PCBA PROCESS MANUAL

Manual	WHQ- SQD- MAN003	Rev.7	Effective from:	Released
Corporate standard:	Number:	Version:	20.09.2017	Status:

WITTUR

PURPOSE

The following procedure has the purpose to define requirements are based on technology requirements IPC/J-STD standards and best practice of SMT manufacturing. All CEM/EMS should follow the requirements and work on continuous process improvement in order to decrease PPM level and meet requirements listed in this standard.

SCOPE

This standard was developed in order to set list of technological process requirements for all CEM/EMS who produce PCBAs for Wittur Group including Wittur design contract manuafaturers or supplier who deliver finish goog based on own design.

REFERENCE

IPC-A-600 Acceptability of Printed Boards
IPC-A-610 Acceptability of Electronic Assemblies
IPC / WHMA-A-620 Requirements and Acceptance for Cable and Wire Harness Assemblies
IPC-7525 Stencil Design Guidelines
IPC-7527 Requirements for solder paste printing
IPC-7711 / 7721 Rework, Modification and Repair of Electronic Assemblies
IPC/EIA J-STD-004 Requirements for Soldering Fluxes
IPC/JEDEC J-STD-033 Joint IPC/JEDEC Standard for handling, packing, shipping and use of moisture / reflow sensitive surface mount devices
IPC-1601 Printed Board Handling and Storage
IPC-TM-650 Test Methods Manual
ISO 9001 Quality Management Systems
JESD625 Requirements for handling Electrostatic-Discharge-Sensitive (ESDS) Devices
ISO 2859-1 OR ANSI/ASQ Z1.4 sampling procedure implemented for incoming material inspection
ANSI/ESD-S-20.20 Protection of Electrical and Electronic Parts, Assemblies and Equipment. **IEC 61340-5-1, IEC 61340-5-2**. Protection of electronic devices from electrostatic phenomena
ANSI/ESD S541; JESD625-A For the Protection of Electrostatic Discharge Susceptible Items. Packaging Materials for ESD Sensitive Items
J-STD-002 Solderability Tests for Component Leads, Terminations, Lugs, Terminals and Wires
J-STD-020 Moisture/Reflow Sensitivity Classification for Nonhermetic Solid State Devices
J-STD-033 Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices
IPC-1066 Marking, Symbols and Labels for Identification of Lead-Free and Other Reportable Materials in Lead-Free Assemblies, Components and Devices
IPC-7530 Guidelines for Temperature Profiling for Mass Soldering Processes (Reflow & Wave)
IPC-S-816 SMT process Guideline and checklist
WHQ_IMS_PR013 – Traceability suppliers

RESPONSIBILITY

PROCESS OWNER	Supplier Quality Development
Corporate	SQD are responsible to apply the procedure for all eligible suppliers and monitoring the deployment of the defined procedures at local level.
Local	Local Quality is responsible to monitoring and reporting the performance and alert the Corporate department in case of serious accidents (Safety and/or Functional), repetitive quality issue, cross-factory quality issue.

RELATED DOCUMENTS

Procedures	WHQ_SQD_PR005 – WPPAP
Instructions	NA
Forms	FR001 [WHQ_SQD_MAN003] PCBA Process Audit Questionnaire FR002 [WHQ_SQD_MAN003] ESD Audit Assessment FR003 [WHQ_SQD_MAN003]PCBA supplier ICT fixture validation FR004 [WHQ_SQD_MAN003] TOTAL LINE REJECTS FOLLOW UP FR005 [WHQ_SQD_MAN003] New A3 supplier analsis FR006 [WHQ_SQD_MAN003] Standard for pre-analysis FR008 [WHQ_SQD_PR002] TFP Dashboard for electronic components
Other	TS003 [WHQ_SQD_PR002] Electronic Components Quality Agreement

Issue history	Date	Ed.	Description of change	Editor	Approvers
First	23.08.2016	01	First released version	V. Vovkanets	
Current	20.09.2017	07	Updated to NEW corporate form and coding	V. Vovkanets	A. Aviles
Effective from	20.09.2017				
Planned revision	20.09.2018				

TABLE OF CONTENT

0. TERMS AND DEFINITIONS

AVL Approved Vendor List **AML** Approved Manufacturing List
AOI Automatic optical inspection
CAD Computer Aided Design
CTQ Critical to Quality
CEM Contractual Electronic Manufacturer
EOL end of line
ESD electrostatic discharge
EOS Electrical overstress
EPA Electrostatic protective area
ERP Enterprise resource planning
FCT Functional in Circuit Tester
FMT Functional module testing
FiFo Fist in Fist out
FPY First pass yield
HIC Humidity indicator card
HBM Human body model
ICT In Circuit Test
IPC Association Connecting Electronics Industries
KPI Key process indicators
PCB Printed circuit board
PCBA Printed circuit board assembled
Panel PCB or PCBA that includes technical edge and several boards.
MLCC Multi-Layer Ceramic Capacitor
MSD Moisture sensitive device
MSL Moisture sensitive level
NPI New product introduction
NTF/NDF No trouble found/No defect found
PPM Parts per million
SAM Scanning acoustic microscopy
SEM Scanning electronic microscopy
SMT surface mount technology
SPI Solder Paste Inspection
STI Specification for industrial testing
SPM – solder print machine
SOP Start Of Production
THT Trough hole technology
WIP Work In Process
WH - Warehouse

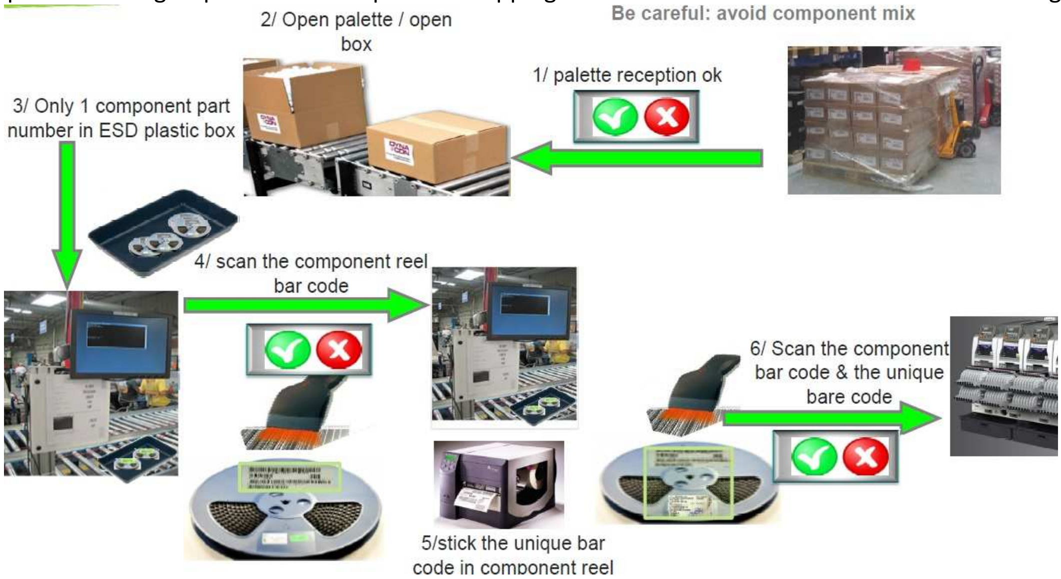
1. MATERIAL HANDLING REQUIREMENTS

1.1 WH generic storage requirements

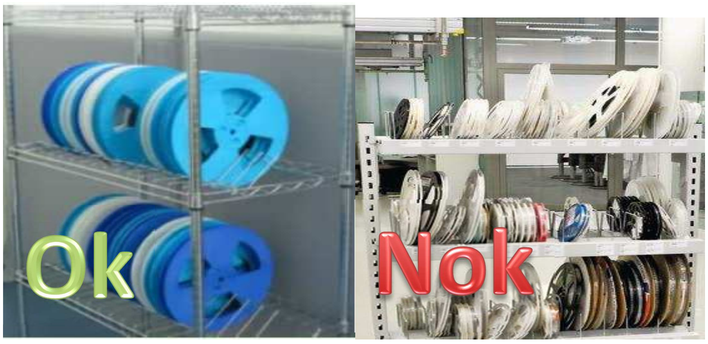
AML/AVL CEM is allowed to deliver the parts only from approved list of manufacturers, vendors and distributors. List of approved suppliers/distributors must be available. CEM has to assure that risk to receive counterfeit parts is eliminated. In order to validate new source CEM must do environment validation tests and agree with Wittur. CEM must have at least 2 sources for standard components to avoid risk of line stop or switch the source in case of quality issue.

In addition a serious **incoming inspection** must be carried out comparing the standard parts with the suspect parts using visual picture, X-ray picture, Electrical result of the ICT and FCT, checking of the dimensions of the part according to the datasheet or measure electrical parameters with multimeter.

There must be dedicated receiving workplace with clear defined process input/output, one piece flow must be respected. **Receiving** must start from scanning and checking of manufacturer label. All info about MPN date code and Manufacturer has to be linked into internal ERP system (for. Ex. SAP). CEM is allowed to print and place additional label on the reel/package, but all the data must be traceable back. CEM shall establish procedures, methods and forms to ensure a two-way traceability for purchased parts receiving inspection to the specified shipping container with info of date code and receiving date.



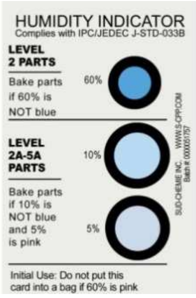
Storage conditions have to assure **shelf life** and avoid damage of components. CEM must have environment climate control system which will keep room temperature conditions and humidity under control. **FiFo** and full **traceability** of shelf and location to be respected: physical location and quantity to be aligned with data on system. Lean and 5S rules also have to be applied at WH. Transportation from WH to production and storage at production is allowed only on special trolleys or carriers. -



CEM must have closed dedicated zone for **non-conform** products. Area must be locked physically and separated, marked by red. Blocking are also has to be shown on the system and traceability on the system must align with traceability physically (PN, q-ty, date code etc).

1.2 PCB handling

PCBs must be delivered on vacuum pack which prevents against humidity and oxidation. All packaging should have humidity indicator inside. If silica gel is used - has to be placed to the side of PCB do not touch pad area. PCBs with damaged packing or without vacuum should be prohibited to use without Wittur approval.



Shelf life is a function of solderability. Solderability of any final finish degrades over time due to oxidation or intermetalic compound formation of different finishes. Per IPC-4552 **ENIG** shall remain solderable for up to 6 months under controlled storage conditions. Per IPC-4553 **Im Ag** shall remain solderable for up to 6 months under controlled storage conditions. Per the IPC the solderability of OSP shall be per the procurement documentation. OSP shelf life depends upon the OSP vendor and grade, usually life range of 3 to 6 months.

Per IPC-1601 **Tin/Lead Hot Air Solder Leveling HASL** has an acceptable shelf life ranging from 6 months to a year. The bags should be inspected to verify there are no holes, gouges, tears, punctures or openings of any kind that would expose either the contents or an inner layer of a multilayer bag. If openings are found, and the humidity indicator card (HIC) indicates maximum humidity has been exceeded, then the parts should be baked In case of damaged packaging or humidity indicator not blue color any more CEM has rights to bake the PCBs under conditions described below:

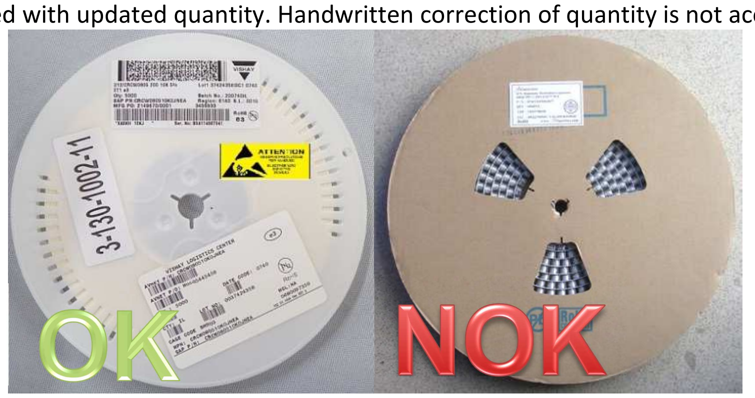
Table 3-1 Recommendations for Printed Board Baking Profiles

Final Finish	Temperature	Time	Comments
Tin	105 – 125 °C	4 – 6 Hours	Baking may reduce solderability. See 3.4.1.5
Silver	105 – 125 °C	4 – 6 Hours	Silver may tarnish. See 3.4.1.4
Nickel/Gold	105 – 125 °C	4 – 6 Hours	Usually no issue with extended bake on Nickel/Gold finish. See 3.4.1.2
Organic Coating	See 3.4.1.1		
HASL/HAL	105 – 125 °C	4 – 6 Hours	Final surface thickness below 0.77 µm [30 µin] may turn into pure intermetallics and render the printed board unsolderable

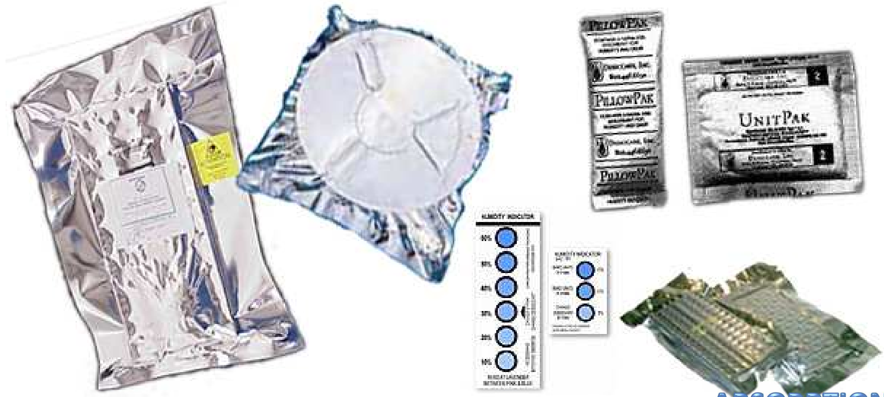
For complete handling, baking and storage parameters refer to IPC-1601.

1.3 Components SMT

All SMT components which will enter to EPA has to have ESD protective packaging in order to avoid ESD on sensitive devices. There should be ESD safe marking on the packaging. Carton reels are not allowed due to not ESD protective packaging and risk of particle contamination. In order to reduce the risk of bending pin and loose of component traceability, it is forbidden to count the component by re-reeling or by open tray packaging for component return from line. The placement tools are capable to count the components; the PCBA supplier must use this functionality. For components which return from the line - new lable must be reprinted with updated quantity. Handwritten correction of quantity is not acceptable.



MSD must be delivered on vacuum packaging. Damaged packing or without vacuum should be prohibited to use without Wittur approval. All packaging should have humidity indicator inside and silica gel for humidity absorption. For MSL >=3 component return from the line, the best practice is the usage of dry-cabinet with a software management system on the timing. Mandatory for very sensitive component like LED with dome resin or MSL>=4 (risk of delamination and damaging).



VACUUM PACKED MSD HUMMIDITY INDICATOR ABSOBRTION GEL

If packaging was damaged and vacuum lost - CEM has right to bake the parts based on **J-STD-033** requirements. MSL level usually notified on packaging of component. For baking requirements please see table below:

January 2007 IPC/JEDEC J-STD-033B.1 - Includes Amendment 1

Table 4-2 Default Baking Times Used Prior to Dry-Pack that were Exposed to Conditions ≤60% RH (Supplier Bake: "MET" = 24 h)

Package Body Thickness	Level	Bake @ 125°C	Bake @ 150°C
≤1.4 mm	2	7 hours	3 hours
	2a	8 hours	4 hours
	3	16 hours	8 hours
	4	21 hours	10 hours
	5	24 hours	12 hours
	5a	28 hours	14 hours
>1.4 mm ≤2.0 mm	2	18 hours	9 hours
	2a	23 hours	11 hours
	3	43 hours	21 hours
	4	48 hours	24 hours
	5	48 hours	24 hours
	5a	48 hours	24 hours
>2.0 mm ≤4.5 mm	2	48 hours	24 hours
	2a	48 hours	24 hours
	3	48 hours	24 hours
	4	48 hours	24 hours
	5	48 hours	24 hours
	5a	48 hours	24 hours

Note 1: If baking of packages >4.5 mm thick is required see appendix B.
Note 2: The bake times specified are conservative for packages without blocking planes or stacked die. For a stacked die or BGA package with internal planes that impede moisture diffusion the actual bake time may be longer than that required in Table 4-2 if packages have had extended exposure to factory ambient before bake. Also the actual bake time may be reduced if technically justified. The increase or decrease in bake time **shall** be determined using the procedure in JEDEC JESD22-A120 (i.e., <0.002% weight loss between successive readouts) or per critical interface concentration calculations.

