openings are greater than 14 mils wide. Although it is possible to print through smaller openings with the large particle size, it is easier using smaller particles. Most solder paste vendors can provide several different particle size ranges depending on the needs of the application. If good solder paste deposit definition is seen on the large pads, but not on the small pads, check to make sure that the proper particle size solder paste is being used.
2. Viscosity too low	Most higher viscosity solder pastes leave more sharply defined solder paste deposits than low viscosity solder pastes. If the solder paste viscosity is too low, it may be difficult to control the quantity of solder paste deposited and wet bridging. Solder paste viscosity decreases as temperature and shear on the solder paste increases. It may not be necessary to change solder pastes, only reduce the operating temperature or squeegee speed.
3. Solder paste too tacky	Although a solder paste must have sufficient tackiness to roll in from the squeegee blade, the solder paste should have a greater tendency to stick to itself rather than to the stencil. This helps solder paste release cleanly from inside the stencil walls as the stencil lifts away from the circuit board. If the solder paste is too tacky, it will be very difficult to control print quality and the occurrence of wet bridging may increase. Tackiness is most easily controlled by the solder paste vendor. It is best to consult the solder paste vendor for material recommendations. They may not always be aware of the specific requirements of one's process.

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
4. Board warpage	Non-planar circuit boards make printing very difficult, because the stencil has trouble making intimate contact with the board. The gaps between the stencil and circuit board may fill up with solder paste resulting in a smear, wet bridge, or excess deposit. Most screen printers are equipped with a vacuum plate which can test and compensate for minor board warpage. If a tight seal is achieved between the vacuum plate and the board, then the circuit board is generally planar enough to get a good print. Off-contact printing with increased squeegee pressure also helps compensate for board warpage.
5. Inconsistent circuit board surface	Even though a circuit board may be planar, it may still be difficult for the stencil to come into contact with all segments of the board. It is rare that one will find a circuit board which contains soldermask and has a perfectly smooth surface. Some types of mask, such as dry film, tend to have smoother surfaces than others, but there is still no guarantee. If the stencil cannot come into contact with the circuit board, then the incidence of excess solder, wet bridges, and smeared prints increases.
6. Circuit board pattern misalignment	It is very common to see the pads on a circuit board vary location by several mils. This does not normally cause difficulties when working with 50 mil center devices or chip components, but can cause problems when doing fine pitch assembly. Very little can be done short of using a vision system to realign the stencil for every print. If a vision system is not available, the other effects that contribute to misalignment should be reduced by decreasing snap-off distance and squeegee pressure.

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
7. Lack of soldermask between pads	Most circuit boards have soldermask separating every pad. Apart from the electrical considerations, soldermask performs several other important functions. It helps make a seal between the stencil and circuit board and forms a well into which the solder paste can be deposited. It is common to observe inconsistent soldermask height between adjacent fine pitch pads. Some circuit boards are designed with no soldermask between close pads, making it virtually impossible to guarantee a clean, consistent print.
8. Inconsistent solder pad height	It is common to find hot air leveled boards with solder deposit heights varying from 1–3 mils. The variation can occur from pad to pad or between different areas of the same pad. Inconsistent height is a problem, because stencil openings are not normally the same size as the pad and some part of the stencil usually comes in contact with the solder deposit, leaving other areas of the stencil away from the board. Solder paste is then squeezed under the stencil in the areas that do not meet the board. Pattern misalignment, stencil misalignment, and stencil movement all increase this problem by moving the edge of a stencil opening closer to the pad center where the deposit is generally thicker. Circuit boards with bare copper or thin solderable coatings are the easiest to print solder paste onto, because they have the most level surface.

5.1.3 Observed Condition: Solder Paste Will Not Roll in Front of the Squeegee If the solder paste does not roll in front of the squeegee, then it is more difficult to fill the holes in the stencil. The solder paste may skip over the holes.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
No information available	

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Too little solder paste on the stencil	In general, solder paste will adhere to itself rather than adhere to anything else. A minimum amount is required to allow the solder paste to come in contact with both the squeegee and board without adhering to one or the other.
2. Insufficient solder paste tackiness	If tackiness is too low, then the solder paste will not be able to adhere to the stencil. If the solder paste cannot adhere to the stencil, then it will slide over the stencil rather than roll.
3. Viscosity too high	Viscosity should be balanced with tackiness to yield solder paste with the best printing characteristics possible. If solder paste viscosity is too high in combination with low tackiness, the solder paste will slide over the stencil rather than roll.

5.1.4 Observed Condition: Solder Paste Adheres to the Squeegee at the End of the Stroke In order for either diamond or flat blade squeegees to perform in a bi-directional print mode they must be able to lift cleanly away from the solder paste at the end of the stroke. Solder paste must remain on the stencil for the return pass.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
No information available	
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Insufficient solder paste on the stencil	Most solder pastes will adhere better to themselves rather than a squeegee blade. If too little solder paste is used, there will be insufficient mass to break the adherence to the blade.
2. Solder paste viscosity too high	If solder paste slides over the stencil and adheres to the blade, then there is a good chance that the viscosity is too high for the application. High viscosity may be a result of the formulation, inadequate shear from the squeegee, or low working temperature.

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
3. Solder paste is too stringy for application	Although some applications favor stringy solder paste, it is generally undesirable for stencil printing. The solder paste vendor should be consulted.

5.1.5 Observed Condition: Solder Paste Hanging Up in the Stencil Holes A balance of properties: tackiness, viscosity, stencil design, and print parameters contribute to the solder paste's ability to release cleanly from the stencil holes. Rarely is there only one cause.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Stencil too thick	Stencils which have a high stencil thickness to opening width ratio are difficult to print with. Most solder pastes will adhere to the walls of the stencil hole. If the force holding the solder paste to the walls of the hole is greater than the force between the paste and the circuit board, there will not be a clean and consistent print. Stencil thickness must be decreased as the dimensions of the stencil hole are decreased.
2. Snap-off distance too small	Cone-shaped deposits are caused by the solder paste adhering to the stencil hole walls as the stencil is removed from the circuit board. A larger snap-off distance can cure the problem by pulling the solder paste out of the hole before it has a chance to set up. Higher viscosity and lower tackiness solder pastes also print more cleanly.
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Solder paste viscosity too high	Solder paste with a viscosity too high for the application may have difficulty being forced to the bottom of the stencil hole or may set up and have difficulty transferring.
2. Improper solder paste tackiness	Solder pastes can have trouble printing if they are either too tacky or not tacky enough. A solder paste which is not tacky enough will not roll. A solder paste which is too tacky will adhere too strongly to the walls of the stencil hole and not transfer cleanly.
3. Solder paste particle size too large	The smaller the size of the solder powder used to make the solder paste, the easier the solder paste will print, because there is less friction between particles. Disadvantages to using very small solder particles include greater solder paste slump and increased occurrence of solderballs. It is best to use the largest particle size available which will still print cleanly.

5.1.6 Observed Condition: Lateral Deposit Skips It is very common to have poor print quality in only one section of the circuit board while the rest of the circuit board is printed cleanly. A number of factors can contribute to this.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Improperly made stencil	Make sure that the stencil is not too thick for the width of the opening being printed. If solder paste is continually hanging up in the hole then this is a strong possibility. Also check the area which is printing poorly for signs of inconsistent etch. If the stencil is off center in the frame, then adjustments should be made to print technique.
2. Damaged stencil	Examine the area of the stencil which is having consistent problems. If the stencil is bent, buckled, or dented (even slightly), then this may be the cause of the problem. A decision must be made, whether to accept the poor quality print or use a new stencil.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
3. Inconsistent vacuum	Most manufacturers will use a vacuum plate to hold the circuit board in place and compensate for minor non-planarity. If there is inadequate vacuum, inconsistent vacuum, or a vacuum leak, then part of the board may not be level. Make sure that no vacuum holes are in the same location as via holes in the circuit board.
4. Squeegee blade not level	Some squeegees have fixed positions, while others rock or float in order to remain parallel with the stencil. If one side of the stencil is not wiping clean, then the squeegee may be level. If large streaks of solder paste remain on the stencil, then the squeegee blade may be damaged or else there may be something peculiar about the way the squeegee blade is fixed in the squeegee blade holder. Some squeegee blade holders use screws or hex bolts to hold the blade securely. When tightened, the bolts may put pressure on the soft rubber squeegee blade causing it to deform. As this happens, the cross-section of the blade elongates in the area of the bolts and the squeegee blade edge becomes non-linear. Since some areas of the blade protrude farther than others, greater pressure is put on the stencil in these areas.
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Circuit board not planar	If the circuit board is warped, then it is very likely that certain areas of the board will be printed differently than others.

5.1.7 Observed Condition: Inconsistent Print When the Squeegee Travels in Opposite Directions Although bi-directional printing should yield the same results when traveling in either direction, this is not always the case. In most cases, the inconsistency can be traced to print parameters.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Inconsistent squeegee speed	Although the squeegee travel speed is supposed to be identical when traveling in either direction of bi-directional print, this is not always the case. Time the speed of the squeegee to make sure that the speeds are identical. With machines that have independent speed settings for both directions, it will be easy to adjust the speed so that it is consistent. If independent settings cannot be made and the speeds are not identical, then the equipment manufacturer should be consulted.
2. Inconsistent travel distance	In order to get a consistent print, particularly when using off-contact print, the distance the squeegee travels past the stencil pattern should be identical in both directions. This will reproduce the amount of stencil peel and solder paste set-up time in both directions.
3. Inconsistent squeegee angle	Independent adjustments to squeegee angle can be made on screen printers having a double flat blade system. Independent blades are used to print in either direction. In a screen printer, which has a single-diamond blade squeegee, problems may arise when the squeegee is not tight or some of the fixturing parts have begun to wear. It is not uncommon to find the angle of the squeegee blade changing slightly when printing in opposite directions as the print head rocks in a worn or loose fixture.

