SMT Trouble Shooting Guide

PCBA DEFECT LIST WITH POSSIBLE CAUSES AND SOLUTIONS

PROBLEM	Column1	POSSIBLE CAUSES	Column2	POTENTIAL SOLUTIONS
Cold Solder		Insufficient reflow temp		Raise reflow temperature
(Dull Joint)		PCB contamination		Prevent vibration of assembly during and immediately after reflow
				Accelerate cool down
		Solder impurities		Check alloy
Solder Balling		Paste quality-Oxidized paste		Increase flux activity
		or insufficient flux-Excess fines		Reduce fines in powder
		Excess paste on pad		Reduce squeegee pressure
				Reduce aperture size in stencil
		Moisture in the paste		Storage, handling and printing environment- humidity reduction
		Improperly processed solder mask		
		Cleaning solvent on the PCB		Ensure cleanliness or clean PCB's before printing paste
		Temp ramp up excessive		Adjust reflow oven settings
		Misprinted board inadequately cleaned prior to reprocessing		Inspected cleaned boards before returning to line
Solder Beading		Excessive solder paste deposit		Reduce stencil thickness or aperture dimension
(side balls)				
		Excessive placement pressure		Reduce component pick and place pressure
Bridging		Excessive placement pressure		Reduce placement pressure
		Reflow rate too high		Adjust heat profile
		Solder paste slump		Increase metal content
		Low metal content - low viscosity		Increase paste viscosity
				Reduce squeegee pressure
		Paste smearing on bottom of stencil		Reduce squeegee pressure
		Runny paste-domed pads		Print on contact
				Increase metal content or paste viscosity
		Inaccurate registration		Reset PCB registration
Bridging		Component placement error or movement		Reset placement machine or PCB registration
(CONT.)				
		Hand repositioning of components		Process controls
		Excessive solder paste		Reduce aperture size
				Reduce squeegee speed
		Stencil Design		Laser cut stencil with trapezoidal apertures and electro-polishing
Poor Wetting		Flux activity		Obtain more active paste
		PCB contamination		Test PCB surface
		PCB plating		Inspect PCB plating thickness
		Preheat time/temp		Ensure preheat time/temp per paste manufacturer specification
Opens		Insufficient solder-partly covered pads		Lower viscosity paste
		Insufficient flux-Excess fines		Check squeegee pressure (too high/low) and speed
		Wicking-uneven heating (often in VP)		Adjust heating profile
				Check thermocouples
		Component coplanarity of leads		Check for high leads
		Soldermask or contaminant on pads		Inspect PCB before printing
		Adhesive on Pads		Inspect PCB before printing

PCBA DEFECT LIST WITH POSSIBLE CAUSES AND SOLUTIONS

Tombstoning	Component solderability	Test solderability of component on bare ceramic
(drawbridging)		Increase activity on flux
	Component placement	Ensure placement is uniform
	Reflow rate too fast	Adjust heat profile
	Uneven heat transfer (often VP)	
	Differential Wetting	Test solderability of component on bare ceramic coupon
Tombstoning	Inconsistent solder paste height	Screen printer set up
(CONT.)		Board planarity
		Fixture planarity
Unmelted	Reflow profile-low peak	Increase temp and/or time above °C liquidus of alloy
Paste		
	Wrong alloy	Check chemistry of alloy or melting point on a hot plate
Disturbed joint	Movement during cooling	Look for cause of movement and eliminate
Cracked Chips	Excessive solder - "bulging joints"	Reduce aperture size or stencil thickness
	Too fast heating rate-e.g. 5/sec	Reduce heat up rate
Insufficient fillet	Insufficient solder-stencil design	Check apertures on stencil
	Soft squeegee	Check squeegee pressure, Consider metal squeegee
	Paste does not roll	Stencil surface
	Improperly etched or designed stencil	New stencil
	Uneven or too small openings for pad size	Consider laser cut
	Stencil too thick	Consider more fluid paste
	Paste sticking in openings	Lower tack
	Sticky paste	Reconsider necessity of thick stencil
	Squeegee too fast	Slower squeegee speed
	Snap off too high- more than .001/diagonal high	On contact printing recommended especially with laser cut
	Too little metal in paste-depends on pitch	Check metal percentage in paste
	Time at reflow temperature too short	Extend time at reflow
	Poor component solderability	Reject components
		Use more active flux system
		Have vendor check plating bath
Excessive fillet	Incorrect pad/lead position	Check stencil design for aperture reduction
	Solder mask thickness adding too much paste volume	Is paste viscosity correct?
	Stencil too thick-width to height ratio under 1 (Wa/Ts)	Ratio recommended greater than 1:1
	Component placement	Ensure placement is uniform
Slump	Low paste viscosity	Obtain correct paste
	Out of spec	
	Wrong rheology	Obtain correct paste
	Too much shear thinning	
	Incorrect use of thinner-too much	Correct paste should not require thinning
	Wrong kind	

	Room/environment temperature too high	Better control of print/place environment
	Insufficient snap off-less than 1.5x stencil thickness	
Clogging (Dispensing)	Dispensing needle clogging with paste	Reduce paste powder size
		Reduce paste viscosity
		Reduce paste metal content
		Shorten needle length
		Increase needle orifice diameter
		Change syringe/cartridge design
		Increase air pressure