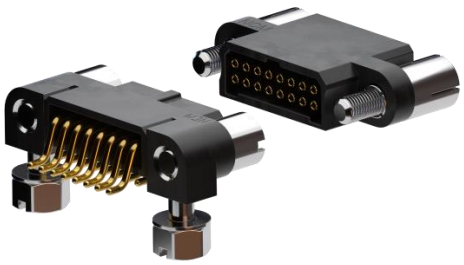
White paper

SOLDERING GUIDE SMT

Table of contents



Introduction

→ For several decades, **surface-mount technology** (SMT) has revolutionized the electronics industry by enabling the assembly of components directly onto the surface of printed circuit boards (PCBs). This method offers numerous advantages, including reduced footprint, increased automation of the assembly process, and lower production costs.

The purpose of this guide is to provide detailed information on the soldering process for surface-mounted components, with a focus on best practices to ensure robust and reliable electrical connections. We will also cover the acceptance criteria defined by the IPC-A-610 standard, as well as the quality control techniques to follow in order to ensure the compliance of electronic assemblies.

Detailed Steps of the Surface-Mount Connector Reflow Soldering Process:

Focus on the pick and place

This is an automated method in which a robot places electronic components onto a printed circuit board.

- Preparation of the Printed Circuit Board (PCB)
- Application of Solder Paste
- Placement of Components
- Reflow Soldering in the Reflow Oven
- Cooling and Solidification of Solder
- De-fluxing

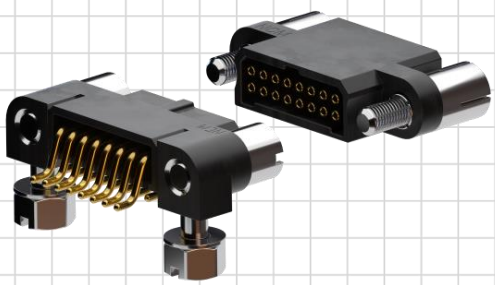
List of advantages of Surface-Mount Technology (SMT) Components compared to Through-Hole Technology (THT) components on PCBs :

Advantages **SMT / THT**	Advantages **THT / SMT**
▪ Low profile	▪ Prototype manufacturing
▪ Automation of the assembly process	▪ Components not available in SMT
▪ Lower costs	▪ Increased mechanical strength
▪ Ability to use both sides of the PCB	

In the SMT versions, Nicomatic offers several mounting options:

- Board-to-board
- Board-to-wire

Fastenings (On-board)



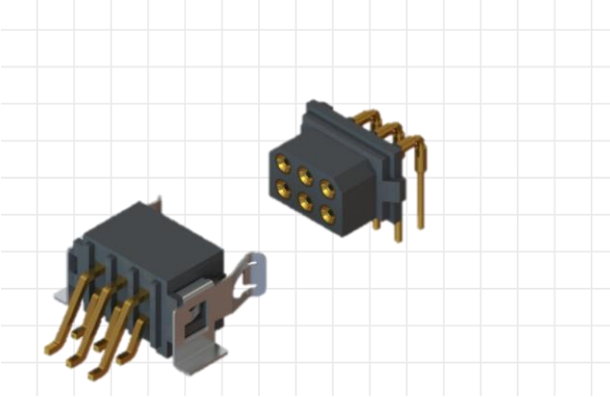
Example: Locking fastenings CMM

Locking fastenings :

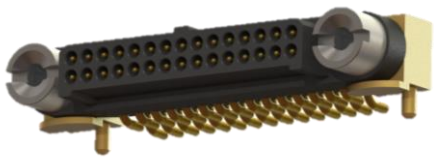
Locking fastenings rely on a screw-nut system, ensuring excellent resistance to tension, compression, and shear. However, their tightening requires manual intervention by the technician.

Soldered fastenings :

These fastenings have the advantage of being soldered simultaneously with the contacts, thus avoiding interruptions in the production line for tightening, as previously mentioned.



Example: Soldered fastenings CMM



Example: Pin in Paste fastenings CMM

The pin-in-paste process refers to the technique of inserting pins into the holes of a printed circuit board and applying solder paste before heating to ensure a secure hold. It relies on the reflow method.

Pin in paste fastenings :

This fastening addresses the issues of tightening fastenings and disrupting the automatic soldering line. However, unlike the soldered version, it offers significantly higher shear resistance, making it a preferable option for a 90° wired version or any other configuration that exerts a transverse force on the fastening.

However, it has lower resistance to axial forces compared to locking fastenings (in tension).