5.2 Screen Printing Screen printing is different from stencil printing. A mesh screen, blocked off in the areas not to be printed by a polymer emulsion, is used in place of a metal mask stencil. Screen printing always requires the use of a snap-off and has different limitations in the type of circuitry which can be printed. It is difficult to print solder paste onto very large land areas with a stencil and, conversely, difficult to print small areas with a screen. Softer squeegees are used when screen printing, because the solder pastes are less viscous, more conformance is required, and screens are easily damaged. Most of the mechanics of screen printing are identical with stencil printing with the exceptions mentioned below.

5.2.1 Observed Condition: Insufficient Solder Paste Deposited To eliminate insufficient solder or open circuits caused by too little solder paste deposited in the printing operation.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Insufficient emulsion thickness	In screen printing, the thickness of print should be approximately equal to the thickness of the emulsion, plus the thickness of the screen weave. Small increases or decreases in this value can be achieved by either changing snap-off distance or squeegee pressure. Large changes in print thickness must be done by increasing the amount of emulsion.
2. Insufficient snap-off distance	In order for solder paste to transfer cleanly from the screen to the circuit board, the solder paste must be pulled out of the screen as quickly as possible with the least amount of resistance. By using a large snap-off distance, the solder paste starts to pull out of the screen as soon as the squeegee passes.
3. Wire blocking opening in a very small hole	When printing fine lines using a screen, it is difficult to get around the problem of wires in the way. If the area to be printed is less than two screen openings wide, there is a very good chance that the average open area will be constricted to less than one opening wide. This may result in poor solder paste deposits, because the particles have difficulty fitting through the opening.
4. Emulsion too thick for application	Just as with stencil printing, if the thickness of the screen is too great for narrow openings, the solder paste will hang up in the openings and not transfer completely. Increasing the snap-off distance will help somewhat, but only to a point. The width of the opening may have to be increased in relation to the thickness of the emulsion.
5. Improper screen manufacture	Occasionally openings in the screen are not complete due to improperly placed emulsion. If one consistently gets insufficient solder in one particular area of the pattern, the screen should be inspected for defects.
6. Damage to screen	Screens are more fragile than stencils and wear more rapidly with use. Damage to the screen which restricts solder paste deposits includes: bent wires, swollen emulsion from solvents in the solder paste or cleaning material, and large particles of solder, squeegee, or other material permanently wedged in the screen openings.
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Solder paste too viscous	Solder pastes used in screen printing have a lower viscosity than those used with stencils. This is because there is greater friction between the solder paste and screen, due to the larger surface area the solder paste is being squeezed past. Thicker solder pastes have a tendency to set up in the screen weave and not transfer consistent amounts of material.
2. Solder particles too large for the screen used	There is a physical limitation to the size particles that will fit through a screen. Although in theory, solder particles will go through any opening larger than they are, this is not a reasonable expectation in a production environment. Solder particles must pass smoothly and quickly through the screen in order to achieve a consistent print. Solder particles should be no larger than one half the average diameter of the openings.
3. Solder paste metal content too low	The most common method of decreasing solder paste viscosity to facilitate screen printing is to decrease the metal content. In doing this the volume of solder after reflow is also decreased. This may result in insufficient fillet size.

5.2.2 Observed Condition: Excess Solder Paste Deposited Excess solder after reflow or solder shorts caused by excess solder paste deposits and wet bridging.

Excess solder and solder shorts occur when too much solder paste is deposited. As with stencil printing, there are several mechanisms for excess solder deposits.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Emulsion too thick	Emulsions that are too thick may cause excess solder paste to be transferred to the circuit board. The nominal thickness of solder paste deposited is equal to the screen weave thickness plus the emulsion thickness.
2. Excess snap-off or insufficient squeegee pressure Excessive snap-off distance will lead to distortion in the printed pattern and misalignment.	It is possible to control the amount of solder paste transferred by altering the snap-off distance and squeegee pressure. By using a high snap-off in combination with a low squeegee pressure, it is possible to extrude the low viscosity solder paste used in screen printing to a greater thickness than would normally be expected. Conversely, if one is getting a greater thickness than is desired, then one should decrease the snap-off distance and increase the squeegee pressure.
3. Improper screen design or manufacture	If bridging or excess solder is noticed in only one particular print area, then the most common cause would be a defect in stencil design manufacture. Look for areas which should have emulsion, but do not.
4. Damaged screens	It is not uncommon for a screen to become damaged through normal use. Scratches, holes, bent screen, etc., can all contribute to excess solder deposits. Very little can be done with the exception of installing a new screen.

5.2.3 Observed Condition: Misaligned Solder Paste Deposits

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Screen stretching	Screens are constantly being stretched, because they are almost exclusively used in an off-contact mode. Over time, the screen material will fatigue and it becomes more difficult to control the print alignment, particularly when printing bi-directionally. Very little can be done, with the exception of decreasing snap-off distance and squeegee pressure. This reduces the amount of friction between the squeegee and screen.
2. Excess snap-off or insufficient squeegee pressure	Excessive snap-off distance will lead to distortion in the printed pattern and misalignment.
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
No information available	

5.3 Syringe Dispensing

5.3 Syringe Dispensing Syringes are used both in repair operations and programmable multipoint dispensing. Syringes used in repair are generally hand-held and employ air or manual pressure to dispense the solder paste. Multipoint dispensers use air or other forms of pressure to dispense solder paste to points programmed into an X-Y table.

In syringe dispensing applications, the most important things are continuous dispensing and consistent sized deposits.

5.3.1 Observed Condition: Solder Paste Flo Stops

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Dispensing pressure too high	Dispensing systems which use pressure on the back of the syringe to push the paste out of the needle require special consideration. Solder paste responds best to low pressure and longer shot times, as opposed to high pressure and short shot times. The high pressure is more likely to separate the flux from the solder paste.
2. Solder paste separation	Some applications appear to separate the solder paste regardless of what is done. This is common when very small diameter needles must be used. Some manufacturers have found that it helps to shorten the length of the needle in order to reduce the amount of friction. Other forms of dispensing equipment have been developed which do not stress the solder paste as much as direct pressure dispensing. These methods use variations of pinch valves or pistons to apply high pressure to very small amounts of solder paste.

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Solder paste stops coming out of the tube before it is empty	Once solder paste stops coming out of the tube, it is very difficult to continue using that tube. Separation of the flux from the powder has already started to occur. It may be possible to increase the dispensing pressure to get some solder paste out, but this will only work to a point. It is far better to use a preventative method to stop initial separation.
2. Incorrect solder paste formulation for dispensing	Many solder paste manufacturers have different solder pastes for stencil printing and syringe dispensing. The rheological properties may be very different. A solder paste designed for syringe dispensing will have a consistent viscosity under different shear rates. In addition, the percent metal by weight is generally lower to give the solder paste added lubricity.
3. Solder powder particles too large for needle size used	If the solder powder particles are too large for the needle used, they will bind in the needle and prevent the paste from coming out. Smaller solder powder particles or a larger needle can be used.

5.3.2 Observed Condition: Inconsistent Solder Paste Size Solder paste deposit size is not generally a concern in repair operations, because the amount of paste deposited can be determined by the operator. But how consistently a solder paste deposits during an automated syringe dispensing application will have a bearing on yield quality.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
No information available	

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Incorrect solder paste formulation	Consult the solder paste vendor if the dispenser has difficulty delivering consistent amounts with each shot. The solder paste may not be designed for the dispensing application or there may be air entrapped in the solder paste. A sure sign of inconsistent solder paste deposits is if the solder paste strings when the dispensing head moves from one pad to another. This is generally caused by too much solder paste being pulled out of the needle and will leave the next deposit short of solder paste.

5.3.3 Observed Condition: Solder Paste Deposit Not Rounded Enough

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Improper needle tip design	Use a beveled needle, which dispenses solder paste to one side of the needle, not directly under the needle.
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Solder formulation not tacky enough and doesn't slump	Use a solder paste which self-rounds or slumps slightly.

5.3.4 Observed Condition: Solder Paste Deposit Not Peaked Enough

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Improper needle tip design.	Use a flat-tipped needle.
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Solder formulation too tacky and slumps	Use a different formulation which is less tacky and doesn't slump.

5.3.5 Observed Condition: Solder Paste Will Not Adhere to the Pad

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Too high a pressure	Use a lower pressure for dispensing.
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Incorrect solder formulation	Use a different formulation that is more tacky.

6.0 COMPONENT PLACEMENT

Surveys have indicated that a major portion of soldered assembly defects result from manufacturing operations prior to soldering. Component placement has been identified as the most prevalent source of these defects. This has become increasingly evident with the increased use of fine pitch technology.

This section presents a general listing of problems encountered in the component placement process, potential causes and suggestions for corrective actions. These solutions may be related to equipment, materials, design or prior processes. Accordingly some of the solutions cannot be immediately implemented on the shop floor.

