In the past years, the process of printed circuit board assembly has undergone a fundamental transformation. Conventional Through-Hole-Technology (THT) has been increasingly replaced by Surface-Mount-Technology (SMT). The reasons for this were both constantly changing requirements such as miniaturization of components, increased density of functions and lower production costs as well as the vast advances achieved in the development of Surface-Mount-Devices (SMD), which made use of the SMT production process possible. Today, SMT has established itself as the standard in printed circuit board (PCB) assembly.

But there are still a few components (mainly electromechanical components such as connectors or relays) that are not available as SMD versions and which, up to now, had to be manually placed on the PCB after the SMT process and in accordance with the classical THT production method. This mixed technology process of pcb manufacturing results in duplicated processes and machinery as well as increased labor costs. This is why Weidmuller has developed a product range for the so-called Through-Hole-Reflow (THR) process which makes it possible to use through-hole components in the SMT process, thus ensuring one hundred percent constancy in SMT production.

With this brochure, we would like to provide you with a practical guide with which you can choose the notes on pcb assembly that apply best to your requirements. If you wish to find out quickly and precisely what solution or product Weidmuller recommends within the scope of the technology, then simply read the text in the offset boxes.
TABLE OF CONTENTS

1 From THT (Through-Hole-Technology) to SMT (Surface-Mount-Technology) – the Evolution of PCB Assembly 4
2 THR – (Through-Hole-Reflow Technology) the Revolution in PCB Assembly 5
3 The SMT Process for Modules with THR Components 6
3.1 The Process Steps at a Glance 6
3.2 Design of the PCB and Stencil 6
3.3 Paste Printing 10
3.4 Component Placement 12
3.5 Reflow Soldering 14
3.6 Quality Control 16
4 SL-SMT - Weidmüller's Innovative THR Components for the SMT Process 18
THT – Assembly with Through-Hole Components

When manufacturing using Through-Hole-Technology (THT), which was the standard process in electronic module production up to a few years ago, the wire terminals of the components are inserted into the through-holes on the printed circuit board. Boards are populated manually or with special placement machines. The hand or wave soldering process is used.

Drawbacks of THT:
• production is time-consuming and costly
• low function density
• increased susceptibility to faults due to a lack of manual precision
• only one sided component placement is possible

SMT – Assembly with SMD Components

With the advent of Surface-Mount-Devices (SMD), Surface-Mount-Technology has increasingly taken the place of THT. And the trend towards producing electronic modules by surface mounting is continuing. As the current standard process in electronic module production, SMT is defined by the method of placing components and the soldering process employed. Thus, when the SMT process is used, PCBs can be populated with SMD components both on one side or on both sides. The boards are assembled with machines, such as "Pick & Place" or "Collect & Place" systems, which handle the components in the packaging in which they were delivered, eliminating the need for special preparation of the components.

The SMT process, employs a reflow soldering method such as infrared, convection or vapor phase soldering. Contrary to the wave soldering process, the reflow process only requires a solder paste deposit applied to the PCB which forms the solder joint during reflow.

Advantages of SMT:
• high quality
• lower processing cost
• smaller components
• high packaging and integration density
• optional single-sided or double-sided component placement

Manual component placement and the use of manual and/or wave soldering for SMT manufactured PCBs are costly and critical processes, which can lead to quality defects. So you can see the need for achieving consistent SMT compatibility; i.e. creating through-hole components which are suitable for the SMT reflow soldering process, thus reducing manufacturing costs and eliminating the wave soldering process, which is susceptible to faults. The special process developed for this purpose is known by various names on the market, namely Through-Hole-Reflow (THR), Pin-in-Paste (PIP) or Pin-In-Hole-Intrusive-Reflow (PIHIR).
Advantages of THR:
• stable connection to the PCB
• only one soldering process
• manual/wave soldering is eliminated
• automatic component placement
• lower overall production cost

For THR components to be processed in the SMT process, certain demands have to be met. These are described in greater detail in Chapter 3.

Through-Hole-Reflow is a process for combining through-hole components with the SMT reflow soldering process. In addition to conventional SMD placement, when THR is employed the SMD components are placed onto the SMT board at the same time as the through-hole components and are soldered during one reflow process.

