Mastering Selective Soldering Processing and Manufacturability

About ITM Consulting
- Process audits
- Process troubleshooting
- Process dispute mediation

About ITM Marketing
- Benchmark reports
- Market studies
- Technical marketing

Selective Soldering Workshops
- Introductory and advanced 2-day selective soldering workshops
- Compresses learning curve for those new to selective or advances in-depth knowledge
- More than 195 people from over 100 companies have attended previous workshops

Selective Soldering Workshops
- Taught by independent industry specialists
- Combined classroom and hands-on curriculum
- ACE, ITM and SMTA certified training program

Topics Covered
- Fundamentals of Through-Hole Soldering
- Solderability Issues
- Solder Alloys
- Flux Deposition and Flux Activation
- No-Clean Processing
- Through-Hole Design Guidelines
- Quality Measurement
- Process Troubleshooting Guidelines
- Process Optimization Methods
Fundamentals of TH Soldering

- Fundamentals of Through-Hole Soldering
- Solderability Issues
- Solder Alloys
- Flux Deposition and Flux Activation
- No-Clean Processing
- Through-Hole Design Guidelines
- Quality Measurement
- Process Troubleshooting Guidelines
- Process Optimization Methods

Capillary Action

- Capillary action is defined as:
  - Phenomena whereby liquidous solder rises in a column formed by a through-hole (TH) component lead and the inside walls of a plated through-hole (PTH)
- Capillary action is directly dependent upon:
  - Surface tension and wetting action of the liquidous solder as well as wetting action of plated surfaces

Through-Hole Vertical Fill

- Acceptable TH solder joints must:
  - Provide evidence of good wetting on both component lead and PTH surfaces
  - And must:
    - Meet various requirements of positive fillet on both solder source side and solder destination side

Solder Joint Formation

- Wetting is essential prerequisite:
  - Wetting means a specific interaction has taken place between liquidous solder and solderable surfaces
- Wetting is only possible if:
  - Liquidous solder comes in immediate contact with metallic surfaces of TH component leads, PTH surfaces and printed circuit board pads
- Surface tension:
  - Extent to which liquidous solder will spread across a surface, or flow into a gap, is dependent upon its surface tension

Vertical Fill Force Model

- Ability of liquidous solder to fill a PTH and form a positive topside fillet on destination side can be depicted as a vertical fill force model, whereby:
  - A positive topside fillet will be formed when the product of the wetting force \( F_w \) times the capillary force \( F_c \) is greater than the force of gravity \( F_g \) plus the peel-off force \( F_p \) of the mechanical de-bridging action
Component Thermal Mass Differential

- It is essential that:
  - All TH component leads, PTH and PCB pad surfaces reach wetting temperature within a short period of time and in a uniform manner
- Different thermal mass of:
  - Various TH components, and/or heat sinking effect of interconnections to a ground plane, will result in thermal mass differentials
- Temperature sensitive components:
  - Can be damaged if their internal threshold temperature boundaries are exceeded

Component Thermal Mass Differential

- Effects of thermal mass differential

TH Soldering Process Parameters

- Selective soldering involves:
  - Application of heat and solder simultaneously to TH components and in localized areas
- Thermal considerations:
  - Application of heat must be within safe time-temperature boundaries for TH components
- Time-temperature boundaries:
  - Are dependent upon thermal requirements of a particular PCBA and solder alloy used

TH Soldering Process Parameters

- Tin-lead (SnPb) alloy:
  - Typical 250-270°C solder bath temperature and 2-4 sec dwell time
- Lead-free (SAC305) alloy:
  - Typical 290-320°C solder bath temperature and dwell time of 0.5-1.5 sec longer
- Regardless of solder alloy used:
  - Basic principles of liquidous flow soldering must take place

Flow diagram of typical liquidous soldering process
Thermal Processing

- Preheat temperature:
  - Is assumed by PCBA topside at the end of preheating period
  - Is generally specified for the proper thermal conditioning of a specific flux type
- Dwell time, or solder contact time:
  - Together with soldering temperature determine total heat flow to soldering area