6.1 Observed Condition: Wrong Component Orientation

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Placement program problems	Verify correct revisions Add rotational data to program Change rotational data in program
2. Different board/program version	Verify correct version
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Components in feeder have wrong orientation	Correct orientation of components
2. Components have mixed orientations at pickup location	Orientate components in feeder
3. Board marking incorrect	Verify orientation with both assembly drawing and schematic

6.2 Observed Condition: Wrong component placed on board

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Wrong pickup callout	Verify/change pickup location
2. Incorrect sequence step order	Verify sequence program
3. Different board/program version	Verify correct board/program version
4. Incorrect placement location	Verify/change placement location
5. Component from previous pickup stuck to chuck	Clean chuck; vacuum pickup tip
6. Incorrect data file used	Verify correct program
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Wrong component put into feeder	Inspect/verify components in feeders
2. Wrong components mixed in with correct parts	Inspect/verify components numbers
3. Parts list is incorrect	Update program with latest parts list

6.3 Observed Condition: Missing Component

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Vacuum pickup not on	Verify there is vacuum at pickup
2. No components at pickup location	Restock feeder
3. Component not picked up	Reprogram for correct pick-up
4. Feeder malfunction a) Low/off air pressure b) No electricity c) Waste tape is clogging d) Excessive vibation or low vibration e) Excessive vibration or low vibration	 Verify correct pressure/ vacuum at feeder Verify power is turned on at feeder and verify feeder is correctly connected Remove excess tape and re-thread Remove jammed component Adjust vibratory feeder vibration control
5. Incorrect chuck used	Modify program to use correct chuck
6. No placement data	Add placement data to program
7. Incorrect placement location	Reteach correct placement data
8. Component did not release from chuck	Clean chuck and/or check for vacuum release
9. Different board/program version	Verify correct board/program version

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
No information available	

6.4 Observed Condition: Component Misalignment

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Placement teach is not accurate	Reteach placement
2. Board is misregistered	Verify board loading equipment and tooling
3. Incorrect pickup chuck selected	Modify program to use correct chuck for pickup or may require vision alignment
4. Component is rotated	Verify correct rotation data in placement program
5. Board revision different than program	Verify program versions
6. Equipment needs calibration	Calibrate to manufacturers maintenance instructions Check pulleys/ gears for slippage on shafts
7. Incorrect data file used	Load correct data file
8. Machine placement tolerance is not adequate for accurate/acceptable placement	Equipment cannot be used for the required accuracy of placement. Acquire equipment with an acceptable placement accuracy.
9. Vision system is not working	Check electrical connections and fuses: a) Check target sizes/ composition; verify compliance. b) Camera is not looking at the component heads; software problem. Verify component within field.
10. Placement teach is above board surface	Reteach placement
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Board is misregistered	Verify fiducials
2. Incorrect footprint on board	Verify with design standards and/or latest parts list and schematic
3. Incorrect component picked up	Verify with parts list and assembly drawing
4. Component is rotated	Review component loading process
5. Board revision different than program	Verify board version

6.5 Observed Condition: Damaged Component

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Pickup/placement head damaged component	a) Verify pickup and component placement. Z-axis locations are not impacting components; adjust height of pickup/placement. b) Pickup head is not correct for particular component; use pickup head designed for that component type/style.
2. Component leads are bent, or missing	a) Adjust pressure of damaged, or missing “square off” fingers. b) Verify that pickup head is correct for that component. c) Fragile leads require vision alignment, not mechanical alignment.
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Component was damaged prior to placement	Inspect components prior to placement. Discard damaged components.
2. Component leads are bent, damaged or missing	a) Verify at incoming or missing inspection b) Improve handling procedures c) Review excising die

7.0 INFRARED/CONVECTION REFLOW SOLDERING

This process is generally carried out with inline conveyorized equipment incorporating multiple heating zones with individual temperature controls. Heat sources used include short wavelength tubes, medium-to-longer wavelength panel secondary emitters, and/or heated air/nitrogen in the convection reflow systems.

In many cases convection reflow systems have overcome some of the perceived drawbacks of this reflow process which included: (1) the need for individual assembly processing profiler as a result of differences in mass and component characteristics, (2) shadowing of some areas from exposure to Infrared Reflow (IR) energy as a result of component size or spacing, and (3) the lack of fast response to set temperature changes.

As with any mass soldering process, defects detected in the soldered assemblies are often related to prior process and material variables. The defects that relate to prior process or material are much larger in number and can be very difficult to classify after the fact. For example, once the assembly is soldered, it is hard to determine whether a component was out of position before reflow, or whether the solder paste deposit was properly registered.

There are some defects which may not make themselves known immediately, such as cracks in chip capacitors or flux residue under parts. Tight process control is a necessity to prevent these defects, as it is far too late to start corrective action when the defects don't appear until the product built is in service.

This section presents a general listing of problems encountered in the infrared/convection reflow soldering process, potential causes and suggestions for corrective actions. These solutions may be related to equipment, materials, design or prior processes. Accordingly some of the solutions cannot be immediately implemented on the shop floor.

7.1 Observed Condition: No Reflow

7.1.1 Observed Condition: No Reflow of Solder Paste/ Cold Solder Joints

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Reflow profile incorrect	Increase/change reflow temperature
2. Thermal load too great	Decrease product loading for machine capacity rate
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Incorrect solder paste alloy	Use the solder alloy specified

7.1.2 Observed Condition: No Solders No solders may be the result of a non-reflow condition or they may be due to problems associated with placement, solder paste application, solderability of the parts, or errors in the design of the assembly that make reflow of some of the joints very difficult. Thus a ''no solder'' may be defined as a condition:

1. In which solder is completely absent from the intended area.
2. Where the solder paste is present, but reflow has not occurred.
3. Where solder is present and has reflowed, but has not wet either the lead, the board, or both.
4. Where the solder is present, has reflowed, has wet the board/lead, and pulled back away from the solder joint, i.e., dewetting.

7.1.2.1 Observed Condition: No Solder—Solder Not Present

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
No information available	
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Clogged stencil/dispenser	Clean stencil/dispenser (see section 5)
2. Print process out of control (See Section 5.0 for more detail)	Adjust squeegee pressure or durometer Change print speed Check stencil alignment Check dispenser accuracy (see section 5)
3. Paste drying out (See Section 5.0 for more detail)	Change process to reduce time paste remains on stencil Increase humidity and reduce drafts in the screen printing area Do not reuse old paste (see section 5)
4. Solder paste viscosity too high	Obtain lower viscosity paste (see section 5)
5. Paste not "wetting" the stencil	Change paste (see section 5)
6. Solder clogging stencil/ dispenser	Use different particle size paste Check paste for unevenness of texture Use different viscosity paste Use paste with slower drying time (see section 5)

7.1.2.2 Observed Condition: No Solder: Solder not Reflowed

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Reflow temperature not high enough	Adjust profile Increase time above liquidus Verify that all heaters are working
2. Machine being thermally overloaded	Reduce rate of product input to match equipment thermal capacity and recovery capability
3. Board layout causing shadowing	Reorientation of board on conveyor Change reflow profile Select alternate reflow process
4. Heat sinking from board/ parts	Change reflow profile Select alternate reflow process Increase preheating stage

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Incorrect solder alloy	Use solder alloy specified
2. Board layout causing shadowing	Change component selection and placement
3. Heat sinking from board/ parts	Reorientation of board on conveyor Change reflow profile Select alternate reflow process

7.1.2.3 Observed Condition: No Solders: No Wetting

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
No information available	
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Lead not wetting	Check part solderability (see J-STD-002) More active paste may be helpful
2. Board not wetting	Check board solderability (see J-STD-003) More active paste may be helpful
3. Part may not be in contact with paste	Check print accuracy Check part placement (see section 6.0) Check lead planarity/skew Check board warp

7.1.2.4 Observed Condition: No Solders: Dewetting

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Excess time above reflow temperature	Change profile to reduce time above reflow
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Board has excess intermetallics	Increase solder thickness on board Reduce storage time Improve storage condition
2. Chip termination has poor leach resistance	Use parts that have barrier layers (see J-STD-002)

7.1.3 Observed Condition: Uneven Reflow Sometimes an assembly will reflow in some sections and not in others. Reasons for this may be similar to the ones for cold and no solders discussed above, i.e., thermal or solderability.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Incorrect profile for assembly	Check profile, customize for each assembly
2. Poor heat transfer	Check positioning of board to insure even heat transfer
3. Uneven heat transfer across oven	Increase convective air across oven flow in oven
4. Other effects	Check items under 7.1.2 above
5. Thermal capacity and recovery capability	Repair or replace oven
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Board layout causing shadowing	Design change required
2. Heat sinking from board/ parts	Design change required
3. Other effects	Check items under 7.1.2

7.2 Observed Condition: Solder Balls Solder balls are often due to excessive outgassing of the paste or boards during reflow or an excess of oxide on the paste particles, components, or boards.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Incomplete paste cure	Pre-cure paste or change preheat portion of reflow profile for cure
2. Conveyor speed too high	Reduce conveyor speed
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Incomplete paste cure	Change paste
2. Moisture in boards	Pre-bake boards
3. Entrapped solvents in boards	Pre-bake boards
4. Oxidized paste	Refer to J-STD-005 for incoming paste controls
5. Solder mask/paste interaction	Change in paste or mask needed

7.3 Observed Condition: Charring of Board/ Components Thermal damage to the boards or components during reflow can be due to oven temperature or uneven thermal transfer during reflow.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Oven temperature too high	Reduce temperature set-points Shield affected areas from direct radiation Increase conveyor speed
2. Loading rate too low for oven settings	Check/change rate
3. Uneven/incorrect positioning of assemblies in reflow	Check positioning to insure heat transfer
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
No information available	

7.4 Observed Condition: Damaged Components Damaged components are a subset of thermal damage in general. Thermal damage to the components may be either charring (see 7.3), melting, or cracking. As with most mass soldering techniques, it has been found that preheat is an essential part of the process. In order to avoid cracking components, an acceptable ramp rate should be established for sensitive compoents.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Temperatures too high	Change profile Keep temperatures below limit for components
2. Components being damaged in pick and place operation (see Section 6.0)	Adjust pick and place equipment
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Components incompatible with IR reflow	Change components for compatibility
2. Moisture in packages: cracking	Observe handling procedures in IPC-SM-786

7.5 Observed Condition: Cracked Joints Physical damage to the solder joints can occur if the assemblies are moved while the solder joints are still liquid or if the assemblies are handled in such a way as to damage the cooled solder joints, e.g., dropped. Stress after reflow, not during, is the most common cause of cracked joints.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Insufficient cool down after reflow	Change thermal profile
2. Too rapid of cool down after reflow, e.g., immediate transfer to cleaner	Change thermal profile
3. Dropping of board after oven	Change handling procedures

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Attempts to straighten warped board	Change handling procedures

7.6 Observed Condition: Insufficient Solder Insufficient solder at the solder joint may be due to the paste/paste deposition process. It may also be due to the design of the assembly or solderability of the parts. Some of these items are treated under 7.1.2.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
No information available	
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Poor paste printing	See 7.1.2 Also see Section 5.0
2. Paste clogging stencil or dispenser	See 7.1.2 Also see Section 5.0
3. Solder wicking away from solder joint	Adjust design for lead/land ratio or land design Change in paste/flux activation (lower) Move via hole

7.7 Observed Condition: Bridging Bridging can be the result of board design, e.g., insufficient clearance between features; poor solder deposition process, e.g., paste smearing; incorrect part placement or misalignment; or damage to the parts, e.g., bent leads.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
No information available	
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Paste deposition process causing bridges	Check process (see Section 5.0)
2. Bent or misaligned leads	Eliminate out-of-tolerance components
3. Paste slump	See Section 5.0 and J-STD-005
4. Component misplacement	See Section 6.0
5. Design spacing	Review layout for possible change

7.8 Observed Condition: Voids Voids are the result of the entrapment of flux or other volatiles in the solder joints during reflow.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Insufficient paste cure	Modify cure or reflow profile Change paste

B. Material/Prior-Process Related	
Potential Cause	Corrective Action
1. Paste solvent entrapment	Change paste for different volatility

7.9 Observed Condition: Opens Opens may be the result of a physical defect in the part, i.e., no physical contact between part and substrate, or a no solder situation (see 7.1.2).

