SCP1000 Assembly Instructions

SCP1000 Series
Absolute pressure sensor

TABLE OF CONTENTS

1 Objective

This document describes guidelines for Printed Wiring Board (PWB) design and assembly of the SCP1000 Pressure Component. It aims to help customers achieve the optimum soldering process.

2 VTI's Molded Interconnection Device Package (MID)

The housing of the SCP1000 component is relatively new packaging concept, which is called a molded interconnection device (MID), which is a modified Quad Flat-pack No-lead package (QFN). The MID housing is leadless component as well as an QFN package. The SCP1000 component is a surface mount plastic package with leads located at the bottom of the package, see Figure 1. In Figure 2 cross sectional pictures are presented. They illustrate how the component leads are located in the SCP1000 housing.

Figure 1. 3D view of the SCP1000 component. The SCP1000 component leads are located at the bottom of the package.

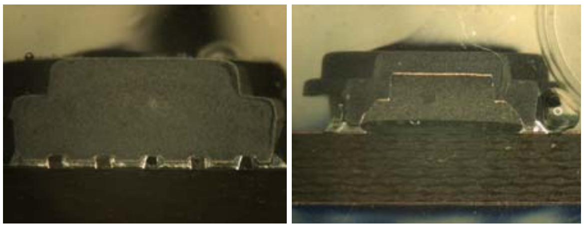


Figure 2. SCP1000 solder pads are located at the bottom of the package.

MID-technology offers a variety of benefits including reduced lead inductance, a small sized "near chip scale" footprint, thin profile, and low weight. The MID component is an injection molded plastic substrate, and the conductive circuit pattern is manufactured on its surface. The SCP1000 component housing is manufactured by the LDS (Laser Direct Structuring) MID-process. The process flow of LDS - MID package is presented in Figure 3.

1. Molding of package

2. Laser structuring of areas to be metallized

3. Cu-plating of lasered areas

4. Final plating over copper

Figure 3. Manufacturing process for the LDS – Molded Interconnect Device.

The solder pads of a MID package are rougher than the pads of traditional lead frame packages. A photo of a MID package solder pad is presented in Figure 4. The rougher surface of the solder pad can produce more voids in the solder joints, which is also observed in guidelines of this technical note.

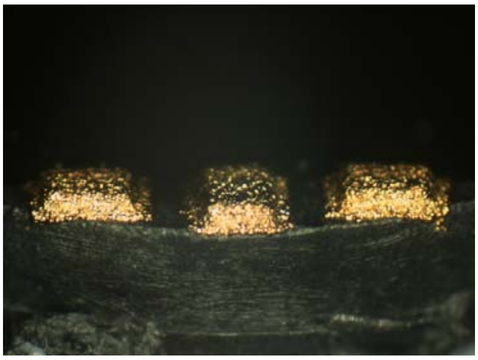


Figure 4. Photo of a MID solder pad. Surface is rougher than with traditional SMD components.

Because the MID platform represents the latest in surface mount packaging technology, it is important that the design of the printed wiring board, as well as the assembly process, follows the suggested guidelines outlined in this document.

It should be emphasized that these guidelines are general in nature and should only be considered a starting point. The user must employ their actual experiences and development efforts to optimize designs and processes for their manufacturing practices and the needs of varying end-use applications.

3 SCP1000 MID Package Outline and Dimensions

The outline and dimensions for the SCP1000 MID package are presented in Figure 5.

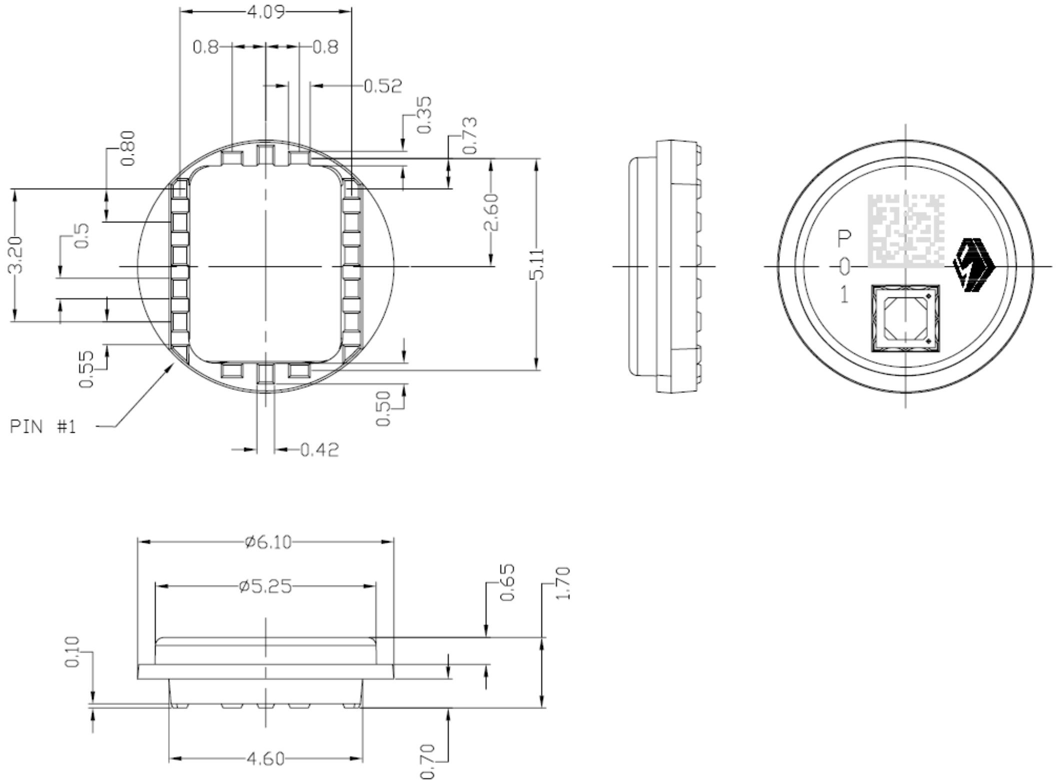


Figure 5. Outline and dimensions for the SCP1000 component.