Thermal Processing

- Process parameters:
  - Are subject to complex interaction of several factors determined by a particular PCBA and selective soldering machine
  - Short wetting times are required to avoid TH component damage
- Flux is used to:
  - Raise energy level and promote wetting of solderable surfaces
  - Flux must be activated at the proper time and temperature and dried to the correct viscosity

Thermal Processing

- Dwell time:
  - Is important criterion effecting heat transfer
  - Is affected by shape and contact length of solder nozzle
- Board-to-nozzle interaction has 3 distinct and interacting factors:
  - Dwell time, immersion depth and contact length
  - Direct interrelationship can be expressed as: Dwell time = contact length / traverse speed

Intermetallic Layer Formation

- To form quality TH solder joints the selective soldering process must:
  - Raise temperature of base metal surfaces and allow sufficient wetting
  - Provide adequate contact for capillary action to occur
  - Provide adequate thermal energy to form intermetallic layer
- Resulting in CuSn intermetallic between:
  - Solder alloy and copper of TH leads
  - Solder alloy and copper in PTH
  - Ideal intermetallic thickness is approximately 1.0-1.5 µm

Solderability Issues

- Fundamentals of Through-Hole Soldering
- Solderability Issues
- Solder Alloys
- Flux Deposition and Flux Activation
- No-Clean Processing
- Through-Hole Design Guidelines
- Quality Measurement
- Process Troubleshooting Guidelines
- Process Optimization Methods

Oxidation Layers

- Minimum solderability standards:
  - Must be met by PCB and component vendors
  - And maintained during in-house handling
- PCB and component storage:
  - PCBs should be kept in sealed packages to minimize contact with air and moisture
  - PCBs and TH components should be consumed on a first-in, first-out (FIFO) basis to minimize effects of oxidation
Surface Wetting

- Flow soldering is defined as:
  - Formation of metallurgical bond between two metal surfaces using a low melting point molten solder alloy that is quenched and solidified
  - Melting point of solder alloy must be below melting point of solderable surfaces
  - Solderability, or the ability of the surfaces to be wetted, must take place in order for an intermetallic compound to be formed creating a metallurgical bond

Zero Force Wetting Time

- TH flow soldering process parameters are:
  - Subject to complex interaction of several factors determined by a particular PCBA and selective soldering machine
  - Short wetting times are required to minimize thermal damage to TH components
  - Some lead-free solder alloys:
    - Exhibit elongated wetting times based on standard laboratory testing methods
    - Traverse speeds should be decreased to increase both dwell time and contact time

Zero Force Wetting Time

<table>
<thead>
<tr>
<th>Solder Temperature</th>
<th>Sn/Pb</th>
<th>SnAgCu</th>
<th>CuSnAg-Pb</th>
<th>SnAg-Pb</th>
<th>SnBi-Pb</th>
</tr>
</thead>
<tbody>
<tr>
<td>240°C</td>
<td>0.13 sec</td>
<td>0.17 sec</td>
<td>1.67 sec</td>
<td>2.05 sec</td>
<td>6.40 sec</td>
</tr>
<tr>
<td>260°C</td>
<td>0.19 sec</td>
<td>0.27 sec</td>
<td>1.87 sec</td>
<td>2.30 sec</td>
<td>6.70 sec</td>
</tr>
<tr>
<td>280°C</td>
<td>0.20 sec</td>
<td>0.29 sec</td>
<td>1.90 sec</td>
<td>2.35 sec</td>
<td>6.75 sec</td>
</tr>
</tbody>
</table>

Bare Board Cleanliness

- PCBs received from board fabricator should be periodically tested for:
  - Ionic and other contamitants that can adversely effect solderability
  - As a general rule bare board cleanliness should not exceed 1.0µg/in² (0.15µg/cm²) of NaCl before soldering
  - At a bare minimum one-time testing should be conducted to establish a baseline

Post-Soldering Ionic Levels

- As a general guideline post-soldering cleanliness of PCBAs should not exceed:
  - 3.0-7.0µg/in² (0.45-1.1µg/cm²) of NaCl as well as chlorides and bromides after all soldering processing
  - This is especially important when using no-clean flux chemistries