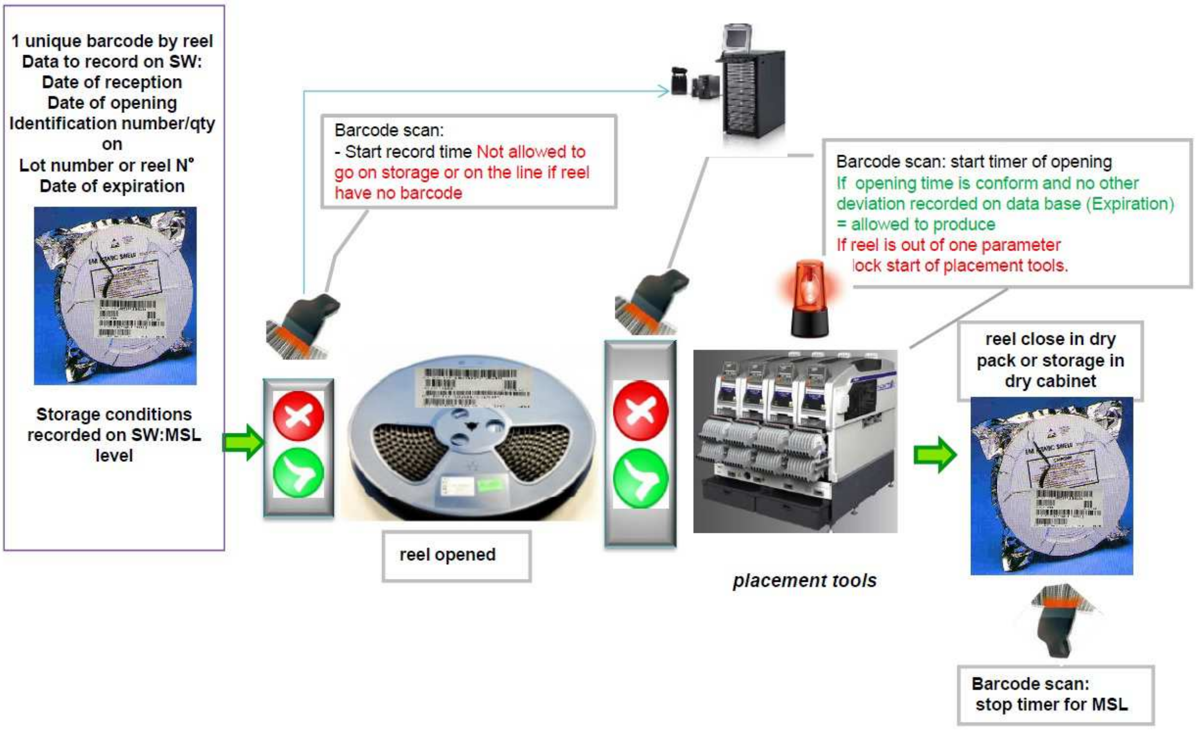
After backing CEM has rights to **repack** the component reel or tray with vacuum machine, but vacuum pressure has to be taken into account as too less vacuum pressure will not provide sufficient protection and too much vacuum pressure will increase the risk of pushing leads of components which can cause bending pin or can damage plastic component body. For reel components has to be used protective tape, so called reel lock, If component delivered in trays - special fixation straps needs to be used. Silica gel and humidity indicators not allowed to reuse.

REEL LOCK TAPE

TRAY LOCK STRAPS

Shelf life minimum shelf life capability in sealed dry-bags of 12 months from the seal date. After extending this time is recommended to bake the parts. SMD product can be sensitive to picking up moisture within the molding compound of the package and, when undergoing board solder reflow, can have delamination occur between the molding compound and the lead frame

For opening time control CEM can develop MSL software control with scanning as technology best practice. If no software control – minimum requirement label for open time counting. For MSL 1 classification means that the package is very robust and not susceptible to moisture influences at reflow; thus, the “floor life” (the time parts can be exposed to ambient temperatures and humidities) is unlimited for MSL 1 (≤30°C / 85% RH). For MSL 2 parts, the floor life is 1 year at ≤30°C / 60% RH.



1.4 Components THT

CEM has to assure as minimum requirement storage condition within room temperature and avoid high humidity influence. Packaging has to prevent components against dust contamination and damaging. Automatic environment control is best practice.

Each package of THT component must have unique lable code per each package and have the same reverse traceability as SMT components. Handwitten lables not allowed - if quantity modified - lable must be reprinted.

FiFo to be automatically controlled by ERP. THT critical components: aluminium electrolitic capacitor, resistor, fuse, THT LED etc - must have ERP controlled expiration time per each package and required max. 24 month. THT components must be submitted to **shelf life requirements** defined by component manufacturer. Note that through-hole devices and devices which will be welded, crimped, or hand-soldered are exempt from MSL considerations, because the package body does not exceed the reflow profile temperatures found in board soldering. 100% tin-plated parts are subject to minor levels of oxidation from exposure to humidity and temperature. In order to decide if solderability properties not lost. CEM has to perform solderability testing if nothing additional is specified by component manufacturer.

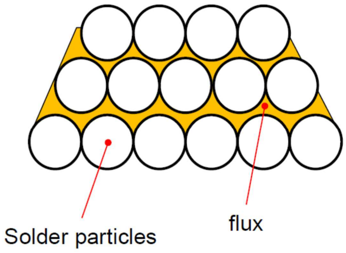
CEM has to assure back traceability of component batch for THT components linked to PCBA serial number, mixing of different batch numbers are not allowed. FiFo for THT components must be respected. CEM has to assure back traceability of component batch for THT components. Unique package code must be scanned linked to PCBA serial number, mixing of different batch numbers are not allowed. FiFo for THT components must be controlled by ERP. If any handling boxes, trays used for THT handling on the line must be marked properly traceable to avoid mixing part numbers and different batches.

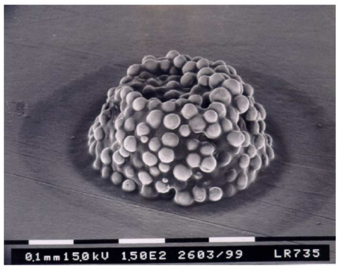
Non-ESD sensitive THT devices must be handled on the line on ESD complaint packaging only due to risk to bring charge to PCBA and damage other components. All the packaging used for THT components must prevent components from damage. Carton packaging must be eliminated from THT area due to risk of particle contamination.

ESD COMPONENT TRAY TRACEABILITY

1.5 Solder paste

Solder paste is comprised of powdered solder alloy suspended in a flux. There is a group of flux ingredients that is generically identified as "activators". It is the activators whose primary function is to remove oxides not only on the surfaces that are being soldered but any oxides that may be present on the solder powder, itself.

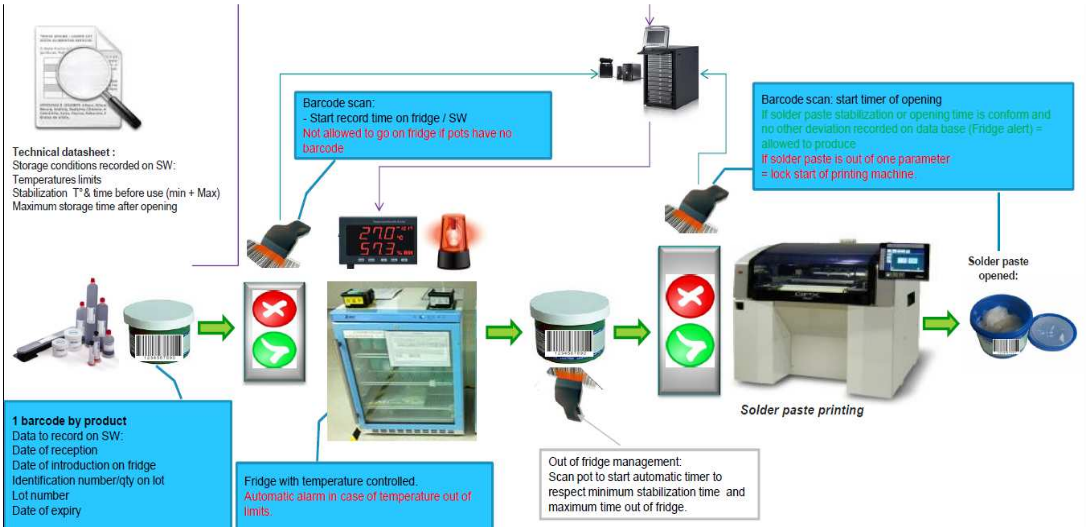




Solder paste storage. Solder paste has to be shipped cold, with an approximate temperature of 0°C to 5°C. Refrigerated storage required (1°C to 10°C), as it prolongs the shelf life of the solder paste. If stored at the cold temperatures recommended by solder paste supplier – shelf life usually 6 month, if storage conditions are respected. CEM has to track temperature on the fridge at least once per 6 hours. Best practice to have automatic alarming system in case if temperature on the fridge went out of tolerance. Solder paste should be inventoried on a FIFO.

Prior to use, allow the solder paste to slowly be brought up to room temperature. It is highly recommended that solder paste be removed from refrigeration at least 6 hours prior to printing. Removing solder paste from storage 24 hours before use is recommended. Not allowed to rapidly warm the solder paste. Not allowed to open the solder paste while warming as it may cause condensation in the solder paste. Not allowed to store solder paste at temperatures in excess of 25°C, as it may result in significant flux separation and/or chemical decomposition. Solder paste that is damaged by heat can lead to poor printing performance due to higher viscosity and/or reduced tack caused by chemical decomposition.

All opened jars or cartridges should be properly labeled with the time and date of opening. Best practice would barcode scanning system to control storage and opening time and Poka-Yoke scanning before use, which would prevent usage of solder paste where storage conditions were not respected. Open jars should be not used longer than 8 hours as after this time paste is losing it properties. Best practice is to have barcode scanning system for storage and **shelf life control** by software. If CEM has not developed such system as temporary solution can be used labelling system for solder paste handling control. Shelf life expiration date, alloy info and lot number must be written on the manufacturer label.

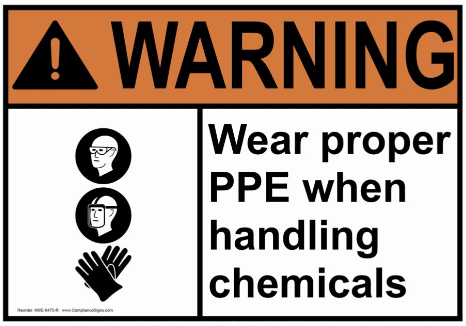


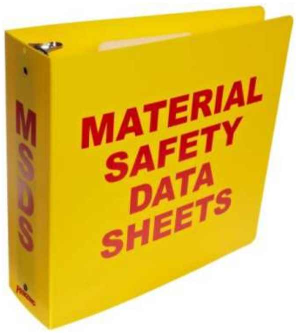
Reuse of solder paste is forbidden. Mix together different type of solder paste or remaining solder paste from different packages is forbidden. Remaining solder paste should not be reused, must be scrapped.

1.6 Hazardous materials storage

Wittur product requires lead free process therefore only lead free solder alloy is allowed. The CEM will comply with Lead-Free / Hazardous substance regulations in force along with Wittur. CEM will apply the country legislation and regulation for the final customer market, for example for the European market apply ROHS directive. CEM has no right to use at the same line leaded and lead free processes, maintenance also has to be spltted, and all alloys must have appropriate marking.

Nevertheless, during SMT process numerous process chemicals are used: cleaning liquids, fluxes, glue, coating, maintenance solutions etc. Process chemicals must be stored and handled according to manufacturer’s datasheet recommendations. Expirations date of process materials and chemicals defined by the manufacturers is obeyed and expired chemicals are scrapped. All chemicals in production has identification and traceability label. Hazardous substances are stored in a dedicated space according to manufacturers' recommendations. Safety rules have to be described and users have to be trained, necessary protective tools to be provided for production. Material safety datasheet has to be present on the line for operators use.

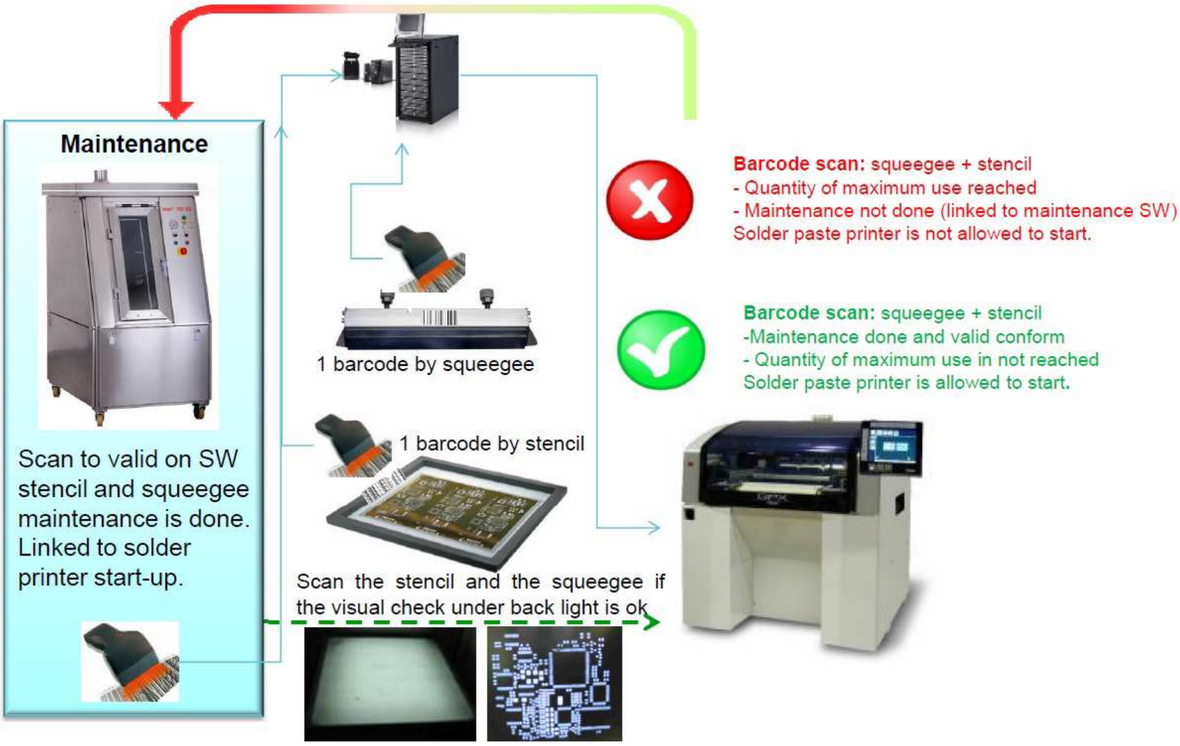




2. MAINTENANCE REQUIREMENTS

2.1 Solder printing

Stencil and squeegee maintenance requirements CEM has to have in place preventive maintenance procedure in place in order to assure good solderability process. In case if stencil aperture is blocked by dust or due insufficient cleaning - this might cause poor solder quantity on the pad – as result insufficient soldering. All stencils, squeegees and support blocks have to have traceability; the best practice would be to have barcode and scanning system with software Poka-Yoke for every step of maintenance and prevention of usage uncleaned stencil. CEM has to have in place liquid stencil cleaning machine offline and do cleaning after every removal from the line or in case stencil was used on the line longer than 8 hours or soldering defect detected on the line - related to blocked stencil aperture. Liquid used for cleaning of stencil and squeegee has to be approved by stencil manufacturer and CEM to avoid aggressive chemicals affect stencil coating or PCB.



Stencil and squeegee storage requirements. Stencil and squeegee after cleaning must be stored on the vertical position in order to avoid bending or damage of apertures and in closed shelves to avoid dust contamination. Traceability has to be assured. Transportation of stencil between maintenance area and production by hand carry is prohibited; special trolley needs to be used. Stencil has to have **shelf life** expiration time based on stencil aperture type (laser cut, etching or electroforming process) and stencil coating type. For squeegee shelf life normally should be defined by manufacturer. Shelf life also needs to be controlled by software Poka-Yoke. CEM has to put in place visual criteria when stencil and squeegee must be scrapped due to damages or over usage.