The THR process was the result of the knowledge that stable soldered joints are still needed for heavy components such as coils or transformers and wherever mechanical forces act on the PCB (e.g. in the case of connection terminals, connectors, relay sockets, etc.). For these types of applications, SMD soldered joints (gull-wing type) are not suitable. For these heavy components, the stress exerted onto the solder joint can damage the solder joint or sheer the component from the PCB.

A SMT compatible product is the solution. A compromise between optimum SMT capability and maximum stability of the soldered joint. With through-hole pins, THR ensures a stable joint between the THT components and the PCB while achieving 100% SMT process compatibility at the same time.

Mechanically non-stressable connection: e.g. “Gull Wing” type cannot endure mechanical stress without damaging or breaking the solder joint.

Mechanically stressable connection: THT absorbs much more mechanical stress without damaging the solder joint.
3 THE SMT PROCESS FOR MODULES WITH THR COMPONENTS

3.1 The Process Steps at a Glance

1. Design-In: design of the PCB and stencil layout for THR components

2. Paste printing: solder paste applied into THR placement holes

3. Placement: inserting the THR component pins in the solder paste

4. Reflow soldering: melting on the solder paste

5. Quality control: assessment of the finished THR soldered joint

3.2 Design-In

The SMT production process begins with the design of the PCB. It is as early as in this first step that the later smooth production process sequence and thus the quality of the PCB assembly process are defined. Compared to the classic SMD or Through-Hole-Technology, THR requires that a few special features be observed in the design of the PCB. Adhering to the design recommendations ensures an optimal production process.

Printed circuit board design for SMD components is generally very easy. Once the design of the soldered joints has been defined, the layout of the stencil holes is often defined about 10% smaller in order to prevent process errors during paste printing.

By contrast, in the case of THR components, a drill hole must be filled with solder paste. For this, the quality of the soldering process depends on what degree of filling of the holes with solder paste can be achieved during the printing process.

To optimize the degree of filling, Weidmüller recommends the following steps:

3.2.1 define the diameters of placement holes and soldering eyelets

3.2.2 calculate the solder volume or the necessary paste volume

3.2.3 determine the degree of solder paste filling

3.2.4 design the stencil layout
3.2.1 Defining the Through-Hole and Solder Eyelet Diameters

To calculate the necessary paste volume, first define the volume of the soldered joint in accordance with the required quality. For an adequate soldered joint, the internationally recognized quality standard IPC A610B calls for a 75% filling height in the PCB (see also Section 3.5 “Quality control”). The designer first defines the relevant diameter of the through-hole and of the solder eyelet. Both parameters determine the necessary solder volume for a given THR component.

**Through-hole diameter:**
For THR soldered joints (contrary to wave soldering), a slightly larger through-hole diameter is advisable because sufficient space is needed for the paste to flow into the holes. Solder pastes also consist of a mixture of soldering globules and flux, and are categorized according to grain sizes. To avoid jamming or friction of the pin with these soldering globules while placing components into the holes, we recommend choosing the smallest grain size possible.

When designing the through-hole diameter in preparation for the automatic placement process, the tolerances of the PCB, the placement machine and the component must be taken into account. For example, for connectors the position of the pin end in the holes is subject to special tolerances, so the respective manufacturer’s task is to optimize the pin end’s position. For Weidmüller, it’s easy due to the short pins and their smaller circle of tolerance.

Weidmüller recommends the following as suitable through-hole diameters for THR soldering process:

- For round pins: pin diameter + at least 0.3 mm
- For rectangular pins: pin diameter + at least 0.25 mm

**Soldering eyelet diameter:**
The volume of the solder meniscus should also be optimized in order to minimize the required solder paste volume. This is achieved by minimizing the soldering eyelet diameter compared to typical diameters used in classic Through-Hole-Technology.