Dross Abatement Strategies

- Several dross abatement practices should be followed:
  - Periodic cleaning of solder pots
  - Routine dross removal
  - Minimize running time and wave height
  - These practices will mitigate dross entrapment as well as minimize accumulation of solder contaminates
Dross Abatement Strategies

<table>
<thead>
<tr>
<th>Practice</th>
<th>Method and Effect</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ruling Time</td>
<td>Reduce solder pot casting time to minimize dross buildup</td>
</tr>
<tr>
<td>Wave Height</td>
<td>Reducing wave height to a minimum reduces dross generation</td>
</tr>
<tr>
<td>Nitrogen Heating</td>
<td>Always maintain nitrogen surafe around solder nozzles and wave pot</td>
</tr>
<tr>
<td>Cold Shre Removal</td>
<td>Prevents shre or excess surf ace from removal of solder pot</td>
</tr>
<tr>
<td>Solder Cross Removal</td>
<td>Removes moisture, residues of solder parts and wave impurities</td>
</tr>
<tr>
<td>Dross Initiation</td>
<td>The use of a dross inhibitor additive is not recommended</td>
</tr>
</tbody>
</table>

Recommended dross abatement practices

Solder Alloys

- Fundamentals of Through-Hole Soldering
- Solderability Issues
- Solder Alloys
- Flux Deposition and Flux Activation
- No-Clean Processing
- Through-Hole Design Guidelines
- Quality Measurement
- Process Troubleshooting Guidelines
- Process Optimization Methods

Tin-Lead and Lead-Free Alloys

- Because most lead-free alloys have higher melting points:
  - Increased preheat temperatures are generally required
  - Solder pot temperature and solder contact time should also be increased
- At the same time solder pot temperature and solder contact time should be controlled to:
  - Avoid thermal shocking, or excessive thermal differential upon PCB and TH components when contacting the solder nozzle

Tin-Lead and Lead-Free Alloys

- Majority of lead-free alloys exhibit:
  - Decreased wetting characteristics and slower wetting times than SnPb solders
  - Flow characteristics are generally more viscous
- Because lead-free alloys have higher melting points generally 30-40ºC more than SnPb:
  - Places greater demand on flux performance
  - Higher activity flux should be considered
  - However volume and uniformity of flux deposition is essential

Surface Wetting Characteristics

- Wetting ability of commonly used lead-free solder alloys such as Sn96.5Ag3.0Cu0.5 (SAC305) to fill a PTH is greatly affected by PCB surface finish
- Examples of PCB surface finish include:
  - Nickel-gold (NiAu)
  - Immersion silver (Ag)
  - OSP over bare copper (Cu)

Surface Wetting Characteristics

Effects of various board finishes on through-hole wetting
Solder Alloy Characteristics

- Because of increased tin content:
  - Lead-free alloys oxide at a more rapid rate when in liquidous state than SnPb solder
  - Due to higher tin content tin oxide consisting of tin-oxygen (SnO) and SnO$_2$ forms at a faster rate
  - Higher processing temperatures results in more oxidation and dross buildup

Copper Dissolution

- Phenomenon of copper dissolution with lead-free TH soldering is a major quality concern:
  - Erosion of copper was present with SnPb solder but to a lesser degree as eutectic alloy functioned as an inhibitor
  - Lead-free alloys dissolve 2 to 4 times more copper than SnPb solder
  - This is due to sluggish wetting behavior and elongated time and temperature process window for lead-free alloys

Pin-to-Pin Variation

- Post-wave rework is often done with a conventional solder fountain
- These systems commonly use a solder flow well that agitates the solder from the center outward
- This results in a scrubbing action that aggravates copper erosion

Solder Nozzle and Solder Pot Correlation

- Correlation between solder joint temperature ($T_1$) and solder pot temperature ($T_2$) should be monitored
- Accurate measurement verifies:
  - Proper flux activation has been achieved
  - Bottom-side PCB $\Delta T$ is less than 100-140°C to minimize thermal shocking
  - Solder joint temperatures are within desired range of 245-250°C
  - Individual solder joints are within ± 1°C
Temperature correlation of solder nozzle, solder pot and solder joints