Solder printing machine maintenance Fist level maintenance to be done during WIP, such as: cleaning liquid adding, cleaning paper replacement, checking of vacuum pressure level etc. Second level maintenance for solder printing machine has to be maintained at least once per week in order to assure oil level for mechanical parts, in order to check general functionality. Camera/sensor inside SPM has to be calibrated at least yearly and R&R study to be done also on yearly base

2.2 SMT placement

Feeder, Nozzle, placement head, maintenance requirements.

Feeder has to be submitted for offline mechanical maintenance and calibration. Based on q-ty of cycles counted by placement machine, timing or error occurred during production.

FEEDER MAINTENANCE AND CALIBRATION

Nozzle also important part of pick and place as it's reposnsibility to keep component. Nozzles must have unique barcode and be scanned. Nozzle has to be submitted for cleaning and visual optical check, as contaminated nozzle can cause placement errors.

CONTAMINATED NOZZLE

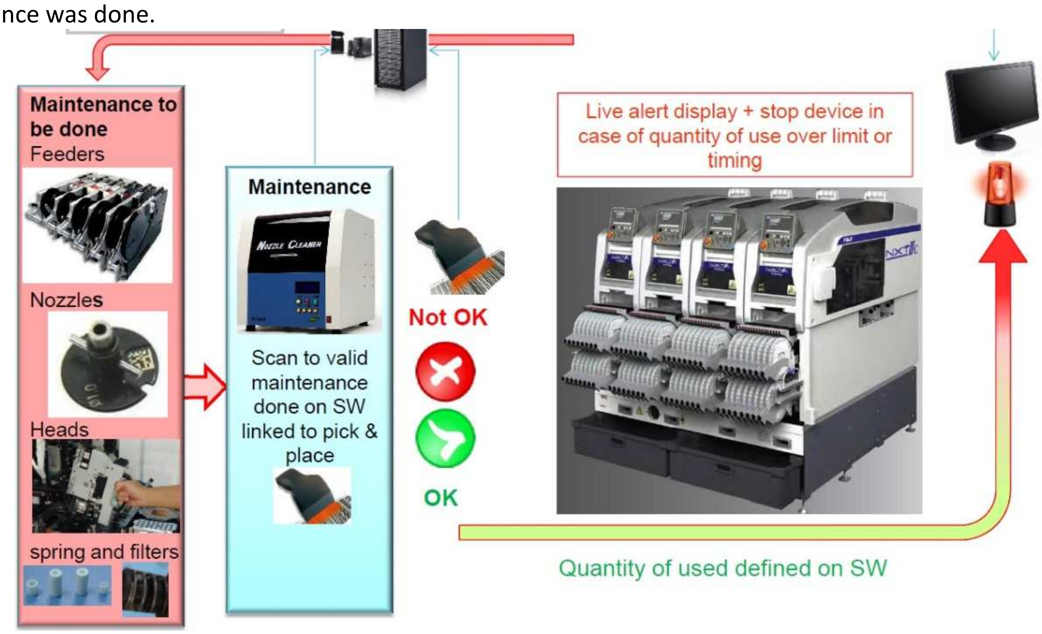
NOZZLE CLEANING MACHINE

NOZZLE INSPECTION

CEM must make sure that maintenance done at least weekly and **placement heads** are checked for vacuum, spring functionanlity and precision. Placement head must have unique barcode and be scanned when installed into machine and at the maintenance. cleaned after every shift or in case of high fall out rate and after cleaning must be inspected.

- defined q-ty of placement cycles has reached, placement machine has to alert and stop until feeder replacement.
- huge failure rate from certain feeder, placement head, nozzle placement machine has to alert and stop until feeder replacement;
- identified visually damage of the feeder parts by operator;
- on weekly base as part of SMT machine weekly maintenance.

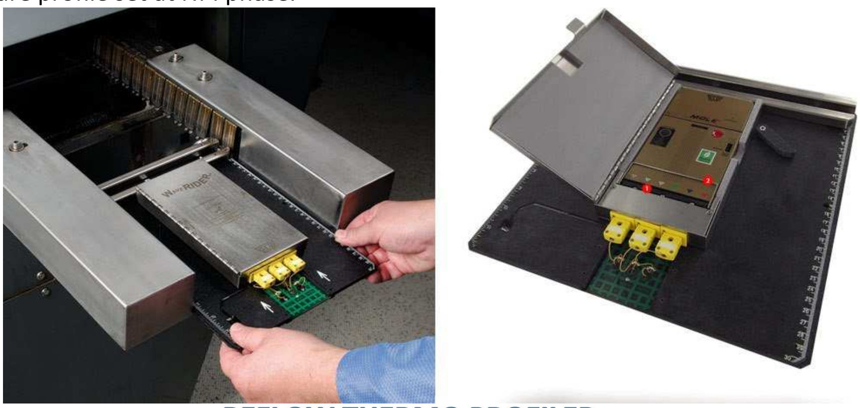
Best practice is to have maintenance software and feeder barcode traceability with unique barcode or number marking. During maintenance to be scanned into software which has to be connected with SMT machine Poke-Yoke in order to assure that maintenance was done.



2.3 Reflow

Reflow involves heating the assembly of board plus components held by solder paste through successively higher temperatures: to remove flux volatiles to start flux activation, to perform the soldering operation, to reflow the solder paste to form a liquid solder volume sufficient to create a sound joint, Finally, the assembly is cooled, to solidify the solder joints and make it possible to handle the board. Due to flux evaporation from solder paste **reflow soldering cleaning maintenance** has to happen at least on weekly base. Also maintenance has to include checking and if necessary repairing of PCBA support belt, motor and other mechanic parts, removing fallen components.

As part of the maintenance CEM must check at least on weekly base exhausting system functionality and no damages on the tube in order to avoid ear contamination by evaporations from machine. After weekly reflow maintenance is done supplier has to measure temperature profile using special equipment in order to make sure that heating zones are applying heat as specified on temperature profile set at NPI phase.



REFLOW THERMO PROFILER

2.4 ICT/FCT.

CEM has to put in place preventive maintenance system for **ICT/FCT** based on timing or operational cycles. For good **maintenance system** CEM has to respect following requirements:

- All ICT fixtures must have traceability with unique barcode or number;
- ICT fixtures must be stored offline on dedicated area, when physical fixture damage or contamination is excluded. Transportation between, production, maintenance area and storage is allowed only on special trolleys;
- CEM has to record all maintenance activity and have standard which defined preventive maintenance frequency and activity to be done. Best practice automatic maintenance software with scanner for recording maintenance result and planning of preventive maintenance connected to the ERP system in order to avoid start of process if maintenance wasn't done. Best practice is also to have cycle counters integrated into tester fixture;
- CEM has to have on place spare parts management for ICT probes, harnesses, cards etc. ICT probes must be checked to avoid bended pins or worn springs. ICT probes to be checked at least once per week as bended probes can cause short with PCBA;
- ICT has to have 2 program modes - debug and production, different user rights to be defined and secured with code;
- ICT platform has to have preventive maintenance at least once per month, at least once per year service maintenance with calibration by equipment supplier or third company.

2.5 Spare parts management

In order to assure good and in time maintenance also to have good reactivity in case of breakage CEM has to have standard for **management of spare parts.** There should be respected following rules:

- Spare parts list to be created with good traceability and defined storage area and conditions;

- Minimal critical level of spare parts to be defined and in case if it's reached – spare parts to be immediately ordered. Spare parts quantity to be defined with respect of usage needs, lead time of delivery, quantity needed per fixture, module, machine etc.

SPARE PARTS STORAGE

ICT PROBES

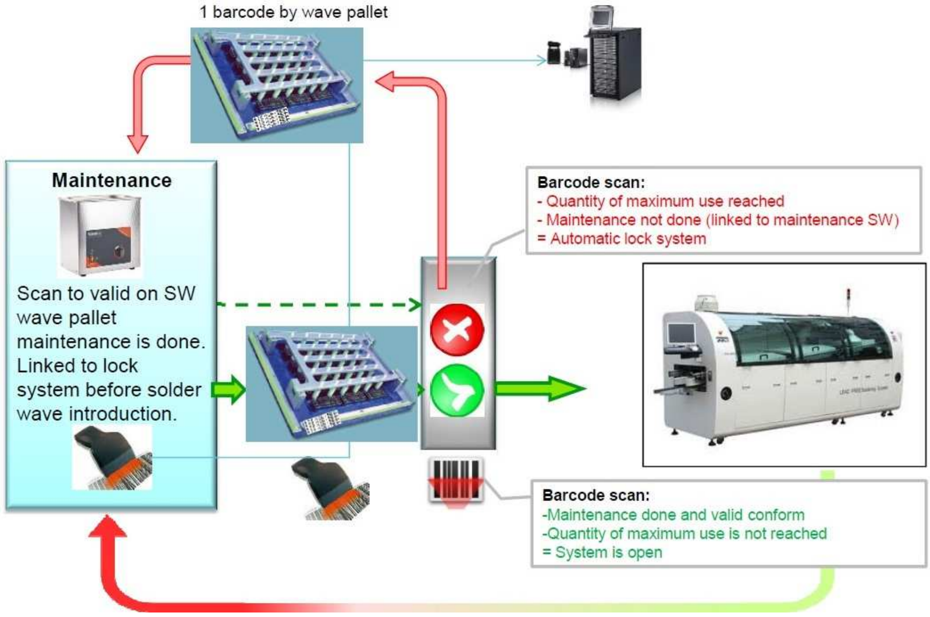
Best practice to have part number for each type of spare part and control it with the software, which notifies automatically in case if minimum level of spare parts reached and send reminder to purchasing department.

2.6 Wave soldering, selective soldering

CEM must develop preventive maintenance standard for each part of the process. Levels of maintenance and frequency must be defined and followed. Wave soldering process contains from main steps which must be maintained:

- Transportation;
- Fluxing;
- Pre-heating;
- Wave soldering & selective soldering;
- Cooling.

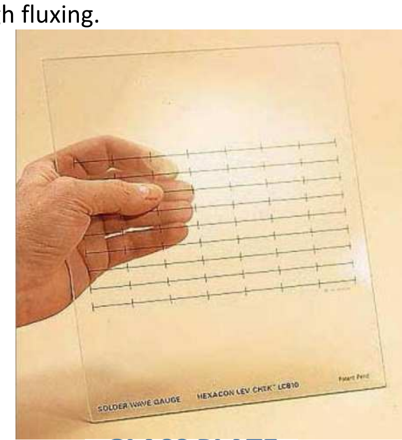
Transportation sufficiency contains on 2 key parameters: PCBA support or carrier (wave pallet) and transportation belt condition (transportation fingers condition). **PCBA wave pallet** is mandatory to protect PCBA against heat on the bottom and fix assembled components with spring on the top. PCBA wave pallet also helping to prevent from solder balls/splashes enter to SMT reflow component area and contamination by flux. CEM must have traceability and preventive maintenance for all wave pallets used for Wittur products. Maintenance should happen min on weekly base and must include cleaning of pallets from flux, dust and solder balls and checking of closing mechanism and fixing springs. Best practice to have software scanning Poke-Yoke of wave pallets.



Transportation belt condition. CEM must minimum on weekly base check conditions of transportation belt fingers. Metal fingers must be cleaned from solder balls and particles and damaged or bent fingers must be replaces due to risk of PCBA drop off into bath

Fluxing maintenance requirements. The're different types of fluxing system, but the most common type of fluxing is nozzle spay. Fluxing method with non-clean flux type recommended. CEM must have preventive maintenance standard of fluxing system which should include cleaning of fluxing nozzle at least weekly. Best practice to have software which control maintenance steps and send reminders if maintenance wasn't done. Shelf life of fluxing nozzle has to be defined and controlled.

CEM must use "glass plate" and run it after every maintenance in order to verify fluxing efficiency (q-ty of flux and coverage. Or also other method of verification is fluxometer method when the flux sensitive paper is placed into carrier frame and run through fluxing.



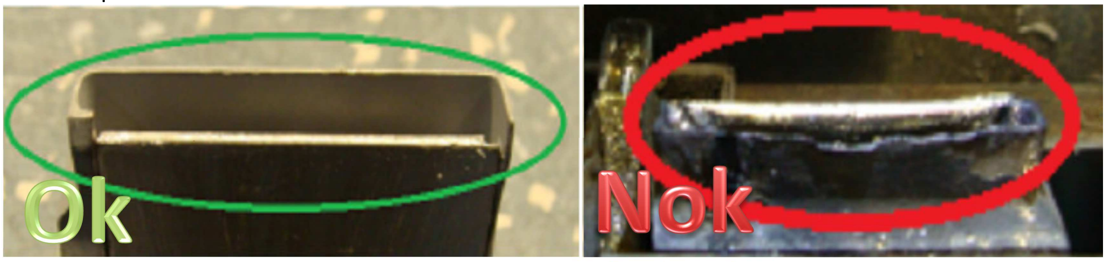
GLASS PLATE

FLUXOMETER PAPER

Pre-heating maintenance requirements. Pre-heating target is evaporation of the flux solvents and decrease temperature shock of components. There're several types of pre-heating system: Convection heating with hot air, radiation heating, combined methods: radiation + convection. Pre-heating has to be cleaned and have preventive maintenance on weekly base. Temperature profile of pre-heating must be measured weekly after each maintenance.

Wave soldering maintenance requirements. Main factors to affect wave solderability would be solder batch alloy and temperature profile. Lead free wave soldering requires high temperatures; therefore key maintenance factor is contamination of solder bath alloy. Solder alloy might be contaminated by solder pot corrosion or oxidation of alloy. In order to assure good quality of alloy CEM must put in place preventive maintenance each 4-6 weeks when CEM must replace alloy and clean the wave bath/pot. Recycling of wave solder alloy is allowed only after removing oxides from it.

Selective soldering maintenance. In case of selective soldering done with special nozzle, the PCBA supplier must define a minimum spare parts in storage >1. It is mandatory to define the maintenance of this nozzle (cleaning and change), it will be adapted according to the quality of soldering and to the line reject in order to replace it. Cleaning of the nozzle inline should happen min. 3 times per shift.



Colling system maintenance requirements. Cooling system must be cleaned minimum on weekly base in order to avoid PCBA contamination by dust. After cleaning CEM must create procedure to verify cooling system functionality. Colling system and conveyor must be covered, risk to damage PCBA must be excluded.

2.7 Depanelization maintenance requirements.

CEM must develop depanelization process preventive maintenance procedure for first (cleaning by operator) and second level maintenance (by maintenance department).

Sawing/blade cut equipment requires:

- Cleaning of equipment and workplace from dust;
- Sharpening of the blade and lifetime management for blade, replacement frequency and spare part management;
- Support plate cleaning;
- Greasing of mechanical parts.

Routing maintenance requirements:

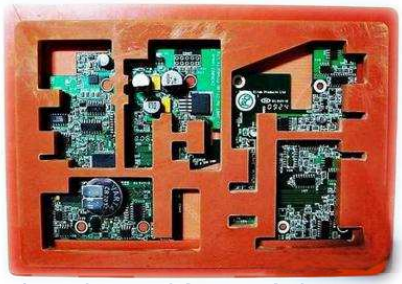
- Routing machine and fixture cleaning;
- Greasing of mechanical parts;
- Routing bit lifetime management , replacement frequency and spare part management;
- Precision sensor annual calibration.

2.8 Coating maintenance requirements

There're several methods to apply coating into PCBA: brush coating, spray coating, dipping and selective coating process. For serial production CEM must use selective coating process. In order to assure robust selective coating process CEM must develop preventive maintenance procedure for coating process and for curing process.

Coating process maintenance requirements. Selective coating maintenance contains from several key requirements: **Cleaning of coating machine, conveyor and carrier** which has to be done every change over or shift change, but no longer than 8 hour. **Nozzle maintenance**. Cleaning of coating nozzle must happen not less than every 8 hours, lifetime of nozzle and spare parts list management must be defined by CEM and strictly followed. Best practice is machine counter of cycles and maintenance software Poke-Yoke. Calibration of sensor for X/Y precision must happen on annual base (yearly).