A. Equipment/Process Related	
Potential Cause	Corrective Action
1. No reflow of solder	See 7.1.2
B. Material/Prior-Process Related	
Potential Cause	Corrective Action
1. No or insufficient paste deposition	Adjust paste deposition process (see Section 5.0 and 7.1.2)
2. Inaccurate parts placement	Adjust placement (see Section 6.0)
3. Poor board/part solderability	Check/control solderability of boards and parts (see J-STD-002 and J-STD-003)
4. Parts with lead skew or planarity problems	Use only “in-tolerance” parts Review component placement (see Sections 2.0 and 6.0)

7.10 Observed Condition: Spatter Spatter is related to solder balling (see 7.2). Accordingly, it may be caused by the paste, the boards (see above), or the paste deposition process (see Section 5.0).

A. Equipment/Process Related	
Potential Cause	Corrective Action
1. Smearing in printing	Adjust printing process
2. Printing of paste on non-solderable surface	Check for alignment of printing
3. Paste not cured	Check cure and/or change reflow profile
B. Material/Prior-Process Related	
Potential Cause	Corrective Action
1. Moisture in paste	Store paste only in dry conditions
2. Moisture/solvents trapped in board	Pre-bake board

7.11 Observed Condition: Component Alignment Component alignment affects both the shape of the final solder joint and the tendency of the solder operation to form solder bridges. Many of these issues have been addressed above (and in other sections, i.e., 5.0 and 6.0). It should be noted, however, that as pitch on surface mount parts decreases, the need for reduced tolerance in placement increases. In addition, post placement, e.g., operator touch up and handling after reflow (see above), influence part alignment.

7.11.1 Observed Condition: Tombstoning Tombstoning is the raising up of one end of a chip component during reflow. The tendency of parts to do this can be related to design, the solder operation, part/board solderability, or differential heat transfer during reflow.

A. Equipment/Process Related	
Potential Cause	Corrective Action
1. Excess movement in reflow operation	Reduce board motion during reflow
B. Material/Prior-Process Related	
Potential Cause	Corrective Action
1. Inaccurate placement of parts	Adjust placement
2. Uneven solderability on pads or leads	Check for solderability variations
3. Pad design incorrect	Redesign required (see IPC-SM-782)

7.12 Observed Condition: Delamination Delamination of the boards during reflow will occur if the board quality is poor or if the process is excessive in temperature. Checks of incoming quality and process parameters are the two approaches to addressing this problem. It must also be noted that excess temperatures or repeated reflow operations can induce delamination.

7.13 Observed Condition: Warp Excessive warp can cause unsatisfactory solder joints. Warp may be a result of the board manufacturing process. It is important that it be determined if boards are warped when they are received or if they deform during the process.

7.14 Observed Condition: Disturbed Joints Disturbed joints are similar to the cracked joint phenomena discussed above. The major considerations are the handling of the board, either during or after the reflow process.

7.15 Observed Condition: Dewetting Dewetting is associated with the solderability characteristics of the components. The solderability is affected by storage conditions, basic metallurgy of the surface, the reflow process, flux used, and composition of the solder used. A discussion of the various causes has been given in 7.1.2.

8.0 VAPOR PHASE REFLOW SOLDERING

The vapor phase process has some unique characteristics. It has a fixed upper temperature determined by the boiling point of the fluid used. It is a rapid heating method and quite tolerant of large mass differentials on a given circuit board or assembly. It is a very uniform method of heating as the vapor surrounds the components, heating from all sides. As with most mass soldering techniques, it has been found that preheat is an essential part of the process and typically this is provided with an infrared preheater. The accepted standard for preheat is a 2 to 3°C per second temperature rise to within 100°C of the reflow temperature, followed by a short stabilization and equalization period prior to reflow.

To aid cooling of the assembly through liquidus, and to improve solder grain density and shorten liquidus time, many machines have a convection cooldown system integral to the equipment.

8.1 General As with any mass soldering process, defects detected in the soldered assemblies are often related to prior process and material variables. The defects that relate to prior process or material are much larger in number and can be very difficult to classify after the fact. That is, once the assembly is soldered, it is hard to determine things like whether a component was out of position before reflow, or whether the solder paste deposit was properly registered.

There are some defects which may not make themselves known immediately, such as cracks in chip capacitors or flux residue under parts. Tight process control is a necessity to prevent these defects, as it is far too late to start corrective action when the defects don't appear until the product built is in service.

8.2 Problems This section presents a general listing of problems encountered in the vapor phase reflow soldering process, potential causes and suggestions for corrective actions. These solutions may be related to equipment, materials, design or prior processes. Accordingly, some of the solutions cannot be immediately implemented on the shop floor.

Differences in machine construction can in some cases cause special situations, i.e., no preheat in vertical batch units. Where appropriate, machine specific items have been listed under ''Equipment/Process Related.''

8.2.1 Observed Condition: No Reflow of Paste or Preforms Most frequently related to reflow time.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Inadequate reflow power	Increase power setting Verify that all heaters are working
2. Inadequate reflow time	Reduce conveyor speed Increase dwell
3. Boiling temperature too low	Check for correct fluid temperature for application
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Solder melting point too high	Check solder paste composition for correct melt temperature

8.2.2 Observed Condition: Dull, graying, crystallized solder Most frequently related to material.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Excessive time in preheat causing solder oxidation	Reduce preheat time
2. Excessive time in reflow causing excessive leaching	Reduce reflow time
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Excessive leaching of passive component terminations	Select components with nickel barrier
2. Gold not removed from active components	Solder coat components
3. Board plating too far from eutectic	Check board plating composition
4. Oxidized component terminations	Reject components

8.2.3 Observed Condition: Opens Most frequently caused by lead coplanarity or solder wicking.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Wicking of solder to fine leads pulling solder away from joint	Increase preheat temperature and/or time from joint (125 to 150°C). Vertical batch use 2% silver or special slow melt solder to slow solder melt
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Solder wicking to adjacent pad or down feed through hole	Correct design deficiencies
2. Poor lead coplanarity	Reject or reform components
3. Misregistered solder paste	Correct application registration
4. Insufficient solder paste	Check for plugged screen or stencil

8.2.4 Observed Condition: Bridges Most frequently caused by solder application registration tolerance.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
No information available	
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Misregistered solder paste print	Correct application registration
2. Too much solder	Check solder application
3. Solder slumping	Check solder paste
4. Inadequate pad separation	Correct design

8.2.5 Observed Condition: Tombstoning Most frequently caused by unequal melt or solder paste quantity.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. One side of component melting before the other	Increase preheat and/or time at preheat
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Unequal pad geometrics cause unequal melt times	Correct design
2. Solder mask under part causing teeter-totter effect	Correct design
3. Unequal pad heating caused by leads or ground plane	Correct design
4. Pads extend beyond component too far so component can pull off one end	Correct design
5. Too much solder causes high tipping forces	Reduce stencil thickness or solder mask too thick
6. Misregistration of solder paste no paste on pad	Check registration and cleanliness of screen
7. Misregistration of component not on paste	Check component location at pick-and-place
8. Solder mask on pad (smear)	Reject fabricated board
9. Displaced components	Check operator or transport handling systems

8.2.6 Observed Condition: Solder Balls Most frequently caused by outdated paste or insufficient preheat.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Solder flux spattering in reflow	Increase preheat temperature and/or increase preheat time
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Oxidized solder paste	Use fresh solder paste
2. Misregistration of solder paste	Check application registration
3. Improperly cured solder mask	Correct condition at board fabrication

8.2.7 Observed Condition: Component Alignment Most frequently caused by pad design or placement error.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Component float to one pad	Increase preheat time and/or temperature
2. Conveyor vibration	Clean conveyor rails Check conveyor tension
3. Tunnel curtains (brush components)	Shorten curtains
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Components float on pads	Correct pad design
2. Components misplaced	Correct pick-and-place alignment
3. Unequal solder melt	Correct pad connection design

8.2.8 Observed Condition: Capacitor Cracking Most frequently caused by thermal shock or placement handling.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Possible thermal shock	Make sure preheat rate is 2 to 3°C per second to within 100°C of reflow Use preheat on vertical batch units (oven).
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Poor capacitor construction	Test capacitors for thermal shock
2. Capacitors cracking in placement	Check centering jaws and placement pressure
3. Post handling cracking	Do not bend boards after reflow

8.2.9 Observed Condition: Plastic Package Cracking Most frequently caused by water entrapped in plastic.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Excessive reflow temperature	Be sure fluid used is 215°C or below
2. No preheat	Set preheat to achieve 2 to 3°C per second rise to within 100°C of reflow
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Excessive water vapor pressure	Bake components or purchase pre-baked components (see IPC-SM-786)

8.2.10 Observed Condition: Low Surface Insulation Resistance Most frequently caused by flux entrapped under component.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Contaminated vapor phase soldering fluid	Run plain substrate and retest Replace or reclaim fluid if resistance too low
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Flux entrapped under component	Improve cleaning process Reform leads to increase clearance
2. Board contamination prior to reflow	Check incoming boards for contamination
3. Wrong flux specified for product type	Check flux and cleaning method for particular application

8.2.11 Observed Condition: Insufficient Solder Fillets Most frequently caused by insufficient solder paste.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Solder wicking up leads	Increase preheat time and/or temperature
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Too little solder paste	Increase size of stencil openings Increase stencil thickness *Check solder application*
2. Solder wicking away from joint on board	Correct design location of feed throughs Correct design of trace to pad connections Use solder mask if not now using

9.0 WAVE SOLDERING

Wave soldering of surface mount assemblies is easy to do providing that the boards and all components have solderable metallic surfaces; the board design meets recommended guidelines for pad layout geometry; the components have been properly placed using a proper adhesive which is correctly cured; and the wave soldering machine is under correct process control. This means using good flux at the correct specific gravity and solder with verified purity, selecting preheat settings, and matching the temperature of the solder wave with a conveyor speed so that this relationship provides sufficient heat for the correct time duration to perform acceptable solder joints. Verifying that all of these are under proper process control is the first overall requirement or guideline for mass production wave soldering of surface mount assemblies and for providing acceptable quality solder joints.