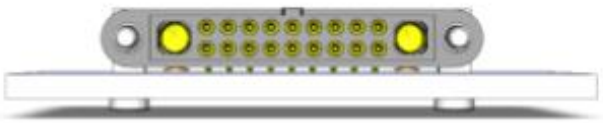
Criteria	Locked	Soldered	Pin in Paste
Shear strength	+++	+	++
Tensile strength	+++	++	++
Soldering automation	+	+++	+++
Repairability	+++	+	+
Positioning accuracy	+++	+	++
Soldering throughput	+	+++	++

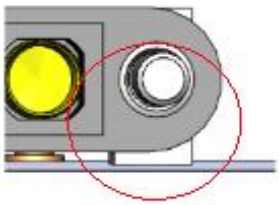
Soldering

Soldering techniques (SMT)

One method for applications requiring high throughput is reflow soldering in an oven. To implement this technique, components are placed on the PCB, and then the assembly is heated in an oven at controlled temperatures, ensuring precise solder melting.

Once the soldering is completed, a de-fluxing process is necessary to remove residues, as these can degrade the solder joint. To facilitate this, Nicomatic offers a washing foot in the design of its products, which slightly elevates the connector to ease the cleaning of printed circuit boards.



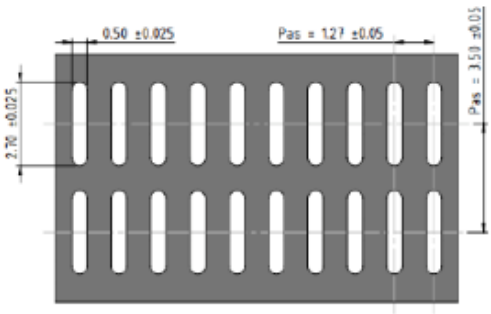


Soldering

Stencils

To achieve high throughput, it is essential to use stencils to deposit solder paste precisely. They help control the amount of material to prevent excess solder, which can lead to short circuits.

They ensure the repeatability and reliability of the process.



These stencils are tailored to the recommended footprint available on the Nicomatic configurator, to facilitate the design of printed circuit boards.

What's a footprint

The footprint is the layout of the various landing areas to be created on the PCB for the connector contacts.

https://configurator.nicomatic.com/

To comply with the IPC-A-610 standard, it is advisable to use stepped stencils in certain cases, as they allow for consistent application of solder paste.

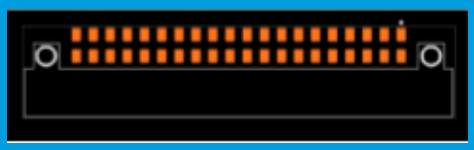
A distance between the pads is also necessary to maintain the minimum electrical isolation distance between the contacts.

Recommendations

PCB footprint

A PCB layout, or "printed circuit board layout," refers to the design plan of a printed circuit board, which involves optimally placing and connecting electronic components on a PCB.
This process considers aspects such as component placement, routing of connection traces, and electrical and mechanical design considerations to ensure the proper functioning of the electronic circuit.

The recommended footprint by Nicomatic is provided on the client drawings according to the connector and the contact version (R or T) ordered.



This facilitates the PCB layout design by considering the connector configuration.

Positioning

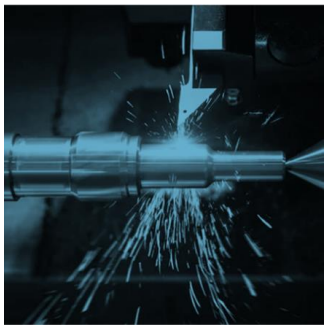
The positioning of components prior to soldering involves several steps:

Actions	Description
Preparation of the PCB	Ensure that the PCB is clean and free of contaminants. The PCB surface should be prepared to accommodate the component.
Application of the solder paste	Use the previously provided soldering stencils to accurately apply solder paste to the PCB pads.
Component placement	Carefully place the SMT component onto the designated positions on the PCB. Ensure that the component's pins align with the PCB pads.
Precise alignment	Use alignment tools, such as automated vision systems or other equipment, to ensure precise positioning of the component before soldering.
Fastenings	See description below

Recommendations

Tightening fasteners

We recommend securing the connectors before the soldering operation to avoid any damage to the solder joint during tightening, such as cracks or, in the worst case, solder detachment.



→ We do not guarantee the signal integrity if the locking is done after soldering.

It is particularly important to follow our recommendations, as we are dealing with surface-mount components, which are at an increased risk of misalignment before soldering.

Soldered fasteners

Nicomatic offers two types of soldered fasteners in its range, available for the CMM 100 and 200. These latches can be found on page 22 of the CMM catalog.

This catalog is available at the following link:

https://www.nicomatic.com/family/3

IPC-A-610 Standard

The IPC-A-610 standard is often regarded as a benchmark in the electronics industry. It defines the acceptance criteria for assembled electronic products. As it is a recommendation rather than a standard, it is not legally binding and does not form part of mandatory legal requirements.

IPC Classes

However, IPC-A-610 is widely used and respected in the industry for establishing quality criteria and acceptance standards.
It therefore serves as an important reference for manufacturers, suppliers, and customers in the electronics industry.