SELECTIVE COATING NOZZLES

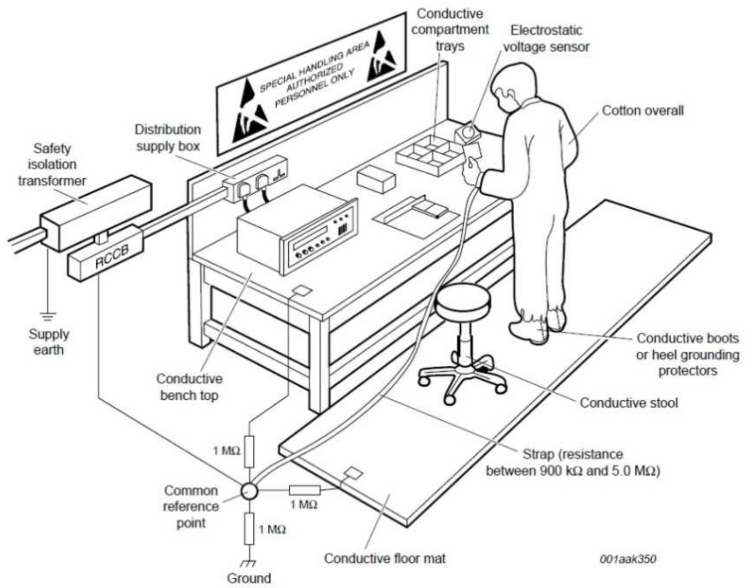


SELECTIVE COATING CARRIER

Curing process maintenance requirements. There're several curing methods**:** humidity curing, heat curing and UV-curing process. For **humidity curing process** CEM must make sure temperature and humidity are under control in the curing cabinet, therefore should be used and calibrated on frequent base thermos sensor with alarming system if out of parameters. CEM must make SPC check for parameters and record them. For **UV-curing process** the key is functionality of curing lamp, CEM must define lifetime of the lamp and replace it based on used cycles. **Heat curing process** requires cleanliness of equipment and validated temperature profile according to coating manufacturer's specification, but for sensitive components rule to be respected: temperature slop not more than 3°C/sec. Preventive maintenance of curing process should happen on weekly base.

3. ESD REQUIREMENTS

ESD requirements have to be respected according to **ANSI/ESD-S-20.20** Protection of Electrical and Electronic Parts, Assemblies and Equipment or **IEC 61340-5-1 + IEC 61340-5-2** Protection of electronic devices from electrostatic phenomena:
CEM must establish ESD team and perform periodic ESD audit not less than every 3 month. ESD Coordinator nominated. ESD teams established. Supplier has established an ESD control program / policy / guideline. The supplier has to prepare an audit plan that addresses each of the requirements of the program and frequency. Visualisation of ESD audit done and next audit date (like calibration lebel) for visual control by production leader.
An ESD test shall be placed before each ESD area access with conditional access, a person without ESD equipment check should not be able to enter to ESD area. ESD result must be valid for time limit. Are ESD Working Areas clearly marked at each entry point. Humidity must be controlled according to the conductivity of the air in order to comply with the level of electrostatic discharge. The floor must have a good conductivity, surface resistivity level must be < 10^4 ohms, the workplace has to be conductive and grounded, chairs have to be grounded by chain.
New operators must be **ESD trained** within 3 months period with documented participation lists. Is there traing matrix for ESD trainig of operators/engineers. Are there regular ESD re-training sessions for experienced operators. Are the training materials sufficient. Is there a practical demonstration part of the training. Is there procedure to control that only ESD trained operators perform the tasks.
Are all **workstations for handling ESD** sensitive devices specially marked. Are all electrical machinery electrically grounded. Are the all workdesks, chears and equipment grounded. Does all ESD Workstations are supplied with connecting points for grounding wrist straps. Does these connecting points have resistance to Ground of 1 M Ohm? Are all work surfaces of ESD Workstation with conductive Surfaces ? (minimum Dissipative? <10^9 Ohm). Are the surface of tables replaced frequently and there're not damages on the surface. Are there Ionizers in use, and aligned correctly (for plastic parts handling). Are ionisers regularly checked for functionality (at least weekly). Are maintenance records of the ionizers available. Does all workstations are numbered, with avaliable recored ESD measurement result. Is resistance to Ground for all work surfaces between 0,5 und 120 M Ohm.



All magazines and, boxes, trays used during production must be conductive and allowed to use them in EPA.The flow of this ionizer should be oriented on the pick-up area of those components. Industry carton boxes not allowed inside EPA, only special ESD carton is allowed. Resistivity level must be > 10^4 to < 10^9 ohms. Do all containers used for ESD sensitive devices have the ESD sign visible on the inner ESD protection. Resistivity level must be > 10^4 to < 10^9 ohms. Are all ESD sensitive device in closed ESD shielding boxes/bags, if used out of ESD protected area? Resistivity level must be < 10^4 ohms. Are other packaging used on the line for mechanical parts ESD complaint? esistivity level must be > 10^4 to < 10^9 ohms.

Designation	Resistance (ohms)	Exponent Format
	10	10^1
Shielding <10^3	100	10^2
Conductive <10^4	1,000	10^3
	10,000	10^4
	100,000	10^5
Dissipative	1,000,000	10^6
$\geq 10^4$ to <10^{11}	10,000,000	10^7
	100,000,000	10^8
	1,000,000,000	10^9
	10,000,000,000	10^{10}
Insulative $\geq 10^{11}$	100,000,000,000	10^{11}

ANSI/ESD S541

Figure 3: Resistance Classifications

Table 2. Summary of Protective Properties

Protection	Property
Low Charging (antistatic)	Materials that have reduced amounts of charge accumulation as compared with standard packaging materials.
Dissipative or Conductive Resistance	Provides an electrical path for charge to dissipate from the package.
Discharge Shielding	Protects packaged items from the effects of static discharge that are external to the package.

Is each person, including engineers, managers and visitors in the EPA ESD compliant – wearing ESD garment, ESD shoes, and checked? Are ESD personnel ESD coats/jackets frequetnly measured for resisitivity level (after each cleaning)? Are all testers for wrist straps, shoes and other static sensors operational and in use? Is their function checked on a regular (monthly) base.

4. SMT: FRONT END REQUIREMENTS

4.1 Start of production

CEM must develop startup check list. Check list must include checking of validity of process parameters, material, documentation availability, operator trainings and meet quality acceptance criteria. Startup check list must include: barcode/labelling/engraving requirements; PCB cleanliness verification; solder print parameters and tools (best practice with software Poke-Yoke), SMT parameter sheet, reflow correct temperature profile per list, AOI startup with “golden board” and first item validation. Without checking all the points from check list by quality leader of SMT area - production is not allowed to start. Startup checklist must be validated every production change or after line stop longer than 30 minutes. All records must be stored and archived by CEM and represented to Wittur upon request. First serial number of running production must be recorded into sheet.

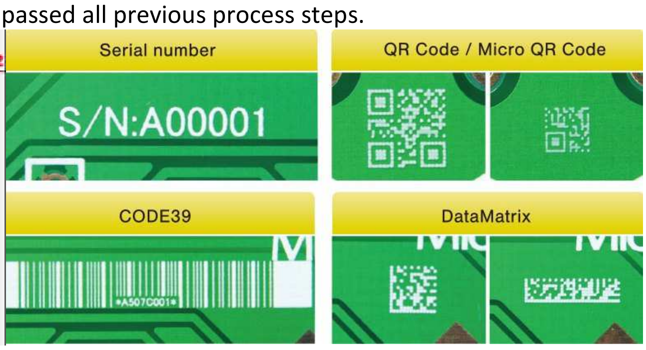
4.2 Trough process traceability

The CEM has to follow product traceability requirements of Wittur. CEM has to establish production traceability system which must cover every process step with software and scanner reading since birth of the board to shipment In case of panel PCBA each specific PCB must have own traceability (label or laser 2D code) and board history.

Traceability system has to cover process steps and show on board history every step and timing: Engraving/Labelling=> Solder printing=> SPI (in case if inline) => SMT placement=> Reflow=> AOI=> Rework after AOI=> THT=> Wave=> Touch up after wave=> ICT=> Rework after ICT=> Coating => FVT=> Rework after FVT=> Packing. Upon request supplier has to show board history records for given serial number. Processing time of PCBA should not exceed 7 days from PCBA birth time (open time of PCB) in order to avoid humidity absorption. Best practice to have processing time control by ERP.

Serial number of PCBA has to be connected to component traceability in order to provide **batch info** to the customer upon request. Any change to the label content has to be accepted by Wittur. The position of the label must be defined in the mechanical drawings referenced. Traceability Label must be readable in Wittur line in order to have traceability of the PCBA until final customer. CEM must scan the final individual label for all the PCBA of the final packaging and the box packaging label printing is done only if all the units are conform and passed all previous process steps.

Principal Operation	Condition1	Operation1	Condition2	Operation2
$START	$PASS	ININ		
ININ	$PASS	PREP	EF	EREP
PREP	$PASS	smdTop	EF	EREP
smdTop	$PASS	TOPDIP	EF	EREP
TOPDIP	$PASS	ODD	EF	EREP
ODD	$PASS	WAVE	EF	EREP
WAVE	$PASS	ATE	EF	EREP
ATE	$PASS	FT	EF	EREP
FT	$PASS	PACK	EF	EREP
PACK	$PASS	$DONE	CUSTA	CUSTA



4.3 Loading/bare PCB handling

All PCBs must be stored at production area on dedicated trolleys which prevents damage of packaging. PCBs must be packed into original packing or repacked vacuum packing, packages with loose of vacuum are not allowed to use without prior backing according to requirements listed on Chapter 6. When operator open the packaging first of all humidity indicator must be checked, if indicator is not blue – PCBs must be baked according to Chapter 6 requirements and repacked. Open packages must be immediately load into the SMT line, handling of PCBs with bare hands not allowed due to risk of organic contamination, which might influence solderability, so all operators must wear ESD gloves, which must be clean.

Before solder printing process CEM must install **vacuum cleaning system** with ionization or alternative cleaning system to avoid contamination. All conveyors prior reflow oven must be covered to avoid dust contamination which might cause soldering defects.

ESD GLOVES

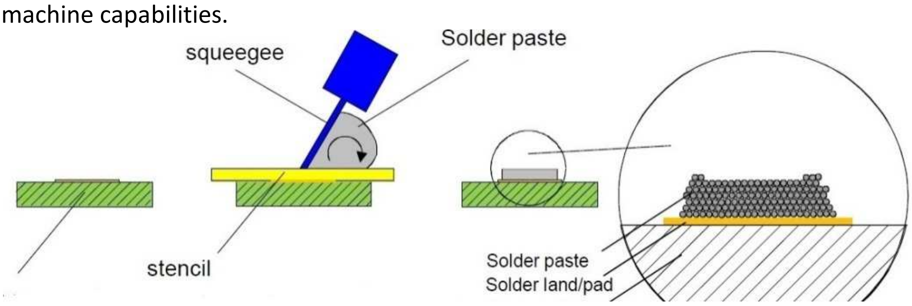
SMT VACUUM CLEANER

SMT COVERED CONVEYOR

4.4 Solder print

The process of depositing solder paste into PCB via stencil apertures. Key factors on good solder printing process:

- Applicability of stencil design to PCB design(apertures design and cut method, thickness of stencil);
- Stencil condition (cleanliness and no damages) ;
- Solder paste condition;
- Support of PCB;
- Squeegee condition(cleanliness and no damages);
- Atmosphere inside printing machine: temperature & humidity;
- Solder print parameters: print speed, squeegee pressure, squeegee angle, separation speed, bottom stencil cleaning frequency and solution;
- Solder printing machine capabilities.



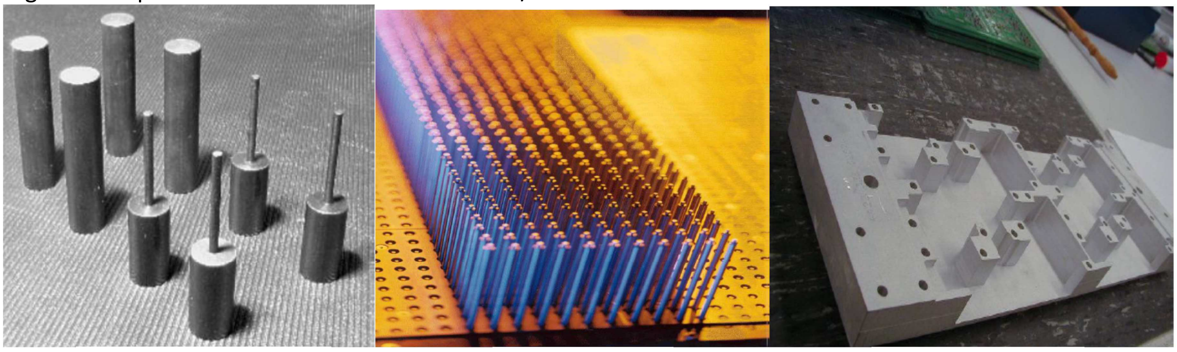
Stencil design must assure sufficient soldering on the pad of PCB. During stencil designing period CEM must to follow PCB pad designed based on Gerber files, also shape of apertures and stencil type must be taken into account. Stencil design must be in accordance to **IPC-7525** Stencil Design Guidelines. Stencil thickness must be 130 – 150 µ.

Stencil/squeegee condition requirements listed on Chapter 7. CEM must put in place software/scanning Poka-Yoke to avoid missing maintenance. Manual cleaning is forbidden. Sharp spatulas for solder removal from stencil are forbidden. Hand carry stencil between production and maintenance is forbidden, special trolleys must be used.

Solder paste condition must be in accordance to the requirements described in Chapter 6. CEM must have software Poke-Yoke which with scanning which will prevent from wrong solder paste usage. Solder paste might be applied manually or best practice to use automatic solder paste dispenser

MANUAL SOLDER PASTE INSERTION AUTOMATIC SOLDER PASTE DISPENSION

Support of PCB CEM must implement sufficient PCB support in order to have good stencil alignment for good paste deposition and also in order to prevent bending of PCB which might lead to crack on chip components or BGA balls for double sided PCBAs. For the paste deposit process which uses vacuum jig CEM must make sure that the support plate is qualified according to product and has unique number, especially for double side PCB. It is mandatory to make stress measurements for double sided PCB in order to be sure that there is no risk of component damage on the bottom side during the printing of the top side. The value must be <1000um/m.



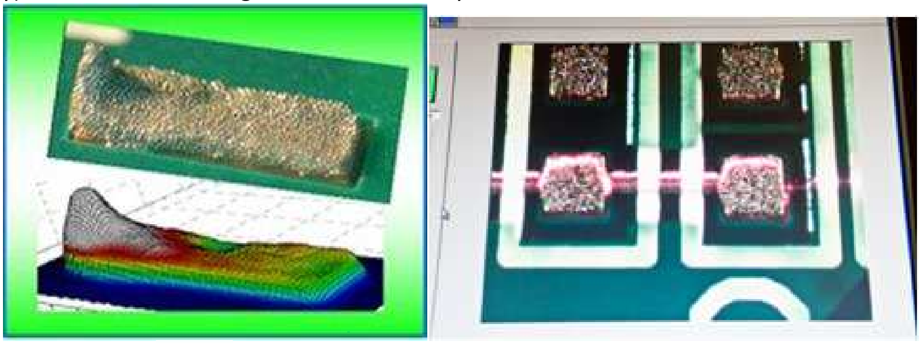
MAGNETIC PINS AUTOFLEX VACUUM JIG

Atmosphere inside printing machine: For optimal results, the temperature and humidity of the areas in which printing, populating, and reflow take place should be maintained at a stable level. A temperature of 23° - 25°C at 45% ± 5% relative humidity is optimal.

Solder print parameters: CEM must develop parameter sheet for solder printing process by production trial. Parameter sheet must include: print speed, squeegee pressure, squeegee angle, separation speed, bottom stencil cleaning frequency and solution etc. Machine program for every specific product must be developed and every start of production CEM must make sure that correct program was chosen comparing program parameters with documented parameter sheet defined on NPI phase. **Solder printing machine capabilities** must be verified on annual base (yearly) and evidences must be recorded.

4.5 Solder paste inspection

The volume of paste printed should be measured either by 2D + thickness profiling or in 3D techniques at least at each start up and end of production as well as before refeeding of solder paste, also SPC during process is requirement. If a PCBA is detect as out of the specification and confirm. It must be scrapped, if not it will be stop at next process step (placement) or lift up**.** Annual (yearly) calibration and Gauge R&R is mandatory.



3D 2D

Parameters to be controlled at SPI:
- Height (between 90 and 180 µ);
- Volume;
- Shape.

Data of inspection must be recorded and stored for warranty period and provided upon request. Process step of SPI must be recorded to board history and linked with AOI in order to cross check suspected defects. FPY of SPI must be monitored and line stop reaction plan to be developed if FPY goes below 90%.