Recommended soldering eyelet diameter for through-hole soldered joints:

\[
\text{through-hole diameter} + 2 \times \text{residual ring width} = \text{soldering eyelet diameter}
\]

The residual ring width usually amounts to 0.3 mm. For THR components such as connectors a slight increase in the residual ring width to about 0.4 mm is advisable for reasons of higher soldered joint stability and repairability.
3.2.2 Calculating the Solder Volume or the Necessary Paste Volume

The solder volume that is necessary for an optimum soldered joint is calculated as follows:

\[
\text{through-hole volume} + \text{meniscus volume} - \text{terminal pin volume} = \text{solder volume}
\]

3.2.3 Determining the Degree of Solder Paste Filling

For an optimum layout of the PCB, it makes sense to check whether the previously calculated necessary paste volume can be achieved at all. Therefore, a test should be conducted to determine the degree of solder paste fill in a PCB hole. For example, a test PCB with the defined hole diameters, should be run through the paste printing process. The filling height of the holes in the PCB can then be checked visually.

3.2.4 Design of the Stencil Layout

If the optimal degree of solder paste fill was achieved, the standard design rule can be used for the stencil hole design: stencil hole diameter about 10% smaller than the soldering eyelet. Diverse process parameters can contribute to a situation in which the necessary degree of hole filling is not achieved. When this occurs there are several options at the printed circuit board designer's disposal for optimizing the degree of fill:

- optimizing the printing parameters (see Section 3.3 "Paste printing")
- optimizing the stencil layout: If there is not enough paste, printing over the soldering eyelet is advisable. Various forms of asymmetrical printing can also be applied. As an alternative, stepped and additive stencils or double printing by means of...
extremely thick stencils can also be applied. If there is too much paste, webs over the breakouts of the stencil may function as a paste brake.

Additional design factors:
For a smooth SMT production process, the following factors should also be observed when designing a module:

• shadow formation in the reflow oven, caused by high housings of the THR components, can be avoided by keeping an adequate distance between the components
• the components' contact faces must be taken into account to ensure that the insulator does not come into contact with the paste
• securing the component by gluing may be necessary in double-sided board assembly if a THR component is placed on the first placement side

Solutions and recommendations from Weidmuller
In a first draft of the printed circuit board and stencil layout, you can safely work with the known standard process parameters.

For their SL-SMT pin headers with short pins (1.5 mm), Weidmuller suggests the following PCB design:
Through-hole diameter: 1.5 mm
Soldering eyelet outside diameter: 2.3 mm
Stencil hole diameter: 2.1 mm (assuming adequate degree of paste filling)
Stencil hole diameter: 2.8 mm (if the degree of paste filling is not enough)
Valid for:
PCB thickness: 1.5 - 1.6 mm
Stencil thickness: 0.12 - 0.18 mm
3.3 Paste Printing

Preparation of the PCB, i.e. applying solder paste on the soldering pads, is one of the most important steps in the SMT production process. The quality achieved here has a crucial influence on the quality of all further process steps. The soldering result in the subsequent placement and reflow process is determined by the necessary paste volume and thus the degree of solder paste filling during paste application. For optimum filling of the THR holes within the usual process tolerances, two different stencil printing processes have emerged for paste printing, namely the open and closed stencil printing processes.

In the open stencil printing process, a metal or plastic squeegee pushes the roll of solder paste over the stencil. The drawback of this process is that both the pressure and the printing speed can only be varied within a very limited range.

In closed printing systems such as "ProFlow", however, a paste deposit with a variably adjustable pressure travels over the stencil. The advantage: better monitoring of the pressure and printing speed permits paste printing by High-Speed Fine-Pitch Technology as well as controlled filling of the THR hole. A range of stencil variants can be used, depending on the chosen technology, from low-cost standard metal stencils to expensive "exotic" variants such as stepped or double printing stencils.

For an optimum degree of solder paste fill, the following process parameters must be taken into account in the paste printing process:

- stencil thickness: generally 120 to 200 µm
- optimum coordination of the solder paste grain size (20-40 µm) and volume percentage of the flux (max. 50%)
- synchronization of squeegee speed and pressure: a higher degree of fill is achieved with open paste systems by using a lower speed or a flatter angle (i.e. 45° instead of 60°)

Various requirements for the through-hole components are derived from these parameters:

- The housing of the through-hole component must not touch the solder paste, so a stand-off height of at least 0.3 mm is necessary
- The geometry of the pin cross-section, the pin tip, and pin length must be optimized to minimize the volume of paste required.
If the degree of solder paste fill is too low, then it is advisable to overprint to achieve the necessary paste volume that wicks into the hole during the soldering process. Overprinting means that the stencil hole diameter is designed to be larger than the soldering eyelet diameter and the paste is printed over an area larger than the soldering eyelet, and out onto the PCB. This can, of course, cause soiling of the underside of the stencil, which can be removed by a shorter cleaning cycle of the stencil.