The guidelines are presented in order of priority where possible. Some process problems are easy to correct by the operator, others are more difficult. Some require a longer time period to resolve or need outside help, and some may be more expensive than others to correct.

9.1 Observed Condition: Solder Skips A solder skip occurs where a solder joint did not form. Solder skips are usually observed in random locations and may be caused by the wave soldering equipment, the materials, or poor process control. They can also result from an incorrectly performed prior assembly operation or the board design layout. If many joints do not solder, for example in a corner or specific area of the SMA, it is most probably related to the wave soldering equipment or to process steps prior to wave soldering. Solder skips in a random manner can be caused by any of the reasons suggested below.

A. Equipment Related	
Potential Causes	*Corrective Action*
1. Turbulent wave OFF	Turn Turbulent wave ON
2. Vibratory device in 2nd wave OFF	Turn ON vibratory wave device
3. Turbulent wave too low	Increase height of turbulent wave
4. Turbulent nozzle clogged	Clean flux or dross in nozzle
5. Carrier bent or damaged	Send carrier/fixture to repair
6. Conveyor fingers bent	Replace bent fingers
7. Rails not level or parallel over nozzles	Reset rails level/parallel
8. Waves/Nozzles not level	Reset pot and nozzles level Use LevChek
9. 1st Wave height lower in spots	Pull pump and nozzles Use LevChek Clean out pot
10. Conveyor width too tight	Adjust conveyor width
11. Conveyor speed too fast for flux volatiles	Reduce conveyor speed
12. Board not run best direction (orientation)	Turn SMA 90, 180 or 270 degrees
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Flux solids too high (15–35%)	Change to low solids flux (1–3%)
2. Flux specific gravity gone too high	Add thinner to correct specific gravity
3. Solder purity exceeds contamination limits	Refresh and maintain the pot
4. Analysis period exceeded	Send solder sample for analysis
5. Oil contaminated (on waves needing oil)	Drain and replace oil, if used

B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
6. Warped boards	Do not use warped boards
7. Adhesives on SMD pads	See Section 4.1
8. Too much adhesive	See Section 4.1
9. Solder mask smear on pads	Improve incoming acceptance procedure
10. Pads not solderable	Don't use unsolderable boards
11. Component lead/terminations not solderable	Use only solderable parts
12. Components(s) too large for pad	Use correct parts as specified in Section 3.0
13. Components misaligned	See Section 6.0
14. Parts placed too close together	Correct placement machine problem See Section 6.0
15. Components skewed too close together	Correct placement machine problem See Section 6.0
16. Parts placed off the pads	Correct placement machine problem See Section 6.0
C. Design Related	
Potential Cause	*Corrective Action*
1. Pads/parts laid out too close	Lay parts/pads farther apart
2. Design recommendations not followed	Redesign where possible per specification See IPC-SM-782
3. Pads too small or pads too far under part	Extend pads out further
4. Larger part shadowing smaller one	Alter layout
5. Body of part too high	Specify lower profile part or redesign difficult parts onto top side for reflow
6. Solder mask too thick	Specify thinner solder mask
7. Solder mask too close around pad	Redimension more pad clearance

9.2 Observed Condition: Solder Bridges

A. Equipment Related	
Potential Cause	*Corrective Action*
1. Wave height/exit conditions incorrectly set	Set exit flow wave dynamics properly
2. Slow conveyor speed washes off too much flux	Increase speed so waves leave flux on
3. Conveyor too fast or insufficient preheat	Decrease speed—better solder drainage
4. Carrier bent or damaged	Send carrier/fixture to repair
5. Conveyor fingers bent	Replace bent fingers
6. Rails not level or parallel over nozzles	Reset rails level/parallel
7. Waves/Nozzles not level	Reset pot and nozzles level Use LevChek
8. Board not run best direction (orientation)	Turn SMA 90, 180 or 270 degrees
9. Dross recirculating from pot to wave onto SMA	Pull pump and nozzles—Clean out pot
10. Hot air knife after wave set incorrectly	Reset air knife
11. Insufficient flux applied	Increase flux application
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Flux—some low solids fluxes cause bridge	Try a different flux
2. Oil contaminated (on waves needing oil)	Drain and replace oil, if used
3. Flux thinned out too much	Specify gravity or titrate
4. Warped boards can cause poor exit flow	Do not use warped boards
5. Pads not solderable	Don't use unsolderable boards
6. Component leads/terminations not solderable	Use only solderable parts
7. Component(s) too large for pad	Use correct parts as specified
8. Components misaligned	Refer to Section 6.0
9. Parts placed too close together	Correct placement machine problem
10. Components skewed too close together	Correct placement machine problem
11. Parts placed off the pads	Correct placement machine problem
C. Design Related	
Potential Cause	*Corrective Action*
1. Pads/parts laid out too close	Lay parts/pads farther apart
2. Design alignment not followed	Redesign where possible per specification See IPC-SM-782
3. Bridge only across last two leads of SOIC	Add dummy pads for drainage
Potential Cause	*Corrective Action*
4. Bridge only on last two quad pack leads	Turn part 45 degrees & add dummy pads

9.3 Observed Condition: Unfilled Via Holes

A. Equipment Related	
Potential Cause	*Corrective Action*
1. Non-uniform fluxing across bottom of board	Check foam specific gravity bubbles, crest uniform
2. Conveyor too fast or insufficient preheat	Decrease speed for more heat in holes
3. Vibratory device in 2nd wave OFF	Turn ON vibratory wave
4. Set-point for vibratory device too low	Increase power
5. Carrier bent or damaged	Send carrier/fixture to repair
6. Conveyor fingers bent	Replace bent fingers
7. Rails not level or parallel over nozzles	Reset rails level/parallel
8. Waves/Nozzles not level	Reset pot and nozzles level Use LevChek
9. Insufficient flux applied	Increase flux application
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Flux specific gravity gone too high	Add thinner to correct specific gravity
2. Solder purity exceeds contamination limits	Refresh and maintain the pot
3. Analysis period exceeded	Send solder sample for analysis
4. Oil contaminated (on waves needing oil)	Drain and replace oil, if used
5. Warped boards—causes assembly to lift off wave	Do not use warped boards
6. Solder mask smeared on pads around holes	Verify incoming acceptance report
7. Holes not solderable or ‘‘Weak Knee’’ condition	Don’t use unsolderable boards
8. Poor plating (not continuous) in barrels	Use good boards that pass Float Test
9. Solder mask or other contamination in holes	Use good boards that pass Float Test
C. Design Related	
Potential Cause	*Corrective Action*
1. Solder mask too thick	Specify thinner solder mask
2. Solder mask too close around pad	Redimension more hole clearance

9.4 Observed Condition: Solder Wave Overflooding Board

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Board too large or assembly to heavy	Install stiffeners
2. Pallet, carrier or fingers adjusted too tight	Readjust for board expansion
3. Wave height set too high or conveyor too low	Adjust wave (lower) or conveyor (raise)
4. Conveyor speed too slow	Optimize conveyor speed
5. Preheat settings too high	Optimize preheat
6. Pallet not equipped with center support device	Redesign pallet
7. Center support device not adjusted correctly	Readjust
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Boards warped	Stiffen or fixture board
2. Heavy parts designed in center of board	Redesign heavy parts to edges
3. Multilayer ground/power planes not balanced	Redesign boards to balance not balanced or relieve ground/power planes
4. Holes or cutouts in board	Mask the holes
5. Board dimensions not optimized	Change length-width ratio Use stiffeners

9.5 Observed Condition: Grainy or Disturbed Joints

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Vibration or jerky action in conveyor	Clean and lubricate in conveyor chains
2. Removing assembly from conveyor too early	Allow assembly to travel to conveyor end to permit solidification of all joints
3. Excessive solder temperature/slow cooling rate	Reduce temperature, increase cooling
4. Too much liquid flux in holes	Provide more preheat to volatilize flux thinners
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Solder alloy contaminated	Analyze and correct solder composition
2. Water or contamination in fluxer air supply	Check and clean air filter on inlet air line
3. High lead in board solder coat	Check board solder coating
4. High gold content	Follow gold removal process per ANSI/J-STD-001
5. Moisture in board material causing outgassing	Bake boards prior to wave soldering
6. Chip terminations contaminated	Do not use contaminated parts
7. Contamination on some component leads	Do not use contaminated parts Pre-tin
8. Off-spec solder on bare board	Analyze and correct

9.6 Observed Condition: Cold Joints

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Pre-heat temperature too low	Check and adjust
2. Conveyor speed too high	Check and adjust
3. Low flux activity	Use more aggressive fluxes
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Contaminated or unsolderable leads/pads	Verify solderability per J-STD-002 or J-STD-003

9.7 Observed Condition: Solder Balls on Assembly Solder balls may be found on either the top-side or the bottom-side of the assembly. Solder balls on the assembly are more common when a low-solids flux or a preparations fluid is used.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Solder splash into pot rebounding onto board	Adjust anti-splash gates
2. Insufficient Activator	Increase conveyor speed Increase flux activator/solids content Increase quantity of flux applied Reduce air-knife pressure
3. Vibratory device in 2nd wave set too high	Reduce energy level to vibrator
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Flux solids content too low	Correct specific gravity Change flux type
2. Flux contaminated	Drain fluxer and refill with fresh flux
3. Solder mask not cured correctly	Check at incoming and correct
4. Solder mask not compatible with flux	Change solder mask or flux

9.8 Observed Condition: Cracked Chip Components or Plastic-Bodied Leaded Components

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Pre-heat profile too steep	Reduce ramp rate by adding more preheaters, reducing conveyor speed, or decreasing solder temperature
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Moisture entrapment	Pre-bake parts per IPC-SM-786
2. Inadequate multilayer chip construction	Refer to vendor or change vendor
3. Pick and place machine impacting parts	Review and adjust tweezer pressure and Z-axis speed or stop position
4. Warped boards	Use boards within flatness specification

10.0 CLEANING

Post-solder cleaning of assemblies is carried out to remove ionic and nonionic flux residues and other contaminants which may degrade product performance and reliability. It also may be required to facilitate bed-of-nails testing.