Flux Deposition and Flux Activation
- Fundamentals of Through-Hole Soldering
- Solderability Issues
- Solder Alloys
- Flux Deposition and Flux Activation
- No-Clean Processing
- Through-Hole Design Guidelines
- Quality Measurement
- Process Troubleshooting Guidelines
- Process Optimization Methods

Solder Nozzle and Solder Pot Correlation

Flux Deposition and Flux Activation

Liquid Flux Chemistries
- Purpose of liquid flux is to remove surface oxides and provide active base metals for solder wetting
  - Oxidation removal occurs via a chemical action that varies with each flux type
  - Fluxes are limited in their cleaning ability
  - Fluxes are not capable of removing heavy oxide layers, surface oils or organic coatings
  - A separate pre-cleaning operation should be considered if such contaminants are present

Thermal Aspects of Flux Activation
- Liquid flux must be present and active to:
  - Clean and protect solderable surfaces before contact with solder nozzle
  - Raise energy level and promote wetting of solderable surfaces
  - Proper thermal activation is essential to dry flux vehicle and activate flux solids
  - High melting point alloys require robust fluxes to survive higher preheat temperatures
  - Distinct advantage of selective versus wave soldering is flux residues cannot become entrapped under aperture pallets

Liquid Flux Chemistries

Common liquid flux types and general flux characteristics

Thermal Aspects of Flux Activation

Liquid Flux Chemistries

Common liquid flux types and general flux characteristics

Thermal Aspects of Flux Activation

Selectively soldering drop-jet application

Aperture wave pallet residues
Preheat Temperature

- Topside PCB temperature is generally specified for proper condition of specific flux type
- Preheating is required to ensure proper thermal activation of liquid flux chemistry
- Total PCBA heat cycle consists of:
  - Preheat time and temperature
  - Dwell time or solder contact time
  - Soldering temperature
- Time-temperature profile is affected by thermal mass differential of PCBA and rate of heat dissipation

Typical Thermal Profile

- Selective soldering process involves:
  - Application of heat and solder simultaneously
  - Solder contact time and solder temperature determine total heat flow to solder joints
  - Solderable surfaces must reach wetting temperature in short period of time to ensure capillary action and maximum vertical hole fill
- Time-temperature boundaries should be considered to avoid overheating of temperature sensitive TH components

Time-Temperature Limitations

- It is essential that all TH components reach wetting temperature:
  - Within short period of time
  - And in a uniform manner
  - To promote uniform capillary action and maximum vertical hole fill
- What must be taken into consideration is:
  - TH components of different thermal mass
  - Heat sinking effects of interconnections to ground planes

No-Clean Processing

- Fundamentals of Through-Hole Soldering
- Solderability Issues
- Solder Alloys
- Flux Deposition and Flux Activation
- No-Clean Processing
- Through-Hole Design Guidelines
- Quality Measurement
- Process Troubleshooting Guidelines
- Process Optimization Methods

No-Clean Thermal Processing

- Most no-clean fluxes have active ingredients such as mild organic acids
- These active ingredients are:
  - Active from time of application until consumed by reaction or volatized by heat
  - Proper thermal processing is essential to ensure level of active residue is acceptable
  - Bulk of active ingredients will be "burned off" by contact with solder nozzle
- Therefore uniformity of flux deposition, preheat and solder pot temperature are critical

Mitigation of Flux Residues

- Since preheat temperature is generally lower than breakdown points of active ingredients:
  - Preheat temperature is more important to solderability than reduction of flux residues
  - However inadequate preheating can adversely affect residue levels
- Direct contact with solder nozzle is essential ingredient to mitigate flux residues:
  - Solder pot temperature is critical
  - Increased dwell time helps remove excessive flux through physical and thermal means
Through-Hole Design Guidelines

- Fundamentals of Through-Hole Soldering
- Solderability Issues
- Solder Alloys
- Flux Deposition and Flux Activation
- No-Clean Processing
- Through-Hole Design Guidelines
- Quality Measurement
- Process Troubleshooting Guidelines
- Process Optimization Methods

Lead-to-Hole Aspect Ratio

- Proper consideration is important to assure complete vertical fill
- For intrusive reflow (PIH): lead dia. + .010” preferred, .012” maximum
- For wave and selective soldering: lead dia. + .010” preferred, .020” maximum
- Accepted best practice: maximum aspect ratio of 1.5:1.0 (PTH dia. vs. lead dia.)