4.6 Surface mount technology component placement

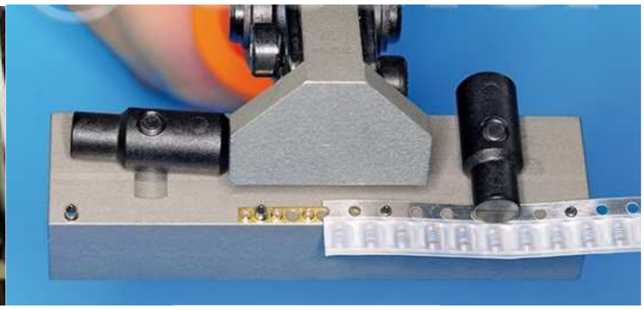
Surface-mount technology (**SMT**) is a method for producing electronic circuits in which the components are mounted or placed directly onto the surface of printed circuit boards (PCBs). An electronic device so made is called a **surface-mount device** (**SMD**). What was traditionally called “pick and place” has two primary functions to satisfy in order to ensure a high quality placement process: it must select and pick-up the correct component from a magazine or feeder; it must place this component on the correct footprints, in the correct orientation, and with the required accuracy.

Quality of good placement depends on key parameters:
- Condition of component;

- Feeder condition;
- Correct type of feeder depending on packaging;
- Placement head and nozzle condition;
- Correct type of nozzle depending on component body;
- PCB support and conveyor with;
- Correct SMT machine set up;
- SMT machine self-detectability.

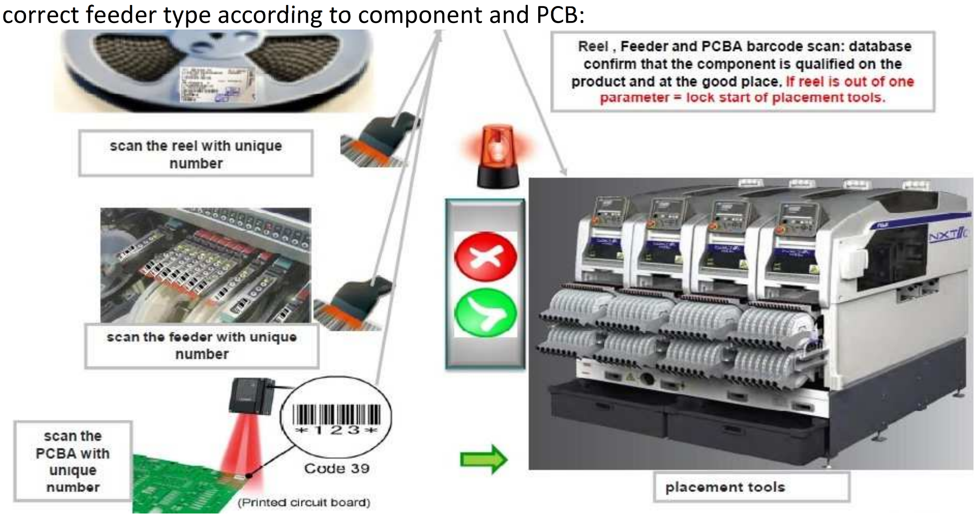
Condition of component depends on respecting requirements described in Chapter 6. Carton reels and boxes not allowed at SMT area. Before usage component must be scanned to the feeder and to the SMT machine. Program should stop producing if wrong type of component used or MSL open time has expired for SMD. CEM must develop parameter sheet where must be stated at which module and slot must be used which component – machine program must be aligned with parameter sheet, which has to be verified every start of production. Splicing not recommended due to the risk of component mixing, missing component due to wrong splicing. For SMT splicing is allowed only with use of special gauge. For ICs and leaded components splicing is forbidden.

SCANNING OF REEL LABEL



SPLICING GAUGE

Feeder condition feeder must be maintained and calibrated as described on Chapter 7. CEM must have software Poke-Yoke which will detect correct feeder type according to component and PCB:



Correct type of feeder depending on packaging. SMT using different type of packaging therefore choosing the correct feeder type is the key parameter. Components typically delivered on tape and reel packaging, but some like connectors might be delivered on tubes or BGAs on trays. For every specific purpose CEM must use correct feeding system and settings. For different feeder types – different SMT modules to be used.

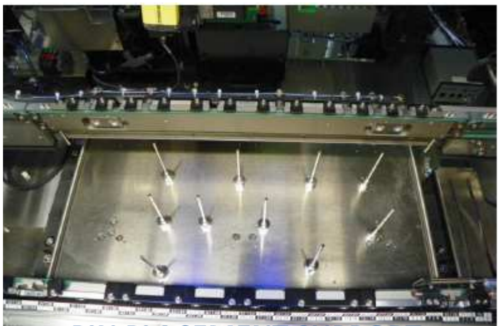

TAPE AND REEL FEEDER


TRAY FEEDER


TUBE FEEDER

Placement head and nozzle condition.
Placement heads and nozzles important part of pick and place process as they responsible for component pick and place accuracy. There're 2 types of placement heads with multiple component placement possibility and head with 1 component placement possibility. Both have advanatages and disadvantages. CEM must follow preventive manintenance requirements and traceability listed on Chapter 7.
Correct type of nozzle depending on component body. CEM must make sure that correct type of nozzle is used for each specific type of component body. Special focus must be on MLCC, LED nozzle as those might be heated by wrong nozzle shape and must be: MLCC, chip resistor - square shape, LED - oval etc.

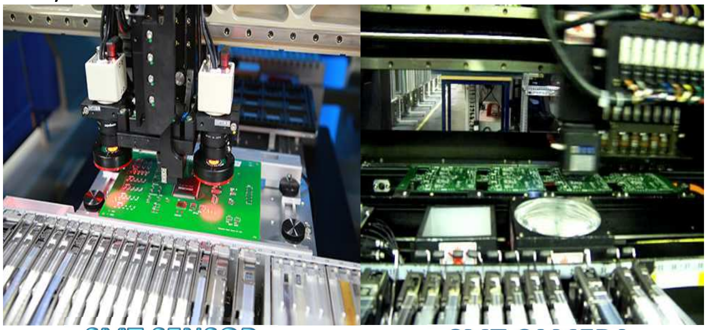
PCB support and conveyor with. In order to avoid falling down PCBA or bending of PCBA during placement CEM must make sure that sufficient PCBA support during placement provided and conveyor with optimally set up. For double sided PCBAs CEM must make sure that support don't damage bottom side components, therefore rubber type of support is strongly recommended.


PIN PLACEMENT SUPPORT


RUBBER PLACEMENT SUPPORT

Correct SMT machine set up. CEM must develop parameter sheet with description of on which module which component must be used, pick and place parameters for each component must be in according to component body type and size. As minimum requirement CEM must have parameters per type of component. Key parameter for sensitive components like

MLCC or LED is placement force which must be not more than 3N as higher force will lead to component damage. Each start up CEM must make sure that correct program is in use and in case of any mistake machine will alarm about error.
SMT machine self-detectability is important part of pick and place accuracy. Most of modern pick and place machines have cameras or sensors which should detect component presence when pick and placement position accuracy, most developed modules can also detect coplanarity of lead on ICs. CEM must make annual calibration of camera/sensor and R&R study.



SMT SENSOR SMT CAMERA

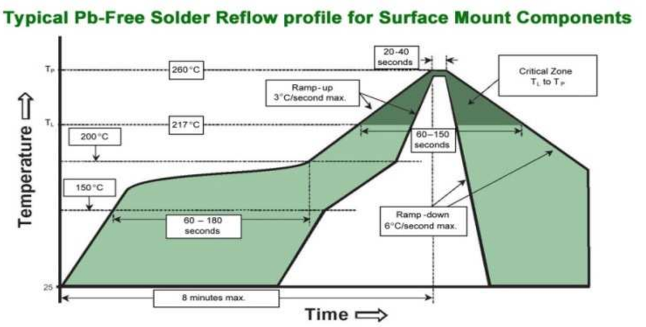
4.7 Reflow soldering process requirements

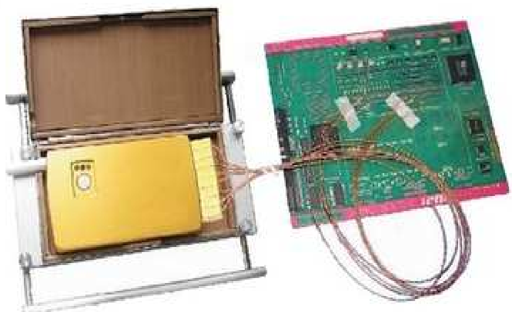
The process can be carried out in a several ways, but volume production is normally performed on a continuous belt oven: Frequent reference is made to the 'zones' the section of the machine in which a particular process takes place. Not depending on technology used (convective, infrared) CEM must make sure that maintenance requirements are followed as by Chapter 7.
Key parameters of reflow soldering machine:

- Product specific temperature profile;
- PCBA transport system;
- PCBA traceability and correct profile Poka-Yoke

Product specific temperature profile At NPI phase CEM must develop product specific temperature profile in accordance to solder paste wettability requirements and sensitive components requirements. Based on this product specific reflow profile program must be developed and approved by Wittur on project phase. All changes on the program must be communicated to Wittur and allowed only after Wittur approval. Solder paste optimal reflow profile usually included into solder paste datasheet and contain info about: pre-heating time and temperature, heating speed, dwell time, liquidus time, min and max peak temperature , cooling time.
In order to respect **reflow sensitive components** as BGA, MLCC, chip resistor, electrolytic capacitor, tantalum capacitor, inductors with ceramic cover etc CEM must make that heating/cooling slop is not higher than 3 C/sec, as higher temperature slop might lead to cracks due to overheat. Peak temperature should not exceed 250°C.

Profile Classification per IPC/JEDEC J-STD-020C Pb-Free Small Body Assembly

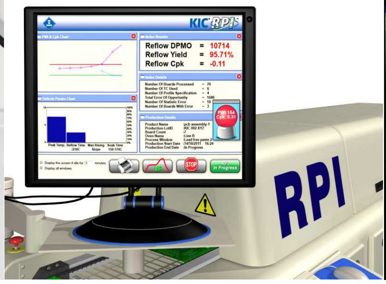


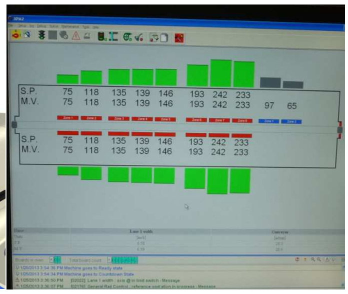


Profile Classification per IPC/JEDEC J-STD-020C Pb-Free Small Body Assembly

Every start up CEM must make sure that correct reflow profile is in use. Best practice is to have software scanning Poka-Yoke in order to make sure that correct program was chosen and error of choosing wrong program is prevented.
PCB transport system. CEM must make sure that during reflow PCBA has got sufficient transport system and board support. Conveyor with has to be optimal to avoid drop of PCBA but also prevent bending. CEM also has to make sure that no risk to damage bottom side components during second reflow (for double sided PCBAs).

Not less important is the PCBA scanning into MES. CEM must make sure that automatic scanning is installed and time is recorded into board history. Reverse traceability of used temperature profile is also mandatory. Best practice is to have SW Poka-Yoke with verification PCBA PN scanning and alarming if wrong program was chosen or wrong side (for double-sided PCBAs). Minimum requirement inline machine temperature zones control and alarming if tolerance has exceeded.



PCBA AUTOMATIC SCANNER SW PROGRAM POKA-YOKE TEMP. ZONES CONTROL

4.8 Automatic optical inspection

AOI process step depends on key factors: AOI machine set up, AOI machine capability and operator training. AOI is the key detection process during SMT. CEM must set up AOI acceptance criteria according IPC A-610 class 2.

AOI must be 100% and inline. **FPY** at AOI must be more than 95% as lower FPY increases number of false failures, which lead to high number of operator decisions to be made and higher risk of escape. Red box for non-conform parts must be identified, all failed pasts must have fail result on board history is they're not false failures. Quality engineering team must analyze AOI data in order to optimize process parameters. **FPY** monitoring is mandatory and line stop criteria must be defined if FPY goes below 90%. Recommended rule 3-5-10 (same defect 3 times one by one, 5 times in 1 hour and 10 per shift).

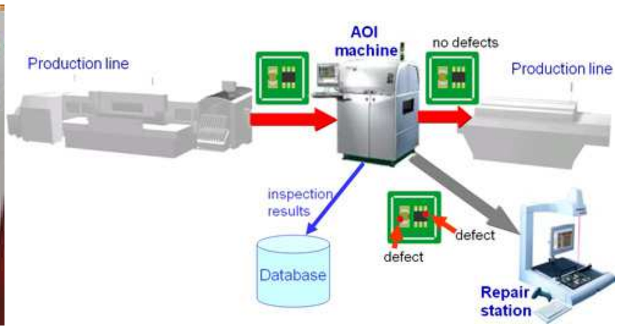
All **operators must be trained according IPC a-610** and training must be refreshed yearly. CEM must develop defect catalogue and train the operators. Line stop criteria must be define and clearly described for operators.

Each **AOI startup** must be assured with "golden board" - defective sample for each reference which supplier must run through AOI in order to verify that detection works properly. Golder board must be specially marked and stored separately from serial parts in order to avoid mixing with serial parts. **AOI must be calibrated** and validated with R&R on yearly base. **5S** must be respected, workplace must be ergonomic and equipped with all necessary tools. Operator allowed handle PCBA only with gloves in order to avoid contamination.

Process input output must be automatic, best practice to have automatic conveyor split into 2 magazines Pass and Fail, operator shouldn't check Pass boards and Fail only if necessary to do inspection with magnification due to risk of mechanic damage with additional handling. If no automatic conveyor – red box must be available at the workplace.

AOI process step must be recorded into board history and linked by SW MES with further process steps to be allowed only after rework/debug record.

AOI AUTOMATIC CONVEYOR LOADING



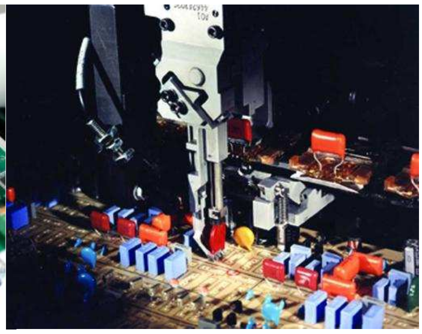
AOI REWORK FLOW

5. THT:BACK END REQUIREMENTS

5.1 Trough hole technology requirements

THT usually is used for larger components which are not applicable to place by SMT like connectors, big electrolytic capacitors, transformers, switches etc. THT refers to the mounting scheme used for electronic components that involves the use of leads on the components that are inserted into holes drilled in printed circuit boards (PCB) and soldered to pads on the opposite side by wave soldering or selective soldering. Components might be inserted either by manual assembly (hand placement) or by the use of automated insertion mount machines.

MANUAL THT PLACEMENT



AUTOMATIC THT PLACEMENT

Key requirements of THT process:

- Assembly standard existing and respected;
- Operators are trained for operation, skill matrix available;
- ESD protection, ionization;
- Full component traceability and FiFo;
- One piece flow to avoid mixing;
- Assembling Poka-Yoke to avoid wrong insertion;
- 5S respected;
- Correct PCBA support during insertion;

Assembly standard existing and respected. CEM must develop assembly standard and enforce it by N+1 of operator. Operator must be trained for defined operation, multi skills are beneficial. Training matrix must be available on the line. **ESD protection** requirements according to Chapter 8. CEM must install ionization close to plastic components in order to avoid ESD. **Full component traceability** and **FiFo**. CEM must have full component traceability: PN, MPN, batch number, date code, delivery date etc. Best practice to scan every new package into ERP system in order to have back traceability linked to PCBA serial number. Mixing of different batches/date codes is not allowed. Components must be handled by FiFo. **One piece flow** must be respected by CEM in order to avoid mixing different components and avoid placement errors. As support of assembling error prevention CEM requested to develop assembling **Poke-Yoke** jig. **5S** is the key requirement for operations which involves manual operations. **PCBA** must get sufficient **support on bottom side** in order to avoid bending - which causes mechanical stress on PCBA which might cause micro-cracks. Conveyor with also must be optimized for PCBA with.

5.2 Wave soldering, selective soldering

Wave soldering process is the process when leaded components get soldering moving through solder alloy bath. Wave soldering process contains from main steps:

- Transportation;

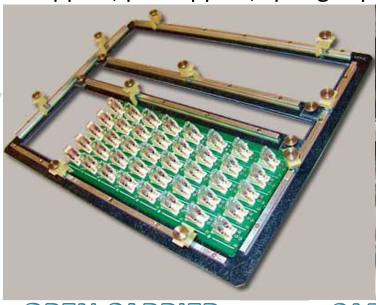
- Fluxing;
- Pre-heating;
- Soldering;
- Cooling.