Solutions and recommendations from Weidmuller

To achieve the lowest possible paste volume in stencil printing, the SL-SMT pin headers from Weidmuller feature short octagonal pins with chamfered pin ends.

Thanks to the short pin length of only 1.5 mm, when using standard pc boards with a thickness of 1.6 mm, a fill degree of only 90% is adequate for an optimal solder joint shape. Therefore, in most cases, normal single-layer stencils for fine-pitch technology and the standard process parameters for the squeegee speed and pressure can be used during paste printing.

Further advantage: during the placement process, the paste is forced out of the through-hole to a lesser extent.
3 THE SMT PROCESS

FOR MODULES WITH THR COMPONENTS

3.4 The Placement Process

Automatic sequential placement is one of the placement processes that is used most frequently in SMT production. Both pick & place systems and collect & place systems (chip shooter) are used. In the pick & place station, components are picked up one at a time from the component feeder by a vacuum nozzle, and placed onto the PCB.

The following parameters must be observed and contribute to the optimum placement suitability of through-hole components, allowing easy integration into the placement process using standard placement systems:

- minimized weight for highest possible placement speed
- minimized component length for a high rotation and transport speed and thus maximum placement performance
- minimized height to avoid a restriction of the travel height over the PCB and thus to prevent collisions
- minimized tooling effort by using components packaged in high quantities to achieve lower product and tooling costs

This results in the following requirements for through-hole components:

- minimized weight for highest possible placement speed
Changes in length because of water absorption

PA 4.6: 1% => 0.5 mm / 50 mm =>
“50.5” mm (total length for a 10 pole pin header)

LCP: 0.03% => 0.015 mm / 50 mm =>
“50.015” mm (total length for a 10 pole pin header)

Dimension changes because of water absorption for different materials.

Solutions and recommendations from Weidmuller

To avoid quality-reducing dimensional changes, the SL-SMT pin headers from Weidmuller are made of LCP (Liquid Crystal Polymer). LCP is particularly distinguished by very low water absorption, and therefore low instance of changes in length of the component.

The SL-SMT pin headers have a low height thanks to the use of short pins (1.5 mm). This is why the height of the tape-on-reel or tray packaging can also be very low. The result: a large number of packaged units in each reel. The antistatic packaging in line with standards also permits fully automatic placement using commercially available pick & place systems.

Further advantages of the short pins of the SL-SMT are optimized travel height and the precision of the pin end position, thus avoiding collisions between the components and the PCB. And, as the SMT-optimized pin headers from Weidmuller are very light, they also maximize placement performance.
3.5 Reflow Soldering

The second most important step in the SMT production process (after paste printing) is the reflow soldering process. This process is characterized by the melting of an existing solder deposit. During the process, about 50% of the paste volume evaporates. The aim of reflow soldering is to enable soldering of all THR joints in all soldering processes.

In industrial applications, there are three different reflow soldering processes currently used:

- Infrared soldering: exploits heat radiation with natural convection as generated by quartz; or area radiators to heat the module.

- Convection soldering: heats up the module by force convection via the circulation of large volumes of a process gas such as air or nitrogen.

- Vapor phase soldering: uses a condensing saturated vapor phase as the transmission medium which dissipates its condensation heat directly to the surface of the medium.

Each of the three processes calls for a special process temperature curve, which also applies to THR components. As more and more companies are required to comply with emerging lead-free soldering requirements, the demands on the temperature load carrying capacity of components continue to rise.

For use in the reflow soldering process, the following factors or requirements are crucial in regards to THR components:

- Soldering heat resistance: the shape and the functionality of the component must not be damaged by the soldering process. In this respect, all components can be checked with a test method that is described in EN 61760-1. This process permits the measurement of THR component's thermal resistance by means of a simulation. Soldering heat resistance is tested for the highest category A, by dipping the component in a soldering bath. The insulator floats on the surface of the solder and all soldering contacts are immersed. The minimum requirement is 260 °C/10 sec. Moreover, the component's suitability for passing through the soldering process twice (double-sided assembly) must be guaranteed.

