Electronic Board Assembly

ERNI Systems Technology

Systems Solutions
- a one stop shop -

www.erni.com
Contents

ERNI Systems Technology

Soldering Technologies

- SMT soldering - Reflow Soldering of Surface Mount Devices
- THR soldering - Through Hole Components, reflow soldered
- THT soldering - Through Hole Components, wave soldered
  - Selective THT process
    - Masked Wave Soldering
    - Mini Solder Wave

- Soldering Equipment

- Testing of electronic assemblies
  - AOI – Automatic Optical Inspection
  - Electrical Testing

Pressfit Technology

- Pressfit zone
- PCBs for pressfit technology
- Tools
- Testing
ERNI Systems Technology

Customer

ERNI Sales Organisation

ERNI Systems Technology

Design + Engineering

Soldering + Integration

Pressfit Technology + Integration

Cable Assembly

Products of ERNI Systems Technology

Electronic Assemblies

Backplanes

Subracks

Cable Assemblies
Soldering Technologies: SMT / THR / THT

Terms & Definitions:

**SMT**  Surface Mount Technology
Mounting technology for SMDs (Surface Mount Devices)
The assembly is carried out using the **reflow soldering** process (e.g. convection, vapor phase).

**THR**  Through Hole Reflow
Mounting technology for components in through hole technology (wired components) for automatic pick & place process.
Components have to be resistant against soldering heat.
The assembly is carried out using the **reflow soldering** process.
THR process also is known as: PIP Pin In Paste
PIH(I)R Pin In Hole (Intrusive) Reflow

**THT**  Through Hole Technology
Mounting technology for components in through hole technology using the **wave soldering** process

SMT process steps
for assembly of SMDs on both sides of the board

- screen printing of solder paste
- SMD assembly on the 1\(^{st}\) side
- 1\(^{st}\) reflow soldering
- turn board upside down
- screen printing of solder paste on the 2\(^{nd}\) side
- SMD assembly on the 2\(^{nd}\) side
- 2\(^{nd}\) reflow soldering
- double reflow capability for components on bottom side
Challenge of SMT soldering process

- Process and design know-how necessary

- Hand soldering is limited (challenge for amateurs)

- Investment necessary in paste printer, pick&place machines, reflow soldering systems, handling systems, etc.

- Higher heat impact on components

- Double reflow soldering capability is important for double side assembly:
  - component weight / soldering area = max. 0.1 g / mm²
  - temperature stability: min. 2x reflow soldering temperature

- Coplanarity of SMD pins

  coplanarity
  max. 100µm

- Reduced stability of PCB solder pads against mechanical load (typical 1N / mm²)

- Complex test equipment for miniaturized components necessary (possibly X-ray inspection instead of optical inspection for BGAs or SMT connectors like MicroSpeed, ERmet zeroXT, …)
**THR – how it works!**

**Definition in IPC-A-610:**

1.4.7 Reflow Soldering of Through Hole Components

A process, by which the solder paste for through hole components is applied using a stencil, to solder these components together with the surface-mount components in the reflow process.

**Challenge of THR soldering process**

- Components should be suitable for automatic pick & place process. Dedicated pick & place-pads can be used.
- Components have to withstand the thermal impact during the reflow soldering process.
- Thickness of PCB should be limited to 2mm.
- Diameter of plated through hole has to be adjusted to pin diameter of component, typical: diameter of plated through hole = pin diagonal + 0.1mm.
- Solder filling of plated through hole has to be according IPC specification: minimum 75% fill. Sufficient solder paste has to be provided.
- Solder Preforms can be used at critical points in order to rise the amount of solder.