Transportation process requirements. Transportation process depends on factors:

- Carrier/pallet or carrier-less transportation method;
- Conveyor type: chain or finger type;
- Transport angle.

The're 2 different methods of printed board transport in soldering machines with aid of **carriers/pallets** or without aid - **carrier-less**. Carriers might be open and closed type. Closed carrier pallet is preferable solution as it can protect SMT components from head and fluxing and also fix position of THT components with springs, additional benefit of carrier – good PCBA support in order to avoid bending. For carrier-less method CEM must develop sufficient support of PCBA against warpage. There're different types of support: wire support, pin support, spring support.

CLOSED WAVE PALLET



OPEN CARRIER

CARRIER-LESS WIRE SUPPORT

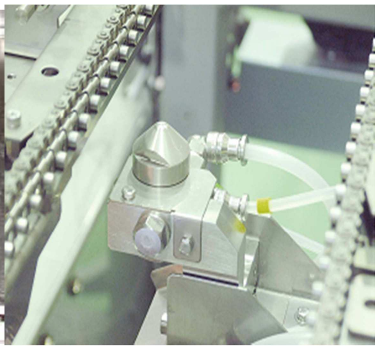
Carriers/pallets must be maintained according to requirements on Chapter 7. **Conveyor transport** chain or finger type. Important is to have preventive maintenance in order to assure risk of bending, damaging of PCBA is excluded due to fingers and risk to drop off PCBA is also excluded. Critical parameter is the **angle of conveyor** at loading and unloading - no risk to drop off or damage PCBA, at wave - angle is optimal for solderability.

Fluxing process requirements. Target of fluxing is to solve several requirements: improvement of metal surfaces wetting, cleaning of metal surfaces from oxides, reduces surface tension of solder, protects solder pad (solder joint) from oxidation during soldering. CEM must use non-clean not aggressive type of flux for lead-free technology. J-STD-004A bring the table how to choose the correct flux type. Here're different **fluxing systems:** wave fluxing, brush fluxing, drum spray fluxing, foam fluxing, and nozzle spray fluxing. Preferable fluxing method – spay nozzle fluxing.

Flux changes in composition when in contact with the atmosphere. These changes take two forms:

- Loss of solvents, which increases both the solids content of the flux and its viscosity. These parameters are normally deduced from measurement of the specific gravity ('SG') of the flux, and solvent loss can be compensated for by adding replacement solvent ('thinners'). However, with rosin-based fluxes used in foam fluxers, the changes are very rapid, so constant monitoring and replenishment is required.
- Oxidation of the flux, which reduces its fluxing effectiveness. Nothing can be done about this, other than replacing it by new flux. A typical recommendation is that flux exposed to the atmosphere should be replaced after 40–50 hours of use. CEM must control shelf life of flux.

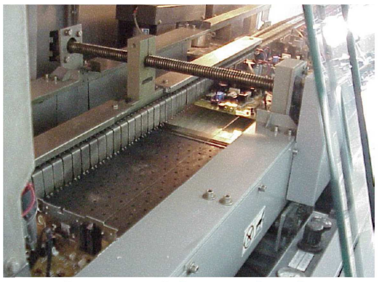
FOAM FLUXING

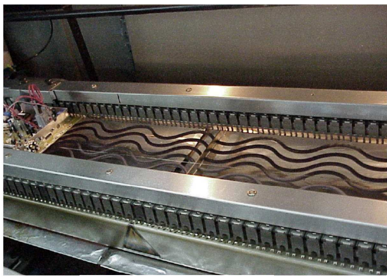


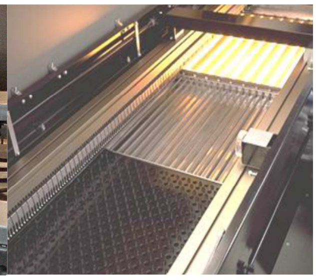
NOZZLE SPRAY FLUXING

CEM must have preventive maintenance program and system to verify fluxing functionality as described on Chapter 7.

Pre-heating process requirements. Target of pre-heating process: evaporation of the flux solvents (proper flux adhesion), acceleration of the activity of the flux, to lower the thermal shock of components and printed board to be soldered and avoid too much heat withdrawal from the solderbath, increase soldering speed by bringing surfaces closer to required soldering temperature. Several types of pre-heating convection heating with hot air; radiation heating: heating coils, quartz lamps, hot plates; combined methods radiation and convection.


HOT AIR CONVECTION


RADIATION


COMBINED METHOD

CEM must follow maintenance requirements as described on Cheater 7.

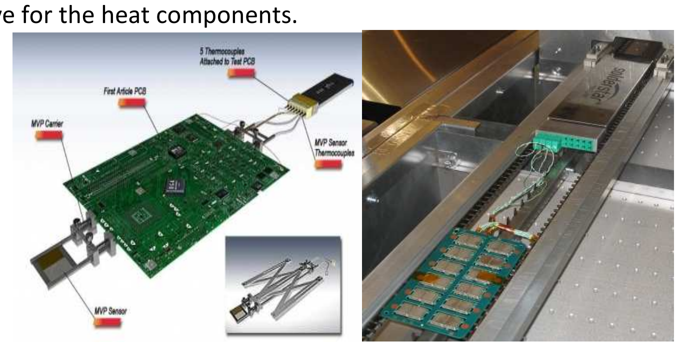
The goals **of the process profile in the preheat zone** are numerous**:**

- The heating should be gradual and not exceed 3°C per second. Higher heating rates have the potential of increasing the temperature differential across the board, which can increase the amount of warp and twist (and possibly local delamination) in the board as it exceeds the glass transition temperature;
- The heating should be somewhat uniform on the top and bottom side of the board. During the preheat, the reflow solder side of the board must by necessity be kept a little lower than the wave solder side;
- The heating should be in line with the needs of the fluxes. No clean fluxes may demand a little more preheat. Excessive preheat should not be used.
- The temperature differential between preheat and wave solder peak should not be hight. Of course, less is better. Some of the largest benefits in reduction of thermal stresses can be achieved in the preheat zone.
- Some of the results seen with higher thermal stresses are: increased solder fillet sizes, increased solder bridging, increased amounts of solder balls, increased amounts of solder peaking, increased failures of thermal stress sensitive parts such as ceramic chip capacitors

Mistakes to be avoided

- Preheating from one side only
- Counting on controllers which measure heater temperature
- Establishing the profile under ideal conditions
- Using the same profile for all assemblies
- Large gaps between preheat and solder wave which result in large temperature drops.

Attention to the profiles as seen in the preheat zone, and then monitoring the important parameters on a regular basis (weekly). Uniformity of heating is improving as more and more zones are being added to the wave solder. The profile must be established by CEM under ideal one board conditions. Thermo- sensors thermocouples must be please on the most critical areas close to the most sensitive for the heat components.



TEMPERATURE PROFILE MEASUREMENT

Wave soldering process requirements. Wave soldering is named because of use of waves of molten solder alloy. The process uses the tank so called solder bath to hold a quantity of molten solder. Components are inserted into PCB and pass across pumped wave or waterfall of solder. Used for both SMT and THT. If wave soldering is used for SMT then the parts must be glued to the board prior to molten solder wave to prevent them from floating off.

There're 2 types of wave soldering: single and double wave. Most solder wave equipment now has dual zones. The first zone is more agitated to increase the wetting action of the solder, and the second zone is usually a smoothing zone to optimize the shape of the fillet and reduce bridging.

SINGLE WAVE

DOUBLE WAVE

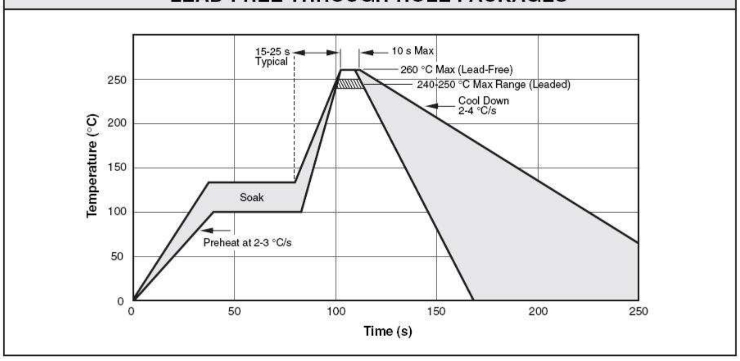
Temperature profile key parameters:

- **The peak temperature** of the wave solder should be kept to a minimum. Some tests have shown that reduction of thermal stress between preheat and the wave solder is better accomplished by reduction of the peak wave temperature (as opposed to increasing the preheat temperature). Wave solder peak temperatures of 250°C +/-10 °C are common, however, this requires larger preheat temperatures;
- **Contact time time in the wave** should be kept to a minimum. Typical times in the total solder wave are 5 to 8 seconds. Time in the wave exceeding 10 seconds can begin to have detrimental effects on the solderability of the parts and the board, result in breakdown of the fluxes making cleaning more difficult, and greatly magnify the stresses applied to the board.
- **Conveyor speed** is critical for soldering due to solder flag defects or not uniform amount of soldering and also bigger temperature shock for chip ceramic components like MLCC. Temperature slop should be not more than 3°c per sec for temperature shock of chip components.
- **Temperature between the waves** should be maintained above the liquidus points. Some equipment have gaps between the two waves, and with improper venting and room environments, the solder temperature can go below the solidus temperature, and then be subjected to another thermal stress in the second wave.
- **Temperature of the reflow solder on the opposite side** should not exceed 150°C. If the temperature of the top side solder exceeds 150°C, then excessive grain growth in the solder can occur and weaken the fatigue strength of the solder joint.
- **PCBA position to the wave** to the PCBA is the key parameter in order to get good contact surface with components/pads. Angle of conveyor to solder wave should be 7-10°. Distance bottom printed board - nozzle chip wave: 6 ± 1 mm. Distance bottom printed board - main wave sump: 7 ± 1 mm. Parallelism of board to waves within 0.2 mm. checking of parallelism of the wave with glass plate must happen every start up.

Mistakes to be avoided

- Increasing wave temperature to improve solderability
- Increasing time in wave to improve solderability
- Increasing temperatures to reduce solder balls, bridges, skips, etc.
- Establishing wave profiles once, and not monitoring frequently
- Adding forced air cooling directly after wave solder
- Temperature profile to be measured during weekly maintenance as described on Chapter 7.

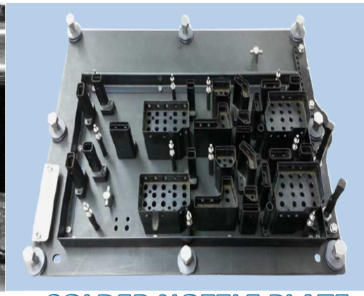
TYPICAL WAVE SOLDER PROFILE FOR LEADED AND LEAD-FREE THROUGH-HOLE PACKAGES



In **selective soldering process**, specific through-hole devices on a PCB are “selectively” soldered from below. Selective soldering uses robotics to individually move each board over stationary soldering nozzles and tooling. Selective soldering also uses two different techniques— drag or dip soldering—to solder individual sites or components. Drag soldering is accomplished on a small single nozzle wave. Drag soldering can reach very tight areas, individual spots or leads. The PCBA can be handled at different speeds and angles to optimize the soldering results. In contrast, dip soldering involves dipping the entire PCBA onto a custom tooled nozzle plate, soldering all joints in one operation. Although multiple connections are soldered at once, individual tooling plates for each type of PCB are required.

DRAG SOLDERING

DIP SOLDERING



SOLDER NOZZLE PLATE

Key parameters of selective soldering are:

- **Wave height monitoring.** These typically should be able to maintain solder wave heights to within +/- 0.005. They are particularly effective when working with the smallest wave nozzle sizes, less than 6mm down to 1.5mm. CEM must have monitoring methodology of wave height relative to a PCBA reference, programmable intervals during extended production runs, and closed-loop feedback is then used to automatically adjust solder pump speeds.
- **Temperature of alloy** Soldering through hole terminations at 280°C & 320°C per alloy. The risk of heat chock for other component is minimized therefore timing is very short.
- **Nozzle** maintenance conditions must be respected as per Chapter 7 requirements.
- **PCBA support** during selective soldering is not less important as at reflow or wave. Supplier can use special gauges to support PCBA. Conveyor with must be under control to avoid warpage or bent condition.

Hot Air Debriding. The concept of using hot air blowing on the solder joints immediately after the wave to minimize bridging and solder fillet size, is an excellent idea. The air temperature immediately at the exit of the orifice is typically set near 275°C, and the impinging air on the part/board assembly is typically less than 230°C. **Cooling Zone.** Thermal shock stresses applied by cooling after the wave can be more detrimental than that applied by the heat stress of the wave itself. The process engineer might have a tendency to supply forced air cooling directly after the wave solder and hot air debriding areas. One is that the board needs to be cooled for people to handle it. Another reason is that the solder needs to be cooled rapidly to establish fine grain solder fillets (represented by nice “shiny” solder joints). Other sources of excessive thermal stresses can be found to be applied by: large heatsinks on the top side of the board. CEM must verify cooling every start up. Cooling slop also should be not more than 3°C per sec.

5.3 ICT/FCT process requirements.

In-circuit test (ICT) is an example of white box testing where an electrical probe tests a populated printed circuit board (PCBA), checking for shorts, opens, resistance, capacitance, and other basic quantities which will show whether the assembly was correctly fabricated. ICT tester contains from 3 main parts ICT test platform (can be used for many references), ICT test fixture (specific for product) and ICT program (software).

In circuit tester (test platform): The in circuit test system consists of a matrix of drivers and sensors that are used to set up and perform the measurements. There may be 1000 or more of these driver sensor points. These are normally taken to a large connector conveniently located on the system

Fixture: The in-circuit test system connector interfaces with the second part of the tester - the fixture. In view of the variety of boards this will be designed specifically for a particular board, and acts as an interface between the board and the in circuit tester. It takes the connections for the driver sensor points and routes them directly to the relevant points on the board using a "bed of nails".

Key parameters for ICT fixture hardware development:

- Force applied to hold the PCBA;
- Test Fixture Probe;
- Single or double fixture;
- ESD protection.

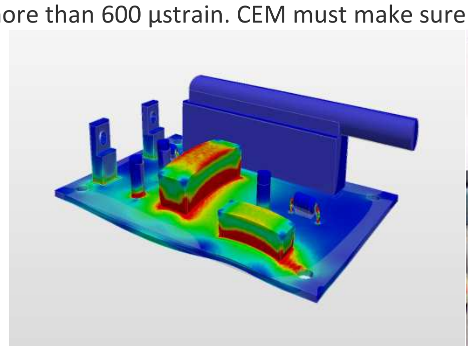
There are several common approaches to applying **force to hold a product** PCB against spring test probes, including mechanical arrangements, pneumatics and vacuum systems. In a **mechanical system** the probe plate and the product PCB are brought together by mechanical cams, levers and linkages. In a **pneumatic system**, the area between the probe plate and the product tested is pressed together by pneumatics that are placed on the top or bottom side of the fixture, sometimes both sides. Finally, in a **vacuum system**, the area between the probe plate and the product tested is a sealed cavity. A vacuum applied to this cavity pulls the two together. Examples include the following: pneumatic drive fixtures, Heavy Duty Gate (HDG) Fixtures, Vacuum Actuation – Vacuum Box Fixtures:

PNEUMATIC FIXTURE

HAVE DUTY GATE

VACUUM FIXTURE

Nature and Magnitude of the Stresses As the test fixture is actuated, the forces identified above begin to stress the board and the components mounted on the PCB. The board will flex upward or downward depending on which applied force has the greater magnitude. CEM must validate stress applier to PCBA by the fixture force during NPI phase. There're 2 methods of this validation: **Certification BSA (Board Stress Analysis):** The stress analysis on the board makes an estimate of the possible inflections that the PCI applies on the fixture. -This test is done before the fixture being built, allowing a more elaborate and complex design of the fixture. **Strain Gage Measurement:** Test of the real flexion of the PCI during the triggering of the fixture. This test uses the same technology as companies as Embraer to analyze the mechanical stress of the parties through sensors (gages) super sensitive. Target for strain gauge measurement force applied to critical components (BGA, MLCC etc) – not more than 600 µstrain. CEM must make sure that damage of components by support pins excluded.