- Thermal coefficient of expansion: very long components such as multiple-pin connectors, which are made of materials that exhibit a significantly different thermal response than that of the PCB material, may deform or sag after soldering or during the cooling phase.
To avoid this problem, the thermal characteristics of the component material and those of the PCB material should be as similar as possible. Standard materials such as PBT or PA 6.6, which are used in connectors designed for wave soldering processes, are therefore not suitable for reflow processes.

- Water content:
  only components made of insulators with minimal water absorption prevent bubble formation during the soldering process, and thus prevent modification of the insulator’s surface.

Even if apparently all factors and requirements for the components participating in the reflow soldering process have been met, though, in some cases the desired production quality cannot be achieved. The most frequent sources of faults are:

- Errors in component design: problems can occur when the accessibility of the soldered joints for the thermal transmission medium (i.e. air) is restricted by neighboring components. This can, however, be avoided in advance with an appropriate component design and board layout.

- Errors in PCB design: copper terminal faces that are too large lead to heat sinks on the soldering eyelets.

- Faults of the soldering machine: the melting temperature that is necessary for the solder paste at the soldered joint is not reached due to inadequate efficiency of the reflow soldering machine or of the programmed process profile.

**Solutions and recommendations from Weidmüller**

Weidmüller manufactures long components such as multiple-pin pin headers out of fiberglass reinforced LCP (Liquid Crystal Polymer). This high temperature-resistant, halogen-free insulator has a melting point of 335 °C and therefore exhibits high stability of shape and very good soldering heat resistance. Weidmuller surpasses EN 61760-1 and subjects its terminal elements to 2 x 260 °C/10 sec. for a double passage through the process. The SL-SMT pin headers, made of LCP, even withstand 2 x 290 °C/30 sec. They are therefore suitable for all lead-free soldering processes. The SL-SMT is designed to take customers into the future, meeting all of their processing requirements.

One further advantage of LCP is its extremely low thermal coefficient of expansion. Measured on the SL-SMT pin header from Weidmüller, it amounts to 0.16% at 260 °C = 0.08 mm for a length of 50 mm, for example. It therefore exhibits a thermal response that is similar to that of common PCB base material, for example FR4. Because of this, there is never any bending or insulator deformation after the soldering process.
3.6 Quality Control

Quality control concludes the SMT production process. The goal of this process step is to enable quick and simple assessment, by suitable means, of the quality of the soldered joints of THR components, either with short or long pins. The same rules apply to quality control of the THR technology as to reflow or wave-soldered THT components.

Inspection method:
Quality control during the process can be accomplished several different ways. Currently, the most popular method is optical or X-ray inspection. The optical test evaluates the shape, reflection and color of soldered joints. When done manually, a magnifying glass or a microscope is used and, in the automated process, a computer-controlled camera performs computer-assisted image evaluation. In X-ray inspection, a so-called radiographic evaluation is conducted by means of an automated X-ray microscope. In addition to the in-process inspections, destruction tests are also important. Carried out on batch samples, they serve to check:

- analysis of the degree of filling by means of a cross-section through the soldered joint
- mechanical inspection of the force needed to pull the pins out of the plated through-holes

Standards:
Various standards have come into force in the past few years to enable objective quality control. In addition to acceptance test criteria for the production quality of electronic modules, the assessment criteria based on the IPC-A-610B has been acknowledged the world over. According to IPC-A-610B, the quality requirements for THR soldered joints are subdivided into three categories. Generally, the requirements of category 3 are decisive for high-power electronics in industrial applications. The following values have been defined for the five assessment criteria:

- circumferential wetting of the primary side (i.e. the component side from the component's point of view), the terminal and the sleeve must not fall below 270°
• vertical solder filling must be at least 75%
• circumferential wetting of the secondary side (i.e. underside from the point of view of the component) must not fall below 330°
• solder wetting of the original land face (i.e. residual ring in the case of THR soldered joints) on the primary side is defined as 0%
• solder wetting of the original land face on the secondary side must be at least 75%