In surface mount technology problems with the cleaning operation are primarily associated with getting the cleaning solution in, and carrying the contaminants out of, the small spaces associated with the close packed, high density assemblies.

The tendency is to focus on flux residues, but in some instances contaminants from other manufacturing operations can be equally damaging to the reliability of the assembly.

Cleanliness assessment is also an important aspect of the cleaning process. Failure of the final cleanliness test is indicative of a cleaning process unable to remove residues. A second source of failure may be the test itself. For the former, the process/materials checks outlined above should provide the strategy needed to identify the problem. For the latter, a method of test calibration is required. (Any test should be calibrated at regular intervals to insure accuracy. For further information see IPC-CH-65).

This section presents a general listing of problems encountered in the cleaning process, potential causes and suggestions for corrective actions. These solutions may be related to equipment, materials, design or prior processes. Accordingly some of the solutions cannot be immediately implemented on the shop floor.

10.1 Observed Condition: Visible Residue Visible residue may have its source from the components or boards, the soldering process, or the cleaning process. The residue may have been on the assembly before soldering (dirty boards or components), it may have been added during soldering (flux or dross), or it may be the result of cleaning (inefficient removal or even added during cleaning).

10.1.1 Observed Condition: Residue at Incoming Incoming residue may be associated with either the manufacture of the boards and components, or their handling and storage.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Residue on components or boards	Pre-clean components or boards
2. Residue on components or boards	Work with supplier to eliminate residue
3. Residue found after storage	Pre-clean or modify storage conditions

10.1.2 Observed Condition: Residue after Soldering The soldering process may add residues from either the fluxing or soldering process. The presence of these residues may not be a problem if the cleaning process can adequately remove them. If it cannot (see below), it may be necessary to adjust the solder process or change the flux used to insure a cleanable assembly.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Cleaner not operating properly, e.g., no spray	Test cleaner, repair
2. Flux exposed to excess temperatures	Change solder temperature/ profile
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Flux and cleaning not properly matched	Match flux to cleaning process (see J-STD-004)
2. Cleaning media saturated with flux	Replace spent cleaning material

10.1.2.1 Observed Condition: White Residue White residue is a sub-class of visible residue. It is mentioned here because it often receives attention in excess of its due.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Rosin flux exposed to excess temperatures and soldering times	Reduce times and temperatures
2. Inefficient cleaning process	Improve cleaning process, e.g., longer cleaning time
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Inefficient cleaning of flux activators	Match cleaning media and flux or change flux
2. Plating salts on the printed wiring board	Pre-clean boards Work with vendor
3. Solder mask fillers leaching out in cleaning	Change mask or boards Improve mask cure
4. Tin or lead salts on board from reaction of flux with solder	Change flux or match flux with cleaning

10.2 Observed Condition: Failure of Conformal Coating Failure of the conformal coating to adhere may be due to either a coating process problem (not addressed here) or a cleaning issue.

A. Equipment/Process Related	
Potential Cause	*Corrective Action*
1. Cleaning process inadequate	Change/improve cleaning (see above)
2. Coating process not in control	Check process control (see IPC-SM-840)
B. Material/Prior-Process Related	
Potential Cause	*Corrective Action*
1. Residues on board not matched with cleaning process	Change cleaning process

11.0 REPAIR/REWORK

11.1 Scope The discussion of repair and rework of Surface Mount Technology (SMT) assemblies herein will be restricted to removal and replacement of misaligned, poorly soldered, damaged or faulty components and the important intermediary step of SMT land preparation. The discussion is also applicable to manual or semi-automated new build of SMT assemblies. For a discussion of repair of circuit elements (i.e., damaged or removed lands, traces, vias, etc.), substrates, and correction of soldering defects such as bridging or icicling, see relevant sections of this document and IPC-R-700C or subsequent revisions thereto.

The discussion will be further restricted to manual conductive and convective heating methods and semi-automated workstations. For a discussion of IR, automated hot bar, laser and other repair/rework methods, see relevant sections of this document or other IPC documents.

11.2 General Each rework/repair method has certain advantages and disadvantages depending on the particular Surface Mount Device (SMD) (lead/termination design, size, body material, etc.), component mounting site (adjacent components, access, substrate type, thermal mass, etc.), and the skill of the operator. For this reason, one should endeavor to choose, in the first instance, a method which is likely to be the most appropriate based on the goals and guidelines in 11.3.

11.3 Goals and Guidelines In the three basic processes of SMD removal, SMT land preparation and SMD installation/replacement, the fundamental goals include the following:

A. Non-destructive Process. No damage or degradation to the board (both substrate and circuit elements), adjacent components, and the component to be installed or removed. This damage may be either mechanical, thermo/mechanical or purely thermal in nature and may result in either immediate failure, latent degradation in performance and lifetime, or a combination of both.

B. Controllable, Reliable and Repeatable Process. The process can be employed and, when necessary, modified by a trained operator in a repetitive fashion with consistently acceptable results.

C. Process Appropriate to Particular Application. The process (or modification thereof) employed is appropriate to the particular application based on the relevant guidelines described below.

D. Operator Friendly Process. An operator of average skill can become acceptably proficient in employing and appropriately modifying the process with a reasonable amount of training and practice.

In addition to these common goals, each of the three basic processes has its own particular goals and guidelines due to differences in thermal considerations and other process requirements.

11.3.1 Non-destructive SMD Removal The particular process goals and guidelines for non-destructive removal of SMDs include the following:

- Preheat assembly and/or component as required
- Evenly heat all joints in a rapid, controllable fashion to achieve complete, simultaneous solder reflow
- Avoid thermal and/or mechanical damage to component, board, adjacent components and their joints
- Immediately remove component from board before molten solder resolidifies

11.3.2 SMT Land Preparation SMT land preparation should be performed prior to the installation (new build) or replacement (rework) of a new component. Avoidance of thermal and/or mechanical damage to the land and substrate is critical. The two primary steps include:

- Remove Old Solder

  This has been traditionally performed with solder wicking material and a soldering iron. Newer techniques include continuous vacuum desoldering with a solder extractor and continuous ‘‘flow’’ desoldering employing a solder extractor and special tip which allows reflow and vacuum aspiration of the old solder to occur simultaneously.

- Add New Solder

  This is accomplished by either prefilling (pretinning) lands by reflowing wire solder with a soldering iron (or some other heating method), or by applying solder paste with a dispenser, typically, but not always, prior to component placement.

11.3.3 SMD Installation The particular process goals and guidelines for safe SMD installation include the following:

- Align and place component to lands (tack if necessary)
- Preheat assembly and/or component, as required
- Pre-dry applied solder paste with gentle heating
- Reflow Solder Joints (individually, in groups or all together) with concentrated ‘‘targeted’’ heat in a rapid, controlled manner while maintaining alignment of component to lands
- Avoid thermal and/or mechanical damage to component, board, adjacent components and their joints
- Clean and inspect

11.4 Primary Heating Methods Primary heating methods are those principally responsible for achieving solder melt during an SMD installation or removal process. These are to be distinguished from methods used for pre-heating and auxiliary heating which are employed in addition to primary heating methods in particular situations, as described in 11.5 Preheating and Auxiliary Heating.

11.4.1 Conductive (by contact) Heating Methods

11.4.1.1 Continuously Heated Devices Handheld conductive heating devices generally fall into one of two categories. The first is continuously heated devices such as soldering irons, thermal tweezers and thermal pick devices which may be held at a selected idle tip temperature prior to use.

Continuously heated devices generally employ tinnable tips to optimize heat transfer to the work.

Continuously heated conductive heating devices in SMT rework and repair situations offer the following potential advantages:

- Can effectively transfer a large amount of heat rapidly
- Can control heat delivery with tip temperature and dwell time
- Can safely access hard-to-reach places and confine heat to limited areas with proper tip design, selection and use

With continuously heated conductive heating devices, the following guidelines and precautions should be observed:

- Must establish good thermal linkage with joints for effective heat transfer

- Contact may disturb component lead-to-land alignment
- May transfer heat too rapidly for use with solder paste or sensitive components
- May obstruct view during alignment and reflow and interfere with joint formation during reflow.
- May disturb lead/land alignment

11.4.1.2 Pulse Heated Devices The second category is pulse heated devices such as lapflo type tools, resistance tweezers and other handheld devices which produce heat directly in the tip or the work with current. These devices generally employ low mass, non-tinnable tips which can remain in contact with solder joints as they cool thereby facilitating proper component alignment.

Pulse heated conductive heating devices in SMT rework and repair situations offer the following potential advantages:

- Can effectively transfer a large amount of heat rapidly
- Can control heat delivery with dwell time, power setting and tip selection
- Slim design tips can safely access tight places and confine heat to limited areas.
- Low mass tips heat up and cool down rapidly
- Non-tinnable tips can contact joint cold, heat to reflow, and remain in contact during solder resolidification to stabilize component alignment
- More gradual heat-up works well with solder paste
- Offers greater (though typically insufficient by itself) margin of safety for thermal shock sensitive components

With pulse heated conductive heating devices, the following guidelines and precautions should be observed:

- Less effective means to control heat delivery since handheld devices are generally not temperature controlled
- Must establish good thermal linkage with joints for effective heat transfer
- Improper contact may disturb component lead-to-land alignment
- May produce unacceptable residual stress in some stiff leads if not coplanar with lands

11.4.2 Convective (by gas/air flow) Heating Methods Convective heating methods are generally found in devices such as semi-automated benchtop workstations, high powered handheld hot air guns and nozzle-focused hot air jet handpieces.