Example of acceptable and unacceptable lead-to-hole aspect ratio

Lead Projection

- Proper consideration is imperative to prevent bridging between TH solder joints
- Minimum recommended lead projection is .040” for 100-mil lead pitch
- Maximum recommended lead projection: L (lead projection) =/< ½ P (lead pitch)

Example of recommended lead projection design rule

Lead Pitch

- 50-mil and 40-mil lead pitch is attainable in production environment
- 20-mil lead pitch is attainable with use of nitrogen air-knife assist as a special de-bridging technique
- Attention to lead projection is paramount for sub 100-mil lead pitch
Selective DFMA Considerations

- With respect to selective soldering it is recommended that attention be given to:
  - Component clearance – between TH pad and adjacent SMT pad
  - Lead-to-hole aspect ratio – lead dia. + .010” preferred, .020” maximum
  - Maximum lead projection – L (lead projection) ≤ ½ P (lead pitch)
  - Minimum lead pitch – 50-mil and 40-mil lead pitch is attainable but lead projection is critical

Adjacent Component Clearance

- The adjacent component clearance for selective soldering is equal to or less than most other soldering methods such as
  - Hand soldering
  - Paste-in-hole reflow
  - Wave aperture pallets
  - Fountain soldering
  - Point-to-point soldering
  - Multi-wave soldering

Recommended clearance between TH pad and adjacent SMT pad:
Critical Keep-Out Areas

- Under most conditions it is essential the selective soldering nozzle be allowed to over-travel the last rows of pins to prevent bridging
  - Multi-row connectors
  - Pin grid arrays (PGA)
  - Therefore a keep-out area must be included in the PCB design phase in lieu of solder thieves

Connector Over-Travel Clearance

Clear area is required at one end of a connector in lieu of solder thieves

PGA Over-Travel Clearance

Minimum of one clear area is required per corner of a PGA in lieu of solder thieves

Interlayer Construction

- The DPMA process should include a review of interlayer construction:
  - Confirm a limited number of layers are connected to any given PTH
  - Ensure optimal vertical plated through-hole fill
  - Avoid drawing thermal energy away from PTH that will adversely affect capillary action

Ground Planes

- Likewise every attempt should be made during the design process to avoid having ground planes directly connected to any PTH
  - Such designs should be re-defined
  - And/or have thermal relief design elements added

Interlayer Construction/Ground Planes

Example of ground plane directly connected to through-hole leads
Thermal Relief Design Rules

- It is recommended that thermal relief elements follow these design rules:
  - ID = drilled hole size + 2x annular ring
  - OD = ID + 0.020"
  - Spoke width = 0.008" minimum, 0.015" preferred
  - Rotate thermal reliefs of alternate layers by 45° to minimize PCB stress in Z-axis direction

Quality Measurement

- Fundamentals of Through-Hole Soldering
- Solderability Issues
- Solder Alloys
- Flux Deposition and Flux Activation
- No-Clean Processing
- Through-Hole Design Guidelines
- Quality Measurement
- Process Troubleshooting Guidelines
- Process Optimization Methods

Solder Joint Inspection Criteria

- When assessing solder joint quality of a PCBA it is essential to determine inspection criteria with respect to the applicable end-user requirements:
  - Class 1
  - Class 2
  - Class 3
  - Or other special end-customer requirements

Solder Joint Inspection Criteria

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Class 1</th>
<th>Class 2</th>
<th>Class 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>1)</td>
<td>Not specified</td>
<td>180°</td>
<td>230°</td>
</tr>
<tr>
<td>2)</td>
<td>Not specified</td>
<td>75%</td>
<td>75%</td>
</tr>
<tr>
<td>3)</td>
<td>250°</td>
<td>250°</td>
<td>300°</td>
</tr>
<tr>
<td>4)</td>
<td>Not specified</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>5)</td>
<td>25%</td>
<td>75%</td>
<td>25%</td>
</tr>
</tbody>
</table>