3D BSA

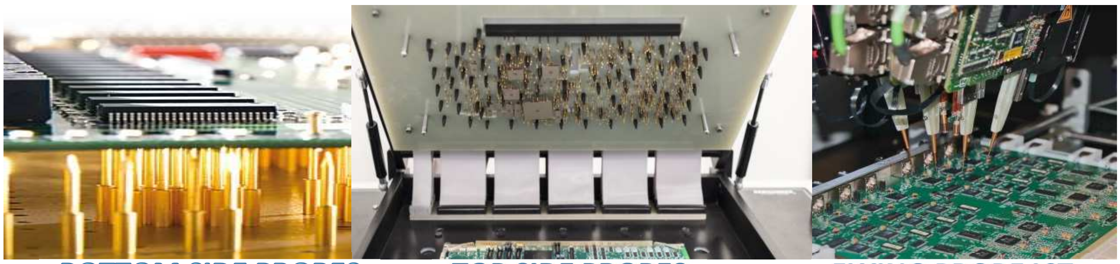
STRAIN GAUGE

Test Fixture Probe. Spring loaded test probes are available in a variety of tip styles, plating, spring forces and travel to accommodate any test target. The test targets can be plated through hole (PTH) leads, plated vias, solder pads, bare vias etc. The internal spring that exerts force on the probe tip is usually selected depending on the surface finish of the test target. High concentration of excessive probe forces can be detrimental around sensitive components which often require Finite Element Analysis (FEA) to determine the correct and safe spring forces and Strain Gage testing for verification of strain levels.

CEM must assure that correct **probe type** is used for testing. SERRATEDs are generally the most stable on long leads. Point styles with fewer tips or sharper internal geometry such as TRIADs are best used on short contaminated leads. When contacting gold plated pads, CHISELs and SPHERICALs are recommended so that marking is minimized. SPEAR, SHARP CHISEL, SHARP NEEDLE or RAZOR to increase tip contact pressure for reliable testing of heavily oxidized or flux-coated solder pads. Use a CHISEL or STAR to contact open holes. Probes contacting PCBA must exclude any damage or product. For through-hole applications use CROWN, for plated vias Piramid

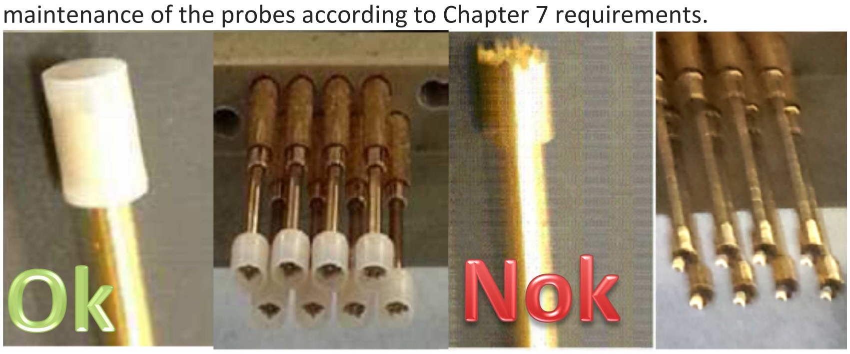
Probes and probe receptacles used in the test fixture must be documented on the fixture in a permanent, legible manner. The preferred spring force is 3 Newton's for dagger probes and 2 Newton's for other probe types. Distribution of probes – if a high concentration of high spring force probes in a specific area can damage the board-under-test. This scenario should be avoided. Probes must be compressed to 2/3 of their travel distance.

Probe Access All modern test fixtures are designed to accommodate, as the standard, bottom access to the UUT. Whenever possible, you should provide access at the bottom of the UUT for each network on the board. Fixtures can be designed to also have access to the tops and sides of the UUT. Top probes locations need to be kept far enough away from tall components to allow for the pressure plate closure arc and possible clearance holes for components. New generation of ICT also uses flying probe fixtures which can be adjusted for any type of product.



BOTTOM SIDE PROBES TOP SIDE PROBES FLYING PROBE ICT

Probe Spacing. It is mandatory to avoid short circuit between tester probes, CEM must use specific probes against short circuit or special probe support for fine pitch tester (<4mm). If there is connector pin without electrical connection on the PCBA, Pin Switch Probe must be used in order to detect bending pin due to risk of not ok mechanical assembly at Wittur or final customer. Regular maintenance of the probes according to Chapter 7 requirements.



Software (test program): Software is written for each board type that can be tested. It instructs the test system what tests to perform, between what points and details of the pass / fail criteria. It is mandatory to make a test coverage study and to work with Wittur in order to reach more than 90% of global test coverage. In case of IC with several pin, the test coverage file must have 1 line by possible short in this file (IC with 6 pin in 2 sides must have minimum 4 lines). The global test coverage will be the average of all component coverage; Missing coverage will be analyzed and cover as much as possible, mandatory for safety function (SPC, external check, incoming measurement value). Untested devices or circuit configurations will be noted in the program. Limited tests, if any, will be flagged in the body of the program by attaching "LT" to the test title.

Test Coverage Consultant report for ICT

ASTER Technologies

Board Name / Number	ANGELA_BD	Data processing report	WRK:ANGELA_BD.htm
Component	51	Component Tested	81.58%
Net	199	Net Access	98.99%
Short coverage	100.00%	Board score	92.93%

Test Techniques Used

Access	Test_Point			Unpowered Analog	Yes	Powered Analog	No
Digital	Yes	Connect Check	No	Polarity Check	No	Vectorless Test	Yes
Interconnect	Yes	IEEE 1149.6	No	Silicon Nails	No	Cover Extend	Yes
Magic Test	No	NPM	No	LightProbe	No		

Device Type	Total number (Parts or pins)	Number of well tested	Number of partially tested	Number of not tested
Integrated Circuit	6 Parts (106 Pins)	100.0% (6)	0.0% (0)	0.0% (0)
Capacitor	10 Parts	40.0% (4)	0.0% (0)	60.0% (6)
Resistor	20 Parts	100.0% (20)	0.0% (0)	0.0% (0)
Crystal	1 Parts	0.0% (0)	0.0% (0)	100.0% (1)
Connector	1 Parts (16 Pins)	100.0% (1)	0.0% (0)	0.0% (0)
Not Mounted	11 Parts			
Mechanical	2 Parts			
Total	38 Parts (13 ignored)	81.6% (31)	0.0% (0)	18.4% (7)

Test program content. Typical ICT programs will include the following four main sections:

- shorts testing
- analog power-off testing
- power-on testing

Shots and continuities testing.

Shorts - 100% of all detectable shorts between accessible nodes with a resistance of 5 ohms or less. **Continuities -** An unnamed (on schematic) but expected resistance of less than 5 ohms nominal between two nodes. Continuity checks are made for orientation of switches and relays. **Jumpers -** A named (on schematic) but expected resistance of less than 5 ohms nominal between two nodes. This includes added wires and shunts.

Analog power off test. Analog components are tested in this section without power applied to the board-under-test. **Resistors** - from 5 ohms to 100 mega ohms (including potentiometers) are verified to their value and tolerance plus system tolerances. **Capacitors** - All testable capacitors from 100pF to 999uF verified to their tolerance plus system tolerances. **Inductors and transformers -** Measured as an inductance test. **Diodes** Measured for forward voltage drop and reverse leakage (Zener to 110 Volts tested for breakdown). **Transistors** Both junctions test as diodes. 6-wire saturation tests can usually be written if ordered.

Power on test This section turns on and applies power, then tests all powered devices following these guidelines: **Voltage** 0-99 Vdc or peak ac, plus 20% over-range. Tests are aborted if improper voltages are detected. **Linear components** Op amps will be tested for full scale output swing; regulators will be functionally tested to verify their output voltage. **Digital devices** All DTL, TTL, CMOS, SSI and MSI devices will be tested. Such testing verifies that the correct device is installed, properly oriented, and functional. RAM/ROM is also tested using Gray code stimuli, which allows every address to be generated and every cell to be checked.

For ICT rules to be respected:

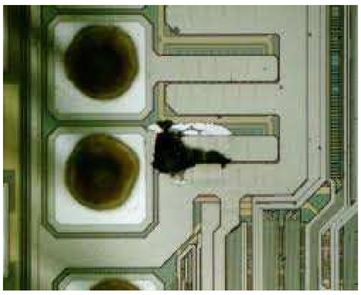
- ESD protection during testing according to Chapter 8 requirements;
- Traceability is included to ERP system and test result visible on board history. Scanners integrated into fixture in order to avoid wrong board to be tested. Test can't be started if any previous step missing on board history;
- Specific product test program must be developed. Risk to choose wrong testing program program is excluded;
- The product will be blocked when the test will start. Fixture must prevent any opening until test is finished in order to avoid reverse current;

EOS damage protection. EOS can be caused by a number of different mechanisms during the in-circuit testing of PCB assemblies. EOS-related failures in semiconductor devices can be classified by their primary failure mechanisms: thermally induced failures, electro migration, and electrified- related failures. IC supplier usually include maximum ratings into datasheet, CEM must take them into consideration when developing ICT.

Absolute Maximum Ratings

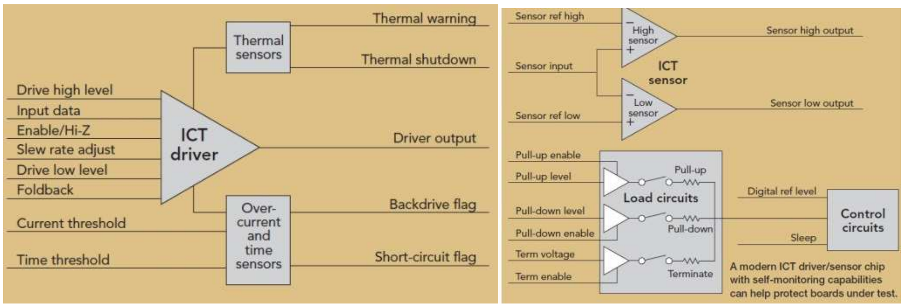
$T_A = 25°C$		VALUE	UNIT
V_{DS}	Drain-to-Source Voltage	60	V
V_{GS}	Gate-to-Source Voltage	±20	V
I_D	Continuous Drain Current (Package limited)	200	A
	Continuous Drain Current (Silicon limited), $T_C = 25°C$	349	
	Continuous Drain Current (Silicon limited), $T_C = 100°C$	247	
I_{DM}	Pulsed Drain Current (1)	400	A
P_D	Power Dissipation	375	W
T_J, T_{stg}	Operating Junction and Storage Temperature Range	–55 to 175	°C
E_{AS}	Avalanche Energy, single pulse $I_D = 128$ A, L = 0.1 mH, $R_G = 25\ \Omega$	819	mJ

(1) Max $R_{θJC} = 0.4°C/W$, pulse duration ≤100 µs, duty cycle ≤1%



Rules to be respected in order to prevent EOS:

- Diode protection with rectifier bridge in order to avoid the self-pick current and voltage in the product during on off;
- Validate that there is no power on any of the device pins when the board is inserted or removed from the tester – no "hot switching";
- Fully discharge any capacitors that are connected to the IC device before removing the board from the tester;
- Ground pins should contact the PCB first and disconnect last to allow first path for dissipation of accumulated static charges on PCBA during transport and serve as discharge path for any leftover current from the completion of ICT testing.
- ICT can include a driver/sensor IC that has the ability to monitor the output current level and determine the duration of the current.



Functional test (FCT) is used at final stage of the manufacturing process. It provides a pass/fail determination on finished PCBs before they are shipped. An FCT's purpose in manufacturing is to validate that product hardware is free of defects that could, otherwise, adversely affect the product's correct functioning in a system application. FCT verifies PCB assemblies (PCBA) functionality. It is important to emphasize that the requirements of a functional test, its development, and procedures vary widely from PCB to PCB and system to system. Advantages of Functional Test:

• Functional test simulates the operating environment for the product under test thereby minimizing the expensive cost for the customer to provide the actual testing equipment

• It can check the functionality of the product anywhere from 50% to 100% of the product being shipped thereby minimizing the time and effort on the CEM to check and debug it.

• Functional test increase overall test coverage (covers untested parts at previous ICT stage or flying probe test) making the product under test more robust and error free.

Star of testing	Supplier must develop tester start-up check list. Starting of testing must begin from checking reference bad samples – "golden samples". Tester must be validated (R&R available), maintained, verified at each start-up.
Test program	Specific test program must be developed; modification of program must be protected by user rights.
SW type verification	Functional tester must detect SW type and have Poka-Yoke to avoid shipping of PCBA with wrong SW type.
Checks for presence / position / orientation of all assembly components	Each single customer interface of the part must be tested before shipment Test jig must be designed to fit and detect if any compoennts missing.
Traceability	TF must be linked to production MES system and have program Poka-Yoke if any of tests had fail result and no rework record. All serials must be scanned and recorded into production MES. One piece flow must be respected, in order to avoid risk of skip the test of mix up good bad part. Input, output, "red box" must be identified.
Part marking / labelling / physical release conditional upon test result	Individual marking / labelling of parts must be conditional upon the successful End-Of-Line test result.
Re-test policy	Each part failing at the first test and re-tested OK is a potential intermittent defect. Re-test policy must be available and understood by operators.

5.4 Rework/debug/touchup requirements

CEM must have "Zero rework" strategy which means minimize rework activity per board. Scanning of board into ERP mandatory as rework must be visible on board history. Maximum rework quantity to be defined and respected, best practice ERP Poke-Yoke if rework operations quantity exceed. It's mandatory to retest the board in ICT and FCT after rework. Next process step should be not allowed by ERP if rework/retest is not done. Minimum request IPC7711 and IPC7721. MSL level of MSD to be taken into account if PCBA processing exceeds open time of PCB max. requirement– PCBA must be baked before rework in order to avoid delamination or popcorn effect.

Workplace must be defined (clean, order, 5S, ESD). CEM can use various equipment like hot air pencil, SMT tweezers, solder iron, magnifier, multimeter etc., but all equipment must be calibrated and suitable for EPA, work desk and chair must be grounded, 5S is the key requirement.

CEM must use only lead free solder wire for rework procedure. Using of spray **flux** is not allowed due to risk of contamination of all PCBA which has to be minimized due to risk of electrochemical migration on MLCC and chip resistor, therefore flux pen or flux gel recommended. Using of flux cleaning also not allowed during rework.

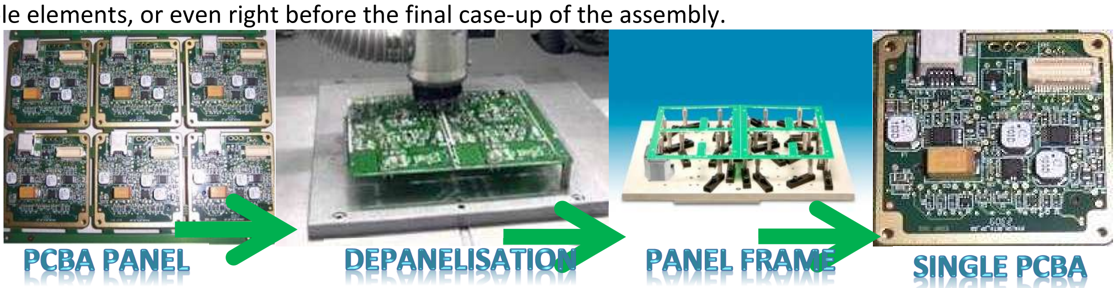
CEM must have **debug/rework procedure** which clearly describes rules for handling of PCBA during rework, how to apply heat etc. CEM will develop rework procedure according critical components rework datasheet: ceramic capacitor, LED, quartz, coal, chip resistor, IC with more than 4 pins or with soldering thermal package (MOS-fet, Dpack). A special jig can be done in order to protect critical component during this operation Key parameter during rework process heat applied to the critical components. CEM must track max. temperature to avoid heat shock of critical components.

Method	Min. Temp.	Max. Temp.
Hot Air Pencil	315 °C	400 °C
SMT Tweezers	200 °C	300 °C
Solder Iron	200 °C	300 °C

As part of in process debug or in case of customer complaints CEM must have **analyze capabilities** including trained employees, workplace & equipment. **Training** must include: knowing of product and product functionality, availability of schematics, drawings necessary for analysis, knowing of electronic circuits and ESD requirements. **Workplace** must be clean, ESD complaint, lighting must be sufficient to perform tasks, ergonomic. In order to perform **non-destructive** analysis CEM must have **equipment** like: multimeter, oscilloscope, high resolution microscope connected to PC, X-ray, environmental chambers (for temperature fluctuation or humidity). For **destructive** analysis CEM must have equipment like: cross/sectioning tool, IC soldering tool, BGA reballing tool etc.