Convective heating devices in SMT rework and repair situations offer the following potential advantages:

- Non-contact process—generally won't disturb joints or obstruct view
- Under most conditions, work well with solder paste
- Can control heat delivery with:

—Gas/air temperature
—Gas/air flow rate
—Dwell time
—Distance from work

With convective heating devices, the following guidelines and precautions should be observed:

- Significant possibility of damaging errant heating - must properly focus and control gas/air flow to avoid this
- Must adequately control exit gas/air velocity (via pressure or flow rate) to avoid:

—Displacement of applied solder paste
—Disturbing the alignment of components AND
—Minimize errant heating

- Inefficient means of primary heat delivery when compared to conductive methods

11.5 Preheating and Auxiliary Heating There are two principal reasons for pre-/auxiliary heating during SMT rework and repair. Preheating is required when there is present a risk of thermal shock in the substrate, components or both. The goal here is to ''thermally soak'' the component and/or assembly to a given temperature and thereby eliminate dangerous temperature gradients which could produce immediate damage, reduction of reliability or degradation over time.

For avoidance of thermal shock, the rate of temperature rise of the work during preheating is critical. For example, many ceramic chip capacitor manufacturers recommend that pre-heating occur at a rate no greater than 2–4°C/sec. until a given minimum temperature is reached.

Pre-/auxiliary heating is also required when the primary heating method cannot bring all of the solder joints completely up to proper reflow temperature at all or in an acceptably rapid period of time due to heat sinking by nearby portions of the substrate, circuit elements and adjacent components. The goal here is to bring the assembly (or a portion thereof) up to a sufficient (yet safe) temperature whereby the rate of heat sinking is reduced to a level at which the primary heating device can affect proper solder reflow.

Pre-heating for SMT repair/rework is typically accomplished by either a temperature controlled conductive heating plate, by controlled heated gas/air flow and to a lesser extent IR heat. Again, controlling the rate of heat-up of the work during the pre-heating phase (as measured by rate of temperature rise) is critical to avoid the risk of thermal shock.

11.6 Vision Systems and Component Placement As high lead count, fine pitch SMDs become commonplace, the task of properly aligning and placing these devices during manual or semi-automated SMT rework becomes more challenging.

Appropriate vision systems with sufficient magnification, resolution, field of view and working distance are critical for viewing alignment of component leads to lands and monitoring joint reflow during SMD installation.

For installation of large body, fine pitch devices, it is often difficult for traditional vision systems to provide the level of magnification necessary to align fine pitch leads to lands with a field of view showing the entire component. New split image systems address this problem by providing sufficiently magnified views of a large component's opposite corners such that accurate lead/land alignment can be achieved.

Proper component handling systems which can adequately establish and maintain X, Y, Z and THETA positioning are also essential for successful alignment and placement during SMD installation.

Vision systems come in various forms including stereo microscopes, trinocular microscopes and CCTV (video) systems. While microscopes are generally more economical, CCTV systems offer greater ease of use and less operator fatigue. See IPC-OI-645 for further information on optical inspection equipment.

11.7 Selecting Optimum Method of Rework/Repair Other than reasons such as purchase and life cycle costs of equipment, operator training and skill costs and other economic considerations, selecting the optimum method for SMT rework/repair depends on a variety of factors. These include:

- Type of component
- Size of component
- Type of substrate
- Component mounting site
  - —thermal mass considerations
  - —adjacent components
  - —safe access to site
- Whether the component is being installed or removed
- Whether the component being removed is being salvaged
- Applicable workmanship specifications
- EOS/ESD control requirements

Every rework/repair method and its attendant equipment has advantages and disadvantages in each particular SMD installation or removal situation. These can be evaluated in terms of their ability to achieve the process goals and guidelines contained herein when used by a properly trained operator.

11.8 Troubleshooting After making an initial selection of the most appropriate SMD installation and removal method based on the guidelines contained herein, each process must be modified by the operator based on observed conditions to achieve optimum results.

11.9 Observed Condition: Insufficient solder reflow to affect SMD removal

Potential Cause	*Corrective Action*
1. High thermal mass condition	Use pre-auxiliary heating
2. Insufficient heat from primary heating device	Use more efficient heating method or device, or increase idle tip temperature
3. Insufficient heating time	Increase heating time

For Conductive Heating Methods:	
Potential Cause	*Corrective Action*
1. Insufficient thermal linkage between heating tip and work	Improve thermal linkage with solder and/or select tip which provides more contact area
For Convective Heating Methods:	
Potential Cause	*Corrective Action*
1. Insufficient air temperature	Use more efficient heating method or increase air temperature
2. Insufficient air flow	Increase air flow
3. Air nozzle too far from work	Bring air nozzle closer to work

11.10 Observed Condition: Component not removable after solder reflow or lifted lands upon attempted removal

Potential Cause	*Corrective Action*
1. Solder resolidifying before removal can be affected	Use method in which removal can be affected immediately upon complete reflow of all joints
2. Adhesive holding component	Shear bond after heating to free component (i.e. theta movement)
3. Not all joints have been reflowed	Use alternate method which provides rapid simultaneous reflow of all joints followed by immediate removal to keep process time to a minimum. Assure that heated tip or air properly contacts and reflows all joints equally.

11.11 Observed Condition: Lands being lifted or damaged during removal of old solder

Potential Cause	*Corrective Action*
1. Lands being overheated	Use lower temperature or alternate method which doesn't overheat lands
2. Lands subject to mechanical force while being heated	Use alternate method or minimize mechanical force to which lands are subjected

11.12 Observed Condition: Insufficient or poor adhesion of solder on lands during pretinning operation

Potential Cause	*Corrective Action*
1. Lands oxidized or contaminated	Clean lands using appropriate method
2. Idle tip temperature of soldering iron too high	Lower idle tip temperature
3. Improper size or shape of soldering tip	Use appropriate size and shape tip

11.13 Observed Condition: Misalignment of component to land pattern during SMD installation

Potential Cause	*Corrective Action*
1. Some leads are not tacked or the component is not being held in place (conductive heating)	Tack some leads or hold component in place during reflow, or use convective method
2. Air velocity too high (convective heating)	Lower air flow rate

11.14 Observed Condition: Dispersion or solderballing of solder paste during reflow

Potential Cause	*Corrective Action*
1. Air velocity too high (convective heating)	Lower air flow rate
2. Air temperature too hot	Lower air temperature
3. Soldering tip too hot (conductive heating)	Use convective method or lower tip temperature
4. No predrying of solder paste	Predry solder using convective method
5. Old or defective solder paste	Use fresh, accepted solder

11.15 Observed Condition: Blistering or measling of substrate, or lifted lands during SMD installation or removal

For Conductive Heating Methods:	
Potential Cause	*Corrective Action*
1. Preheat temperature too high	Lower preheat temperature
2. Tip temperature too high	Reduce tip temperature
3. Too much pressure applied	Reduce pressure on tip and improve thermal linkage with solder joint(s)
4. Poor contact with solder joints	Assure good thermal contact with solder joint(s)
For Convective Heating Methods:	
Potential Cause	*Corrective Action*
1. Preheat temperature too high	Lower preheat temperature
2. Air temperature too high	Reduce air temperature
3. Air volume or velocity too high	Reduce air volume or velocity or use more focused convective heating method

ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

ANSI/IPC-T-50 Terms and Definitions for Interconnecting and Packaging Electronic Circuits Definition Submission/Approval Sheet

The purpose of this form is to keep current with terms routinely used in the industry and their definitions. Individuals or companies are invited to comment. Please complete this form and return to:

IPC
2215 Sanders Road
Northbrook, IL 60062-6135
Fax: 847 509.9798

SUBMITTOR INFORMATION:

Name: ______

Company: ______

City: ______

State/Zip: ______

Telephone: ______

Date: ______

❏ This is a **NEW** term and definition being submitted.
❏ This is an **ADDITION** to an existing term and definition(s).
❏ This is a **CHANGE** to an existing definition.

Term	Definition

If space not adequate, use reverse side or attach additional sheet(s).

Artwork: ❏ Not Applicable ❏ Required ❏ To be supplied
❏ Included: Electronic File Name: ______

Document(s) to which this term applies: ______

Committees affected by this term: ______

Office Use

IPC Office

Date Received: ______
Comments Collated: ______
Returned for Action: ______
Revision Inclusion: ______

Committee 2-30

Date of Initial Review: ______
Comment Resolution: ______
Committee Action: ❏ Accepted ❏ Rejected
❏ Accept Modify

IEC Classification

Classification Code • Serial Number

Terms and Definition Committee Final Approval Authorization:

Committee 2-30 has approved the above term for release in the next revision.

Name: ______ Committee: IPC 2-30 Date: ______

Technical Questions

The IPC staff will research your technical question and attempt to find an appropriate specification interpretation or technical response. Please send your technical query to the technical department via:

tel 847/509-9700 fax 847/509-9798
www.ipc.org e-mail: answers@ipc.org

IPC World Wide Web Page www.ipc.org

Our home page provides access to information about upcoming events, publications and videos, membership, and industry activities and services. Visit soon and often.

IPC Technical Forums

IPC technical forums are opportunities to network on the Internet. It's the best way to get the help you need today! Over 2,500 people are already taking advantage of the excellent peer networking available through e-mail forums provided by IPC. Members use them to get timely, relevant answers to their technical questions.

TechNet@ipc.org

TechNet forum is for discussion of technical help, comments or questions on IPC specifications, or other technical inquiries. IPC also uses TechNet to announce meetings, important technical issues, surveys, etc.