**Note:**

For THR technology **sufficient solder paste** volume has to be applied, in order to fill up the distance between solder pin and metallized hole during the reflow soldering process. For this an appropriate area has to be printed with solder paste, occasionally also the solder resist surrounding the pad.
**Combined soldering technologies: SMT / THR / THT**

- screen printing of solder paste
- SMD assembly
- THR components assembly
- reflow soldering
- adhesive application for fixing of SMDs (for subsequent wave soldering) using:
  - stencil printing (only for SMT)
  - dispensing
    (if terminations of THR components penetrate the printing area)
  - SMD assembly
  (for subsequent wave soldering)
  - baking of adhesive (in reflow oven)
- THT components assembly
- wave soldering

**Note:**
For the **combination of SMT and THR process** without wave soldering, THR component should always be placed on the 2nd soldering side. Thus also heavy THR components can be soldered by the reflow process.
Selective THT process

Selective wave soldering is recommended, if a lot of SMDs are already assembled to the PCB and only few THT components have to be soldered afterwards. With selective soldering only a part of the PCB is exposed to the solder wave. This is opposite to the standard wave soldering process.

Masked Wave Soldering

One variant of this process is masked wave soldering. Hereby sensitive areas of the PCB or already soldered SMDs are covered by a mask, made of thermal resistant plastic material.

Example:
Solder mask for 2 boards in a solder frame. In the black areas the PCB is protected against the solder bath.

Mini Solder Wave

The solder joints are produced selectively by moving a solder nozzle or the PCB in X- and Y-direction accordingly.

The solder nozzle with variable diameter of 6, 8 or 10 mm can also be moved to hardly accessible solder joints on the board.

The nozzle itself is flooded with hot nitrogen (protective gas), in order to impede oxidation. The result is a high reliable solder joint.

A preheating of the PCB delivers an adapted temperature niveau (substantially for the leadfree soldering process).

Note:
If pressfit technology will be applied to the board after the wave soldering process, pressfit holes have to be covered while soldering. This can be realized by peelable solder resist, adhesive tape or solder mask in a selective soldering process.
Soldering Systems

SMT lines for leadfree and leaded reflow soldering

- Stencil printing using squeegee blades
- Stencil thickness: 120 – 200µm
- Different solder pastes
  - lead free: SAC305
  - leaded: SnPb
  - or adhesive
- Single / double printing
- SMD component placement
- Standard / special components
- Placement accuracy by:
  - Laser and vision recognition
  - Multi-nozzle laser head (4 nozzles)

Reflow soldering system
- Nitrogen control (protective gas)
- Integrated cooling system

Temperature profiling for
- Lead free soldering
- Leaded soldering
- Adhesive curing (for wave soldering)

THT lines for leadfree and leaded wave soldering

- Manual placement of components in through hole technology (THT)
- 7 Workplaces for manual component placement
- 7 Heating zones
- Automatic solder supply
- Automatic solder angle adjustment
- Integrated cooling system
- Lead free solder: Balver Sn100Ce
Test of electronic assemblies

**AOI - Automatic Optical Inspection**

Digital Scan System for detection of
- text
- polarity
- solder defects
- solder filling rate
- component failure
- miss alignment
- tombstoning
- dirt, dust, etc.

Telecentric lens for compensation of image distortion
Optimized LED illumination system
Automatic calibration of pixel brightness
- high picture resolution 18 µm
- high scanning speed 460 x 500mm in 19s
- board thickness range 0,5 – 5,0mm

**Electrical Testing**

Electrical testing of electronic assemblies
- Functional testing
  (automatically and manually)
- In-Circuit-Tester
  (partially with integrated functional test)
- Repair after functional testing
  (if necessary)

Test equipment
- Genrad digital- and analog tester
- In-house designed and produced test equipment
- Customer designed test equipment

Evaluation of test software
- In-house evaluation of test software
- Adaptation of CAD data

Test adapter
- Needle bed adapters
- Customer specific adapters
- In-house design and manufacturing
Pressfit Technology

As a technology leader in Pressfit Technology ERNI offers tools, components, know-how in production and test technology for interface boards, backplanes and complete systems.

As a pioneer in Pressfit Technology ERNI has driven this production technology substantially during the last decades. For various applications Pressfit Technology is the most economical solution, and the only one, if soldering is not suitable for complex constructions. The automatic machines, designed by ERNI, combine the assembly and the Pressfit Technology to one production step to increase the productivity.

Terms & Definitions:

The Pressfit Technology describes a solderless electrical connection technique for printed circuit boards PCBs). Hereby a pressfit pin has to be pressed into a metalized hole (plated through hole = PTH) of a PCB. The characteristic is the diagonal of the pin profile, which is larger than the diameter of the PTH.