Typical solder joint inspection acceptance criteria

Post-Soldering Inspection Protocols

- When monitoring TH solder joint quality several solder joint formation elements can be measured:
  - Presence or absence of bridging source side
  - Presence or absence of excessive solder source side
  - Through-hole vertical hole fill
  - Destination side wicking distance
  - Destination side circumferential wetting angle
  - Intermetallic thickness and intermetallic microstructure (harsh environment applications)
Inspection Methodologies

- Following test methods should be used:
  - Presence or absence of bridging or excessive solder (microscope)
  - Through-hole vertical hole fill (X-ray or potting and cross sectioning)
  - Destination side wicking distance and circumferential wetting angle (endoscope-based microscope)
  - Intermetallic thickness and intermetallic microstructure (scanning electron microscope)

Pareto Analysis of Defect Type

- TH solder defects should be examined with the objective of improving and refining the selective soldering process
  - This should include an analysis of defect reports for recent production batches
  - Defects from NPI and prototype runs
  - Defect data should be formatted into a Pareto chart to determine types of defects and the frequency of occurrence
  - Is the first requirement in determining root cause and corresponding corrective actions

Defect Frequency and Location

- Further analysis of solder defects must be made to identify location, frequency of occurrence and component type
  - This information is helpful in determining component specific related root cause such as
    - Inadequate heat transfer
    - Component thermal mass
    - Lead-to-hole design guideline violations
    - Component solderability related issues

Example of through-hole solder defects sorted by defect type

Example of TH solder defects sorted by reference designator
Process Troubleshooting Guidelines

• Fundamentals of Through-Hole Soldering
• Solderability Issues
• Solder Alloys
• Flux Deposition and Flux Activation
• No-Clean Processing
• Through-Hole Design Guidelines
• Quality Measurement
• Process Troubleshooting Guidelines
• Process Optimization Methods

Defect Condition/Root Cause Analysis

• Some of the more common process related selective soldering defects include:
  – Insufficient hole fill
  – Insufficient solder
  – Blow holes and voids
  – Excessive solder
  – Bridging
  – Solder balls
  – Poor hole filling
  – Pin-to-pin solder short
  – Adjacent solder short
  – Sunken solder joints

Insufficient Hole Fill

• Possible Causes:
  – Inadequate flux
  – Topside PCB temp. too low
  – Lead-to-hole ratio too small

• Preventive Actions:
  – Verify flux deposition
  – Verify preheat temp.
  – Check wave height
  – Check lead-to-hole aspect ratio
  – Verify internal ground planes

Insufficient Solder

• Possible Causes:
  – Inadequate flux
  – Dwell time too short
  – Poor solderability
  – Contaminated pads

• Preventive Actions:
  – Verify flux deposition
  – Reduce drag speed
  – Verify preheat temp.
  – Check component contamination
  – Check board contamination

Blow Holes and Voids

• Possible Causes:
  – PCB temp. too low
  – Flux vehicle out-gassing
  – Moisture in PCB
  – Un-cured solder mask

• Preventive Actions:
  – Verify topside PCB temp.
  – Verify flux deposition
  – Check for moisture in laminate
  – Check for defective mask material

Excessive Solder

• Possible Causes:
  – Lead length too long
  – Solder speed too low
  – Excessive flux
  – Solder temp. too low
  – Contaminated solder

• Preventive Actions:
  – Reduce lead projection
  – Increase drag speed
  – Verify flux deposition
  – Increase solder temp.
  – Check for solder contamination
**Bridging**

- Possible Causes:
  - Lead length too long
  - Inadequate flux
  - Solder temp. too low
  - Solder speed too fast

- Preventive Actions:
  - Reduce lead projection
  - Verify flux deposition
  - Increase solder temp.
  - Decrease drag speed
  - Re-program peel-off movement

**Solder Balls**

- Possible Causes:
  - Excessive flux
  - Topside PCB temp. too low
  - Nitrogen level
  - Solder mask porosity

- Preventive Actions:
  - Reduce flux vehicles
  - Verify topside PCB temp.
  - Decrease nitrogen level
  - Greater tendency with selective vs. wave