With regard to quality control, these standards result in several requirements for through-hole components. It is necessary to take into account that two different designs can be used for the THR process:

• pins shorter than the PCBs thickness
• pins longer than the PCBs thickness (approx. 1.0 – 1.5 mm overhang)

For short pins, the following requirements apply:

• the soldered joints on the primary side of the PCB must be visible
• for components where the pins are under the insulator, the height of the stand-off or the product design must be adequately dimensioned for visual inspection

The following requirements apply for long pins:

• the soldered joints on the secondary side must be visible
• for comparison, visibility of the soldered joints on the primary side is not necessary

During destructive inspections for THR components, demands are also placed on the PCB. The force needed to pull the pins out of the soldered joint is checked to ensure the quality of PCB production:

Required pull-out force:
- for short pins: approx. > 150 N
- for long pins with solder meniscus on the primary and secondary sides: approx. > 220 N

Comparison:
For surface connections with SMD components (gull-wing type leads), the required pull-out force amounts only to about 15 – 20 N.

Solutions and recommendations from Weidmuller

Short pins with a solder meniscus only on the primary side attain a comparably high stability on the board which is otherwise only possible when using the conventional THT. Hence THR components - like Weidmüllers SL-SMT - fitted with short THR pins, have a considerable advantage over real surface-mount connections. They achieve approximately 10x greater tensile strength than standard SMDs.

The special design of the SL-SMT pin headers from Weidmuller permits visual inspection of a PCBs primary side, even in the case of variants with an 180° outgoing direction, thus creating the prerequisite for quality control during the process.

However, it is important that the view of the control side of the components is not concealed by other tall components on the PCB. Weidmuller recommends therefore, that designers take into consideration the placement possibilities of all components on the primary, or quality inspection side, to ensure proper visibility of the soldered joints.
The new standard in the SMT process of printed circuit board assembly comes from Weidmüller: the SL-SMT pin headers. Every SL-SMT pin header incorporates Weidmüller’s many years of experience in the SMT production process and the concentrated know-how about the demands it places on THR components. This is why SL-SMT pin headers not only fulfill these requirements, they also offer you the optimum workflow in automatic, two-sided PCB assembly, thus reducing your production costs by up to 30%.

SL-SMT are Reliable!

SL-SMT are made of high-grade LCP (Liquid Crystal Polymer).

Your advantages:
• maximum pitch integrity and dimensional stability for the automated placement process
• thermal response similar to that of common PCB materials to prevent bowing or sagging
• temperature-resistant for all common soldering processes
• maximum suitability for the future: halogen-free, capable of being recycled, and suitable for future lead-free soldering requirements
• maximum design flexibility: available in versions from 2 to 24 pins

SL-SMT are Extraordinary!

SL-SMT use short pins with a 1.5 mm pin length.

Your advantages:
• double-sided SMT placement with a PCB thickness ≥ 1.5 mm
• required paste volume is minimized
• simplified paste printing process
• easier outgassing of the flux during the soldering process
• guaranteed attainment of the necessary temperature in the soldering process thanks to a reduced metal mass from short pins
• lower component height avoids collisions in the placement process
• reduced packaging height permits the use of standard feeders
• optimized quantity on each packaged reel
SL-SMT are Innovative!

SL-SMT pin headers feature an integrated paste space with stand-off height of at least 0.3 mm.

Your advantages:
• very simple stencil design for standard layout or for overprinting
• design-compatible with standard pin headers because there is no additional stand-off

SL-SMT are Application-optimized!

SL-SMT feature octagonal pins which are specially designed at the ends.

Your advantages:
• minimization of the placement force needed for setting the pins into the paste by the placement machines

SL-SMT are Intelligent!

SL-SMT are available in several intelligent packaging designs - suitable for several pin number variants.

Your advantages:
• only one feeder width is necessary for several pole numbers
• compatible with all common pick & place placement systems
• lowest possible packaging height for a higher quantity per reel and higher placement efficiency
• antistatic ESD materials for problem-free placement

Optimized pin tip

Pin header with 180° outgoing direction and open ends: SL-SMT 5.08/8/180

Product specific packaging design