During the press-in process there will be a deformation in the pin geometry as well in the copper sleeve of the PTH.

Two different kinds of pressfit zones are available:
- pins with massive pressfit zone
- pins with compliant / flexible pressfit zone

The press-in process with the accompanied force of the pin onto the copper sleeve creates a gas-tight, corrosion protected electrical connection, distinguished by high quality and reliability. Pressfit Technology is described in the European standard EN 60352-5.
Pressfit Zone

For the manufacturing of a pressfit connection different shapes of pressfit zones are suitable. From the compliant pressfit zones the “eye of the needle” is most commonly used.

The „eye of the needle“ combines many attributes for the manufacturing of a reliable connection:

- relatively low press-in force with low impact on the copper sleeve (easy to repair)
- high flexibility offers good durability (reliability)
- only little variance in press-in force
- similar press-in and press-out force
- suitable for the application of different contact materials

Terms & Definitions:

The compliant or flexible pressfit zone will be deformed elastically during the press-in process. Thus a relatively low press-in force can be achieved.

With this flexibility, a permanent force onto the copper sleeve is generated, which provides a reliable connection, also under environmental conditions.
Printed Circuit Boards for Pressfit Technology

The plated through hole for pressfit technology consists of a copper sleeve with an appropriate finish.

Basic requirement for the manufacturing of a pressfit hole is the use of a drill with the dedicated diameter. For calculation of the finished hole diameter the following formula can be used:

\[
\text{drill diameter} - 0.1\text{mm} = \text{diameter of the finished hole}
\]

The tolerance of the finished hole diameter is typically specified to +/-0.05mm.

The following example shows the hole construction for a diameter of \(0.6 +0.05 / -0.05\) mm

For the protection of the copper surface, different (leadfree) finishes are available:

- **Immersion Tin**, with a thickness of about 1.0µm, is the most commonly used finish for pressfit technology
- **Electroless Nickel, Immersion Gold (ENIG)**, with a thickness of about 5µm: Nickel barrier with gold flash (0.05 – 0.1µm)
- further organic, chemical or galvanic finishes available
Press-in Process

For the press-in process the following parts and tools are necessary:
- printed circuit board
- pressfit connector
- pressfit tools
- press-in equipment with the capability for the total force of the pressfit component.

During the process neither the PCB nor the connector should be damaged. Therefore dedicated tools for the support of the PCB bottom side are necessary.

Press-in Tools

Examples of press-in tools for connectors according DIN 41612 / IEC 60603-2

- **for male connectors**
  - upper tool for support of male contacts
  - press-in force onto dedicated male contacts
  - lower tool for support of the PCB

- **for female connectors**
  - upper tool: flat stamp with guiding function
  - press-in force onto dedicated male contacts
  - lower tool for support of the PCB

**Note:**
During the press-in process, it is essential that both the PCB and connector insulation body are not placed under excessive stress, otherwise damage may occur. It is recommended that a **space distance of 0.05 - 0.1mm** is maintained between the connector and the PCB. In every case, to ensure optimum efficiency and quality, it is recommended to use the appropriate ERNI press machines that utilize **limit value controls**.
Advantages of Pressfit Technology

- Excellent electrical contact with high reliability (lower failure rate than soldering)

- Mountable on both sides of the PCB

- Hybrid assembly possible (together with soldered components)

- For multilayer boards with various thickness

- No thermal stress

- No extended requirements of leadfree process

- Pressfit components are replaceable (repair capability)

- I/O (transfer technology) and wire wrap
Pressfit tools and testing

- Tools for different types of connectors available
- Process control and monitoring
- Component data base available
- Monitoring and recording of press-in force

Fully automated backplane test equipment

- Automatic Optical Inspection (AOI)
- Electrical and functionality test
- Test solutions for high contact density
  (> 100,000 test points, PCB dimensions up to 1400 x 600 mm)
- Interconnection test (cross-connect) - „no openings”
- Insulation test (up to 1500V) - „no shorts”
- Impedance matching test - „no discontinuities”
- Electrical testing of passive components (resistors, capacitors, …)