**Poor Hole Filling**

- Possible Causes:
  - Inadequate flux
  - Topside PCB temp. too low
  - PCB contamination
  - Poor solderability

- Preventive Actions:
  - Verify flux deposition
  - Verify topside PCB temp.
  - Check component contamination
  - Check board contamination

**Pin-to-Pin Solder Short**

- Possible Causes:
  - Lead length too long
  - Lead pitch too small

- Preventive Actions:
  - Reduce lead projection
  - Reduce drag speed
  - Re-program peel-off movement
  - Increase nitrogen level

**Adjacent Solder Short**

- Possible Causes:
  - Inadequate TH to SMT adjacent clearance
  - Lead length too long

- Preventive Actions:
  - Implement keep-out DFM guidelines
  - Re-program nozzle movement
  - Re-program peel-off movement

**Sunken Solder Joints**

- Possible Causes:
  - Non-wetting or de-wetting
  - Inadequate flux
  - Topside PCB temp. too low
  - Lead-to-hole ratio too large

- Preventive Actions:
  - Check solderability
  - Verify flux deposition
  - Verify preheat temp.
  - Check lead-to-hole aspect ratio
TH Solder Defect Cause-Effect Matrix

- An assembly process map of TH soldering must include all four aspects that have a direct bearing on the quality of solder joint formation including:
  - Process parameters
  - Printed circuit board assembly
  - Board fabrication techniques
  - Board design

Process Optimization Methods

- Fundamentals of Through-Hole Soldering
- Solderability Issues
- Solder Alloys
- Flux Deposition and Flux Activation
- No-Clean Processing
- Through-Hole Design Guidelines
- Quality Measurement
- Process Troubleshooting Guidelines
- Process Optimization Methods

Optimization of Process Parameters

- Underlying principles of process characterization:
  - Ensure preheat and soldering temperatures are reduced to a minimum
  - Minimize PCBA thermal stress during the selective soldering process
- Process characterization work should be:
  - Carried out in accordance with a well-defined soldering process work plan
  - All quality measurement metrics should be carried out in accordance with a well-defined inspection work plan

Process Optimization Objectives

- Objectives of a process optimization activity:
  - Optimization of flux deposition and TH penetration
  - Establish actual temperature excursions
  - Obtain flux activation at lowest possible preheat settings
  - Monitor flux activation window and flux survivability
  - Monitor solder joint formation temperature and solder nozzle temperature
  - Monitor critical interlayer temperature
  - Minimize thermal shock to PCBAs between preheat stage and soldering unit

DoE Step by Step

- Tasks
  - Verify machine is in good working condition
  - Correlate topside PCB temp. and preheat settings
  - Identify defects to be monitored
  - Establish sample size based on past defect frequency
  - Verify defect inspection and data collection protocol
- Structure
  - Establish range of process variables, or factors:
    - Flux deposition
    - Topside PCB temperature
    - Solder pot temperature
    - Solder dwell time or traverse speed
    - Determine min., mid-point and max. value levels
  - Map DoE matrix of machine settings
DoE Step by Step

- Methodology
  - Measure ‘positive’ and ‘negative’ process indicators
  - Conduct as full factorial experiment consisting of:
    - Four (4) factors at three (3) discrete levels
    - Take into account all possible combinations
    - Review effects on each factor or response variable
    - Review effects of interactions between factors

- Analysis
  - 4-factor experiment conducted at 3 discrete values:
    - Mathematically expressed as \(3^4\) or 81 possible combinations
      - Of which 24 are significant combinations
      - Analyze by analysis of variance between groups (ANOVA)
    - Graphically represent results with interaction plots

DoE Methodologies

- Establish range of machine settings based on:
  - Regulate flux deposition limits based on flux mfg. recommendations
  - Determine topside PCB temperature limits based on flux mfg. recommendations
  - Correlate topside PCB temperature limits with corresponding preheat settings
  - Measure flux activation and survivability
  - Determine solder pot temperature and solder dwell time or traverse speed based on flux mfg. recommendations

- Based upon the previous machine settings to process parameter correlation, a range of experiments should be defined:
  - It should be noted that each variation of a given process parameter should be conducted while other process settings are retained at their mid-point to assess the impact of each individual process parameter