5.5 Depanelization process requirements

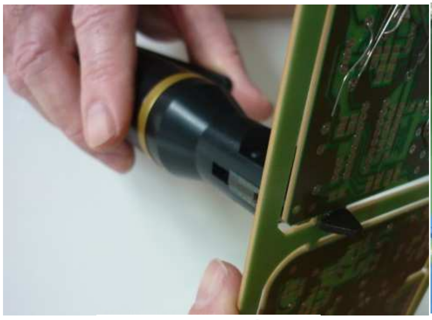
Depanelization is a process step in high-volume electronics assembly production. In order to increase the throughput from (SMT) lines, PCBs are often designed so that they consist of many smaller individual PCBs that will be used in the final product. This PCB cluster is called a panel or multiblock. The large panel is broken up or "depaneled" as a certain step in the process - depending on the product, it may happen right after SMT process, after in-circuit test (ICT), after soldering of through-hole elements, or even right before the final case-up of the assembly.



Main depanelization technologies:

There are six main depanelization cutting techniques currently in use:
- Manual cutting;
- Sawing/blade cut;
- Punching
- Routing
- Laser cut

Hand break This method is suitable for strain-resistant circuits (e.g. without SMD components). The operator simply breaks the PCB, usually along a prepared V-groove line, with the help of a proper fixture or cutting tool. Hand break is forbidden for serial process due to lack of process control. **Punching** is a process where single PCBs are punched out of the panel through the use of special fixture. It is a two-part fixture, with sharp blades on one part and supports on the other. The production capacity of such a system is high, but fixtures are quite expensive and require regular sharpening. Punching not allowed due to mechanical high stress.


MANUAL CUT


PUNCHING

Sawing/blade cut A saw is able to cut through panels at high feed rates. The disadvantages are: ability to cut in straight lines only and higher stress than for routing. A **sawing** uses a rotary blade, sometimes 2 blades: one on top and one on bottom, rotating using motor. The operator moves a panel along line, usually with the help of a special fixture. **Blade cut** uses straight shape blade to cut the panel. An aluminum based jig must be used to secure the PCB in place and position of the panel.

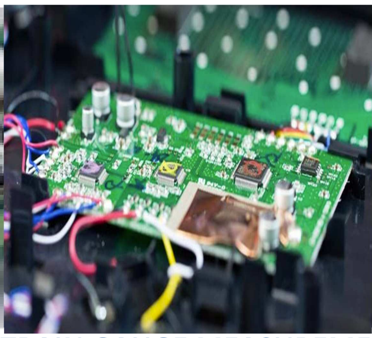

SAWING


BLADE CUT

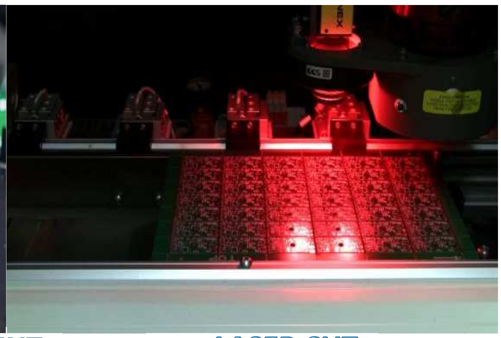
Routing is preferred Wittur process due to high precision and lower risk of scrap. A Depaneling **router** is a machine which uses a **router bit** to mill the material of the PCB. Routing must be part of PCBA traceability at ERP and board must be scanned prior to routing. Specific routing program must be created according to panel drawings.

Key parameter to control for routing process:
- **The hardness of the PCB** material wears down the bit, which must be replaced periodically. Routing requires that single boards are connected using tabs in a panel.
- Routing produces much **dust** that has to be vacuumed and ionized. It is important for the vacuum system to be ESD-safe.
- **Fixturing** of the PCB must be tight - usually an aluminum jig or a vacuum holding system must be used.
- **Feed rate** and **rotational speed**. They are chosen according to the bit type and diameter and should remain proportional (i.e. increasing feed rate should be done together with increasing the rotational speed).
- Routers generate **vibrations** of the same frequency as their rotational speed, which might be important if there are vibration-sensitive components on the surface of the board. The strain level must be measured during NPI stage and not higher than 600 µ strain.

ROUTING BIT



STRAIN GAUGE MEASUREMENT



LASER CUT

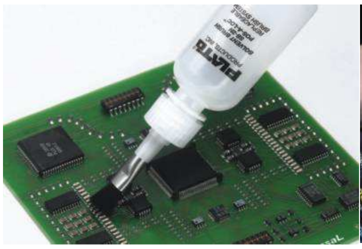
Laser cutting UV laser depanelization makes use of a 355 nm wavelength (ultraviolet), diode-pumped, Nd: YAG laser source. At this wavelength the laser is capable of cutting, drilling and structuring on rigid and flex circuit substrates. The laser beam, capable of cut widths under 25µm, is controlled by high-precision, galvo-scanning mirrors with repeat accuracy of +/- 4 µm. Advantages: accuracy, precision, low mechanical stress and flexible contour and cut capabilities. For laser cutting process periodic calibration of laser must be done.

5.6 Coating process requirements.

Conformal coatings are designed to protect printed circuit boards and related equipment from their environment. Typically applied at 25-75µm, these coatings ‘conform’ to the contours of the board allowing for excellent protection and coverage, ultimately extending the working life of the PCBA. Conformal coatings can be used in a wide range of environments to protect printed circuit boards from moisture, salt spray, chemicals and temperature extremes in order to prevent corrosion, mold growth and electrical failures.

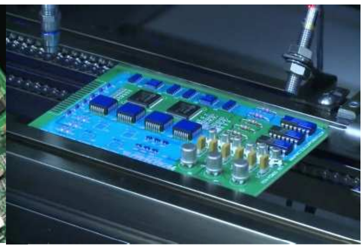
Application Method

Conformal coatings can be applied via spray, dip or brush methods either by manual or automated application. **Brush coating** This works by flow coating the material onto the board and is suitable for low volume application, finishing and repair. The finish tends to be inferior cosmetically and can be subject to many defects such as bubbles. **Spray application coating.** This coating can be completed with a spray aerosol or dedicated spray booth with spray gun and is suitable for low and medium volume processing. Those coating methods mostly applicable for prototype, sampling production or rework, must not be used at serial conditions. **Dipping coating** method not allowed for Wittur products due to critical components to be not coated.



BRUSH COATING

SPRAY COATING



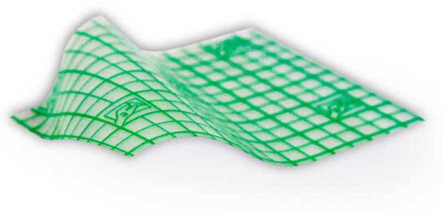
SELECTIVE COATING

Selective coating by machine This method is the best choice for high volume applications as it is a fast and accurate way of applying the desired thickness coating, to precisely the areas of the board where it is required. It works by using a needle and atomized spray applicator, non-atomized spray or ultrasonic valve technologies that can move above the circuit board and dispense / spray the coating material in selective areas. Flow rates and material viscosity are programmed into the computer system controlling the applicator so that the desired coating thickness is maintained.

Key process requirements for selective coating process:

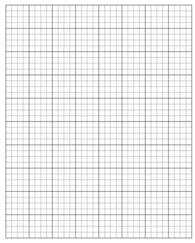
- Before starting coating process, selective coating program must be verified with paper or foil test, evidences must be stored minimum for warranty period.
- Specific automatic machine program must be developed. Critical components to be not coated: aluminum electrolytic capacitor, transformer, relay, switch, CHL capacitor, connector, fuse-holder, LED. CEM must develop procedure which describes area to be coated and to be not coated. a special palette can be used in order to protect this area;
- Coating process step must be included into PCBA board history and tracked, automatic scanning process to be installed prior to enter the coating machine;
- The head movement must eliminated the risk of component damaging during dispensing;

- The coating nozzle must have auto check for axis reference X,Y, Z at the production startup in order to avoid component damaging.
- In order to reduce the risk of short circuit between IC pin, due to conductive particle drop off by the varnish. The PCBA must be vacuum cleaned before coating operation (no wave solder ball or splash).



TRANSPARENT FOIL

THICKNESS GAUGE



PAPER FOR COATING TEST

Curing and drying

Moisture (Humidity) Curing: Silicone coatings offer rapid curing when temperature and humidity are controlled. Typical silicone coating curing conditions are 60 to 80°C at 10 to 15% relative humidity with timing at curing cabinet. Since relative humidity falls as temperature raises it is often required to enhance the moisture content in the airstream. **Thermal (Heat) Curing:** Solvent based coatings cure or dry as solvents evaporate and off-gas from the liquid coating, leaving behind the solids that comprise the protective coating. To accelerate the curing or drying process thermal curing ovens are often used. Thermal curing can also be employed for silicone based coatings as well as a secondary cure mechanism for some UV Curable Coatings. Usually curing ovens are used. CEM must develop temperature profile for curing oven and follow the maintenance procedure on weekly base.

Ultraviolet (UV) Curing: Some adhesives and conformal coatings cure upon exposure to Ultraviolet (UV) energy and/or Visible Light with wavelengths in the range of ~254-460nm. UV curable coatings are 100% solids and contain no solvents. One problem associated with UV curable coatings is shadowing caused by components on the printed circuit assembly. In recent years, coating manufacturers have addressed this issue by offering formulations that offer a secondary curing mechanism. The secondary curing mechanism is typically moisture cure or thermal cure. UV laps are used for this type of curing. CEM must have lifetime calculation for the lamps and replace them on as part of preventive maintenance procedure.

CURING CABINET

CURING OVEN

UV CURING

Thickness and measurement

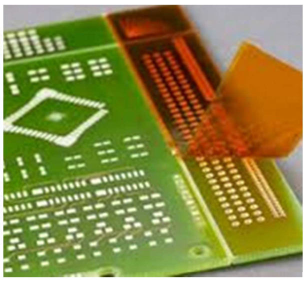
Coating material when dry (after curing) should typically have a thickness of 30 –130 µm (0.0012–0.0051 in) when using acrylic resin, epoxy resin, or urethane resin. For silicone resin, the coating thickness recommended by the IPC standards is 50–210 µm (0.0020–0.0083 in). Wet film & dry film conformal coating measurements. CEM must measure conformal coating at least during start up and as SPC in process control.

Wet film conformal coating measurement The wet film conformal coating thickness method ensures quality control while the coating is still wet. Wet film measurements are useful for conformal coatings where the dry film thickness can only be measured destructively or over application of conformal coating could be problematic. The wet film gauges are applied to the wet conformal coating and the teeth indicate the thickness of the conformal coating. The dry film thickness can then be calculated from the measurement.

Dry film conformal coating thickness measurement An alternative method to wet film measurement is using a non-contact technique using eddy currents. The system works by placing the test head on the surface of the conformal coating, the measurement is almost instantaneous and provides an immediate repeatable result for thickness measurement of conformal coating.

Coating reliability testing. CEM must perform the adhesion test described in the IPC-TM-650 2.4.1. on frequent base for each new batch of coating. Basically, this test consists in scarifying the varnish, in applying an adhesive tape and in removing it with a defined angle and speed. To pass the test, no coating should stick to the tape.


DRY COATING


TAPE ADHESION TEST

5.7 Visual inspection

Usually visual inspection contains from 2 process steps: 100% visual control and out of box sample inspection (OBA). For sufficient inspection the following factors to be applied:

- Work station;
- Environment;
- Work instruction;
- Operator training;
- Non-conformity management;
- Quality management.

Work station. Work station, for inline 100% check special equipped workplace must be defined, for OBA - independent of the production line and in an identified zone. Magnification is sufficient to inspect PCBA (min 5 times). Lighting intensity strong enough to enable the operator to detect the defects. Inspection workplace is ESD compliant according to Chapter 8 requirements. Clear and separated zone identified for: 1) products awaiting inspection 2) products with completed inspection. All equipment for proceeding to the CSL1 activity is available (tools, devices, gauges). Measuring equipment calibrated. Are all accessories available and used (gloves, magnifier, labels) Do all conditions prevent from risk for mixing, damaging, dropping the parts. Is the workplace ergonomic.

Environment. The noise smooth enough to avoid any disturbance to the inspector during the inspection activity. Temperature adequate to perform the inspection activity. 5S respected within and around the work station

Work instruction existed and up to date, aligned with CP Master samples / boundary sample clearly identified, available on work station and used by operator. Failure catalog available and updated. ERP with scanning recording system in place for registering the quantity and the detected defects during inspection activity.

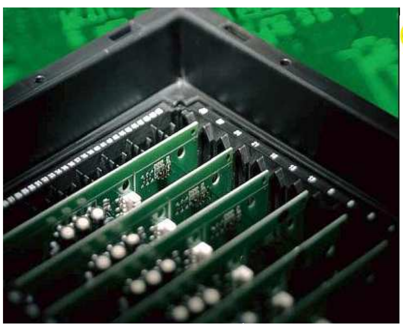
Operator training. Operators are trained to perform the tasks according IPC A610, class 2 requirements. In case of several operators, a matrix exists with the list of trainees. Operator must have a scheduled break or switch the workplace. Work instruction is known and respected.

Non-conformity management. Red box located in the area. All parts must be scanned into ERP with fail or pass result. Defects marked in real time with clear identification/description. Information of defects communicated to the production line and to plant quality in order to place actions. Line stop criteria must be defined and respected.

5.8 Packaging requirements

In process packaging requirements

In process transportation must assure ESD protection and exclude mechanical damage. Allowed to use ESD boxes with separators or PCB magazines. In case of using magazines CEM must develop separators Poka-Yoke or block row to respect minimum distance save for PCBA to be not damaged and prevent against falling down. ESD trays for in process transportation not recommended, but if used must assure side support and prevent damage of one PCB to other.


ESD BOX


ESD TRAY


ESD MAGAZINE

Finish good packaging for PCBA

In order to be sure that the PCBA will not skip any process step, the CEM must scan the final individual label for PCBA/finish good of the final packaging and automatic the label printing is done only if all the units are conform (ok at all process step).

Each PCBA packaging type must fulfill 3 functions:

- For ESD risk prevention;
- Mechanical damage prevention;
- Dust contamination and humidity impact prevention.

CEM can use returnable and non-returnable packaging types

Returnable packaging should be cleaned after every rotation; CEM will define how to clean it. All of this empty packaging will be stored reverse in order to be protected against dust. CEM must use ESD containers protection with antistatic separators inside according to the packaging specification for PCBA transportation packaging risk of mechanical damage must be excluded.



ESD PLASTIC BOX WITH COVER

ESD CARTON BOX

Non-returnable packaging (carton boxes) can be also used for long distance shipments. In that case PCBA/finish good must be places into ESD shielding bag. Preferable is to use ESD protective carton type for boxes and separators. Non-returnable packaging must assure ESD compliance, mechanical and environmental protection and must contain from ESD shielding bags, carton boxes and separators and wrapped with foil outside of box in order to protect from dust, humidity and mechanical damage. **Shipping requirements.** Shipping has to exclude any damage of finish good. CEM must defined pallet maximum height and q-ty of boxes per pallet in order to assure optimal delivery conditions of finish goods.



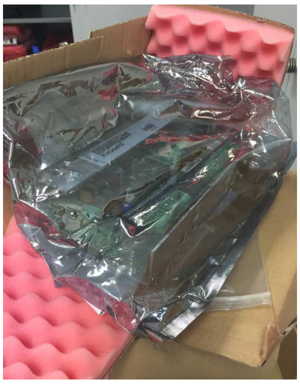
ESD BAG

CARTON BOX

SHIPPING PALLETE

Spare parts packaging. Samples or spare parts also must meet requirements stated above regarding ESD protection and mechanical protection. CEM can use ESD shielding container or carton container with use of ESD shielding bag.

ESD Shielding Container



Not allowed to use for transportation bubble foil bags or pink ESD bags (dissipative) as they're not shielding materials and don't protect outside of ESD protected area.

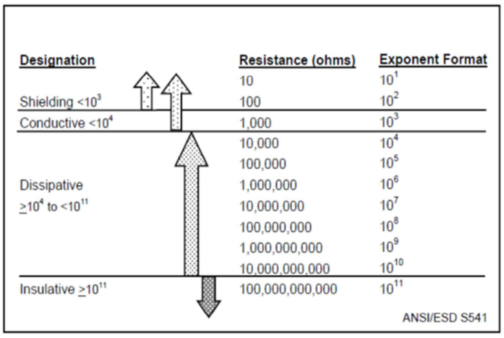


Figure 3: Resistance Classifications

Table 2. Summary of Protective Properties

Protection	Property
Low Charging (antistatic)	Materials that have reduced amounts of charge accumulation as compared with standard packaging materials.
Dissipative or Conductive Resistance	Provides an electrical path for charge to dissipate from the package.
Discharge Shielding	Protects packaged items from the effects of static discharge that are external to the package.