ChipNet@ipc.org

ChipNet forum is for discussion of flip chip and related chip scale semiconductor packaging technologies. It is cosponsored by the National Electronics Manufacturing Initiative (NEMI).

ComplianceNet@ipc.org

ComplianceNet forum covers environmental, safety and related regulations or issues.

DesignerCouncil@ipc.org

Designers Council forum covers information on upcoming IPC Designers Council activities as well as information, comment, and feedback on current design issues, local chapter meetings, new chapters forming, and other design topics.

Roadmap@ipc.org

The IPC Roadmap forum is the communication vehicle used by members of the Technical Working Groups (TWGs) who develop the IPC National Technology Roadmap for Electronic Interconnections.

LeadFree@ipc.org

This forum acts as a peer interaction resource for staying on top of lead elimination activities worldwide and within IPC.

ADMINISTERING YOUR SUBSCRIPTION STATUS:

All commands (such as subscribe and signoff) must be sent to listserv@ipc.org. Please DO NOT send any command to the mail list address, (i.e.<mail list> @ipc.org), as it would be distributed to all the subscribers.

Example for subscribing:
To: LISTSERV@IPC.ORG
Subject:
Message: subscribe TechNet Joseph H. Smith

Example for signing off:
To: LISTSERV@IPC.ORG
Subject:
Message: sign off DesignerCouncil

Please note you must send messages to the mail list address ONLY from the e-mail address to which you want to apply changes. In other words, if you want to sign off the mail list, you must send the signoff command from the address that you want removed from the mail list. Many participants find it helpful to signoff a list when travelling or on vacation and to resubscribe when back in the office.

How to post to a forum:

To send a message to all the people currently subscribed to the list, just send to <mail list>@ipc.org. Please note, use the mail list address that you want to reach in place of the <mail list> string in the above instructions.

Example:
To: TechNet@IPC.ORG
Subject: <your subject>
Message: <your message>

The associated e-mail message text will be distributed to everyone on the list, including the sender. Further information on how to access previous messages sent to the forums will be provided upon subscribing.

For more information, contact Hugo Scaramuzza
tel 847/790-5312 fax 847/509-9798
e-mail: scarhu@ipc.org www.ipc.org/html/forum.htm

Education and Training

IPC conducts local educational workshops and national conferences to help you better understand emerging technologies. National conferences have covered Ball Grid Array and Flip Chip/Chip Scale Packaging. Some workshop topics include:

Printed Wiring Board Fundamentals
Troubleshooting the PWB Manufacturing Process
Choosing the Right Base Material Laminate
Acceptability of Printed Boards Designers Certification Preparation
New Design Standards
High Speed Design
Design for Manufacturability
Design for Assembly

IPC-A-610 Training and Certification Program

"The Acceptability of Electronic Assemblies" (ANSI/IPC-A-610) is the most widely used specification for the PWB assembly industry. An industry consensus Training and Certification program based on the IPC-A-610 is available to your company.

For more information on programs, contact John Riley
tel 847/790-5308 fax 847/509-9798
e-mail: rilejo@ipc.org www.ipc.org

IPC Video Tapes and CD-ROMs

IPC video tapes and CD-ROMs can increase your industry know-how and on the job effectiveness.

For more information on IPC Video/CD Training, contact Mark Pritchard
tel 505/758-7937 ext. 202 fax 505/758-7938
e-mail: markp@ipcvideo.com www.ipc.org
www.ipc.org

IPC Printed Circuits Expo℠

IPC Printed Circuits Expo is the largest trade exhibition in North America devoted to the PWB industry. Over 90 technical presentations make up this superior technical conference.

April 4-6, 2000 **San Diego, California**	**April 3-5, 2001** **Anaheim, California**	**March 26-28, 2002** **Long Beach, California**

Exhibitor information:
Contact: Jeff Naccarato
tel 630/434-7779

Registration information:
tel 847/790-5361
fax 847/509-9798
e-mail: registration@ipc.org
www.ipcprintedcircuitexpo.org

APEX℠ / IPC SMEMA Council
Electronics Assembly Process Exhibition & Conference

APEX is the premier technical conference and exhibition dedicated entirely to the PWB assembly industry.

March 14-16, 2000 **Long Beach, California**	**January 16-18, 2001** **San Diego, California**	**Spring 2002** **TBA**

Exhibitor information:
Contact: Mary MacKinnon
tel 847/790-5386

Registration information:
APEX Hotline: tel 877/472-4724
fax 847/790-5361
e-mail: apex2000@ipc.org
www.apex2000.org

How to Get Involved

The first step is to join IPC. An application for membership can be found in the back of this publication. Once you become a member, the opportunities to enhance your competitiveness are vast. Join a technical committee and learn from our industry's best while you help develop the standards for our industry. Participate in market research programs which forecast the future of our industry. Participate in Capitol Hill Day and lobby your Congressmen and Senators for better industry support. Pick from a wide variety of educational opportunities: workshops, tutorials, and conferences. More up-to-date details on IPC opportunities can be found on our web page: www.ipc.org.

For information on how to get involved, contact:
Jeanette Ferdman, Membership Manager
tel 847/790-5309 fax 847/509-9798
e-mail: JeanetteFerdman@ipc.org www.ipc.org

ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

Application for Site Membership

PLEASE CHECK APPROPRIATE CATEGORY

Thank you for your decision to join IPC. IPC Membership is **site specific,** which means that IPC member benefits are available to all individuals employed at the site designated on the other side of this application.

To help IPC serve your member site in the most efficient manner possible, please tell us what your facility does by choosing the most appropriate member category.

■ **INDEPENDENT PRINTED BOARD MANUFACTURERS**

Our facility manufactures and sells to other companies, printed wiring boards or other electronic interconnection products on the merchant market.

WHAT PRODUCTS DO YOU MAKE FOR SALE?

- ☐ One-sided and two-sided rigid printed boards
- ☐ Multilayer printed boards
- ☐ Flexible printed boards
- ☐ Flat cable
- ☐ Hybrid circuits
- ☐ Discrete wiring devices
- ☐ Other interconnections

Name of Chief Executive Officer/President____

■ **INDEPENDENT PRINTED BOARD ASSEMBLERS EMSI COMPANIES**

Our facility assembles printed wiring boards on a contract basis and/or offers other electronic interconnection products for sale.

- ☐ Turnkey
- ☐ SMT
- ☐ Chip Scale Technology
- ☐ Through-hole
- ☐ Mixed Technology
- ☐ Consignment
- ☐ BGA

Name of Chief Executive Officer/President____

■ **OEM – MANUFACTURERS OF ANY END PRODUCT USING PCB/PCAS OR CAPTIVE MANUFACTURERS OF PCBS/PCAS**

Our facility purchases, uses and/or manufactures printed wiring boards or other electronic interconnection products for our own use in a final product. Also known as original equipment manufacturers (OEM).

IS YOUR INTEREST IN:

- ☐ purchasing/manufacture of printed circuit boards
- ☐ purchasing/manufacturing printed circuit assemblies

What is your company's main product line?

____

■ **INDUSTRY SUPPLIERS**

Our facility supplies raw materials, machinery, equipment or services used in the manufacture or assembly of electronic interconnection products.

What products do you supply?

____

■ **GOVERNMENT AGENCIES/ ACADEMIC TECHNICAL LIAISONS**

We are representatives of a government agency, university, college, technical institute who are directly concerned with design, research, and utilization of electronic interconnection devices. (Must be a non-profit or not-for-profit organization.)

Please be sure to complete both pages of application.

Application for Site Membership

IPC
ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

Company Name

Street Address

City | State | Zip | Country

Main Phone No. | Fax

Primary Contact Name

Title | Mail Stop

Phone | Fax | e-mail

Senior Management Contact

Title | Mail Stop

Phone | Fax | e-mail

Please check one:

❑ $1,000.00 Annual dues for Primary Site Membership (Twelve months of IPC membership begins from the time the application and payment are received)

❑ $800.00 Annual dues for Additional Facility Membership: Additional membership for a site within an organization where another site is considered to be the primary IPC member.

❑ $600.00** Annual dues for an independent PCB/PWA fabricator or independent EMSI provider with annual sales of less than $1,000,000.00. **Please provide proof of annual sales.

❑ $250.00 Annual dues for Government Agency/University/not-for-profit organization

TMRC Membership ❑ Please send me information on Membership in the Technology Marketing Research Council (TMRC)

AMRC Membership ❑ Please send me information for Membership in the Assembly Marketing Research Council (AMRC)

Payment Information

Enclosed is our check for $___________________

Please bill my credit card: (circle one) MC AMEX VISA DINERS

Card No. ____Exp date _____________

Authorized Signature ____

Mail application with check or money order to:

IPC
Dept. 851-0117W
P.O. Box 94020
Palatine, IL 60094-4020

Fax/Mail application with credit card payment to:

IPC
2215 Sanders Road
Northbrook, IL 60062-6135
Tel: 847 509.9700
Fax: 847 509.9798

PLEASE ATTACH BUSINESS CARD OF OFFICIAL REPRESENTATIVE HERE

ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

Standard Improvement Form

IPC-S-816

The purpose of this form is to provide the Technical Committee of IPC with input from the industry regarding usage of the subject standard.

Individuals or companies are invited to submit comments to IPC. All comments will be collected and dispersed to the appropriate committee(s).

If you can provide input, please complete this form and return to:

IPC
2215 Sanders Road
Northbrook, IL 60062-6135
Fax 847 509.9798

1. I recommend changes to the following:

___ Requirement, paragraph number __________

___ Test Method number __________, paragraph number __________

The referenced paragraph number has proven to be:

___ Unclear ___ Too Rigid ___ In Error

___ Other ____________________

2. Recommendations for correction:

____

____

____

____

3. Other suggestions for document improvement:

____

____

____

____

Submitted by:

Name ____________________ Telephone ____________________

Company ____________________ E-mail ____________________

Address ____________________

City/State/Zip ____________________ Date ____________________

ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES

ISBN #1-58098

2215 Sanders Road, Northbrook, IL 60062-6135
Tel. 847.509.9700 Fax 847.509.9798
www.ipc.org