- Measure discrete metrics such as:
  - Through-hole vertical fill ‘positive’ indicator
    - Express as percentage of vertical hole fill
    - The higher the value the better
  - Presence of bridging ‘negative’ indicator
    - Express as frequency of occurrence
    - Measure all solder joints
    - The lower the value the better
  - Graphically represent DoE results as shown on following slides
In the above example the following machine settings were determined to produce optimal results:

- 2x35% (double spray) flux deposition
- 105ºC topside PCB temperature
- 290ºC solder pot temperature
- 3.00mm/sec traverse speed

The DoE methodology does work:
- For this example post-selective soldering rework was reduced from 60% to 25%
**Validation Run**

- The purpose of a validation run is to confirm optimal process parameters with fully functional PCBAs processed through:
  - All follow-on inspection and test operations such as ICT, boundary scan, functional test, burn-in and thermal cycling
- Ideal time to determine effects of moisture in PCB laminate if experiencing defects such as:
  - Insufficient vertical hole fill, solder voiding or solder balls
  - Conduct with two (2) controlled lots, one with PCBs baked for 4 hours @ 120ºC and one with unbaked PCBs

**Defect Mapping Techniques**

- In cases where random anomalies are detected on an infrequent basis:
  - Defect mapping can be used to identify a pattern of frequency of occurrence
  - An example of defect mapping is shown on the following slide
- In this case analysis confirmed:
  - Highest insufficient hole fill frequency of occurrence at pins 2 and 121 due to design guideline violation
  - Random insufficient hole fill on second row connector soldered due to nitrogen shroud burning off some required flux

**Solder Alloy Contamination Levels**

- Most lead-free solder alloys:
  - Dissolve copper at a fast rate and can exhibit sluggish behavior over prolonged operation
  - Copper-tin (CuSn) intermetallic will sink and disperse throughout solder pot
  - Some lead-free alloys will leach machine elements creating iron-tin (FeSn₂) contamination
- Tests have shown some lead-free solder alloys cause corrosion to base metals due to aggressive nature of tin at high temperatures effecting:
  - Solder pots, impellers and solder nozzles

**Commonly Used Base Metals**

<table>
<thead>
<tr>
<th>Material</th>
<th>Alloys Used</th>
<th>Corrosion Resistance</th>
<th>Half-Life</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>304 stainless steel</td>
<td>Inconel</td>
<td>No corrosion resistance</td>
<td>2.5 months</td>
<td>2.5 years</td>
</tr>
<tr>
<td>316 stainless steel</td>
<td>Inconel</td>
<td>No corrosion resistance</td>
<td>2.5 months</td>
<td>2.5 years</td>
</tr>
<tr>
<td>Surface coated case</td>
<td>Stainless steel</td>
<td>Good resistance to corrosive effects of tin</td>
<td>5 years</td>
<td>Can be reprocessed</td>
</tr>
<tr>
<td>Surface coated case</td>
<td>Stainless steel</td>
<td>Good resistance to corrosive effects of tin</td>
<td>5 years</td>
<td>Can be reprocessed</td>
</tr>
</tbody>
</table>

Summary of commonly used base metals.
Solder Alloy Contamination Levels

• It is highly recommended that lead-free solder alloys be analyzed:
  – Every 8,000 boards or once every 3 months whichever occurs first
• Analysis should be conducted for:
  – Copper (Cu)
  – Iron (Fe)
  – Lead (Pb)
  – As well as other possible contaminants

OSP/Copper Dissolution Tech Tip

• When selective soldering high volumes of boards with OSP finish you can minimize the effects of copper dissolution by
  – Using SAC300 solder alloy instead of SAC305
  – SAC300 (Sn96.95Ag3.0Cu0.05) provides a larger guard band to remain below the 1.0% Cu threshold in lieu of the traditional SAC305 (Sn96.5Ag3.0Cu0.5) alloy

Questions...

Contact Information

Bob Klenke
ITM Consulting
Tel: (414) 899-3772
Email: bob_klenke@itmconsulting.com
www.itmconsulting.com