	IPC Class 1	IPC Class 2	IPC Class 3
Category	General electronics	Dedicated service electronics	Electronics for harsh or risky environment
Life cycle	Short	Long	Very long
Quality	Low	High	Infallible
Example	Toys, lamps	Computers, server racks	Aerospace, Defense , medical

Inspection techniques

→ Inspection of solder paste deposition on the PCB

This step is crucial to ensure the correct amount of solder paste is applied. Excessive paste can cause short circuits, while insufficient paste may compromise proper contact between the component and the board.

It is also important to monitor the area covered by the solder paste to ensure it is properly centered on the board, avoiding the two issues mentioned earlier.

→ Visual inspection after pick & place

The goal here is to ensure that no defects occurred during the Pick & Place process. This involves verifying that components are properly positioned on the landing pads, as components that are too close together pose a risk of short circuits. If such issues arise, it is easier and less costly to address them before the reflow soldering phase.

→ Visual inspection after reflow soldering

Once reflow soldering is complete, it is necessary to inspect the overall condition of the solder joints. Several techniques can be employed depending on the available tools. Initially, a visual inspection can detect potential microcracks or solder bridges between components. Optical microscope inspection may be required, with a magnification level recommended by the IPC-A-610 standard. However, if doubts remain, X-ray inspection or scanning electron microscopy can be used for a more detailed analysis.

If the solder joints are partially visible, a visual inspection of the visible areas is necessary. However, if the solder is completely concealed, the IPC-A-610 standard recommends performing an inspection of the process before soldering or utilizing one of the previously mentioned non-destructive inspection methods, such as X-ray inspection.

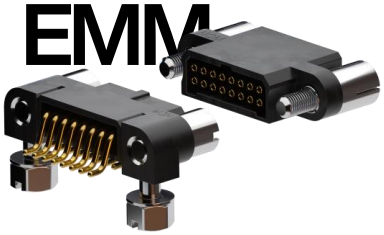
Compatible Nicomatic products

Nicomatic product types :

CMM

Ultra-compact rectangular micro-connector with a 2 mm pitch. Meeting or exceeding the MIL-DTL-55302F and BS-9525-F0033 standards.

EMM



1.27 mm pitch micro-connector providing a 20% space savings compared to a standard micro-D. Meeting or exceeding the MIL-83513 standard.

DMM

Connector derived from the CMM series. Complies with the performance requirements of MIL-DTL-83513G. Available in a wide range of configurations.

This user guide applies, among other things, to various products in the Nicomatic range, particularly connectors equipped with SMT contacts, such as the R and T contacts. The R contact corresponds to a 90-degree configuration, while the T contact is a straight contact configuration.

Since 1976, Nicomatic has continuously developed its expertise in micro-connectors designed for harsh environments. Numerous connector configurations are available for the **CMM**, **EMM**, and **DMM** ranges, allowing for optimal system integration.

Contact us!

Contact us | Nicomatic

Quality Report

Test report on website :

https://configurator.nicomatic.com/fr/labreports

- Solderability report
- Solder heat resistance
- Gold plating study

Self-declaration available upon request

Glossary

→ SMT (Surface Mount Technology)

It is a method of mounting electronic components directly onto the surface of a PCB rather than inserting them into pre-drilled holes.

→ THT (Through-Hole Technology)

Through-hole technology is a method of mounting electronic components where the pins are inserted into pre-drilled holes in a PCB.

→ Contacts R

90° SMT mid-frequency contact

→ Contacts T

Straight SMT mid-frequency contact (Available in an extended version)

→ CMM (100 et 200)

2mm pitch micro-connector per MIL-DTL-55302F standard with plastic housing.

→ EMM

1.27mm pitch micro-connector per MIL-DTL-83513 standard with plastic housing.

→ IPC-A-610

Industry standard defining acceptance criteria for visual inspection of printed circuit boards and electronic assemblies.

→ PCB Layout

Layout plan of electronic components and conductive traces on a printed circuit board (PCB).

→ Footprint

The three-dimensional design of an electronic component that specifies its physical dimensions and the arrangement of its pins or connection pads on a printed circuit board (PCB).

Glossary

→ PCB (Printed circuit Board)

Insulating board on which electronic circuits are printed or etched, providing a platform for mounting and connecting electronic components.

→ Flux application

The process of applying flux to electronic components or soldering surfaces on a PCB to improve wettability and solder joint quality.

→ Flux removal

The process of removing flux residues and other contaminants from electronic components and the PCB after the soldering process, typically carried out using chemical or mechanical cleaning methods.

→ Pads

Metalized area on a PCB where electronic components are soldered to ensure an electrical connection.

→ Latch

A mechanical device that holds one or more connectors in a fixed position to ensure a secure connection between electronic components and the PCB.

→ Pick and place

An automated process used in electronic manufacturing to pick electronic components from their storage location and place them precisely onto the corresponding pads on the PCBs.