4 Tape and Reel Specifications

- Carrier Tape width 12.00 +0.30 mm / -0.10 mm
- Cover Tape, width 9.3 mm
- Polycarbonate Reel, 330mm diameter

Note: Tape&Reel materials are suitable for baking for 24 hours at 125°C

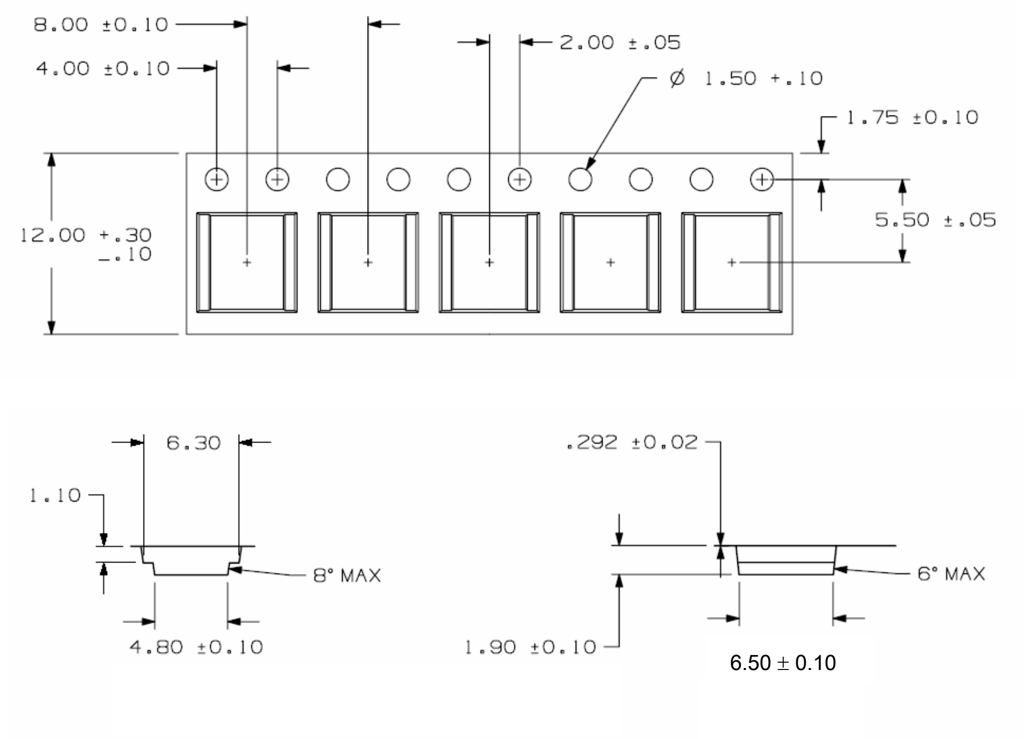
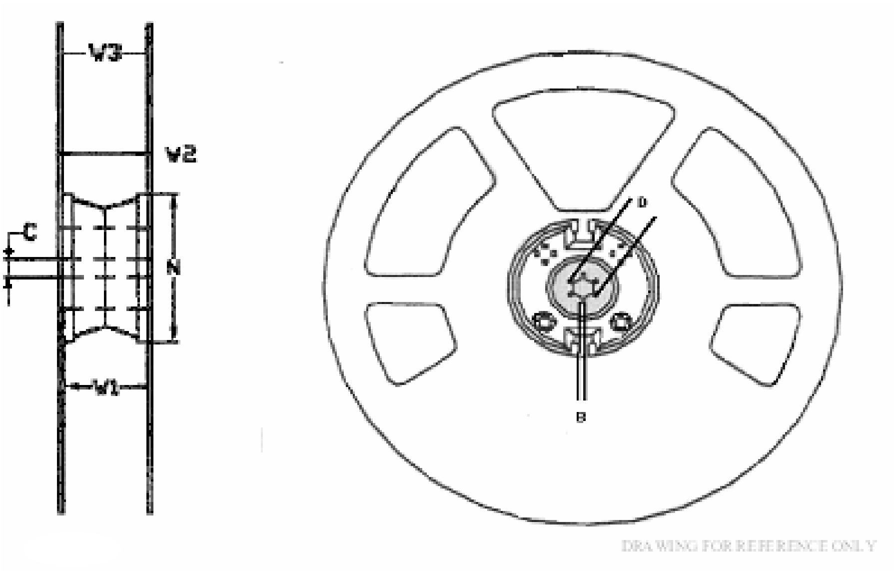


Figure 6. Tape dimensions [mm].



Diameter	B (min)	C	D (min)	N Hub	W1	W2 (max)	W3 (min)	W3 (max)
330	1.5	13	20.2	100	12.4	18.4	11.9	15.4
		±0.2		±1	+2.0			

Figure 7. Reel dimensions in mm.

4.1 Method of Taping/Polarity and Orientation

The polarity of the part on tape looks as follows:

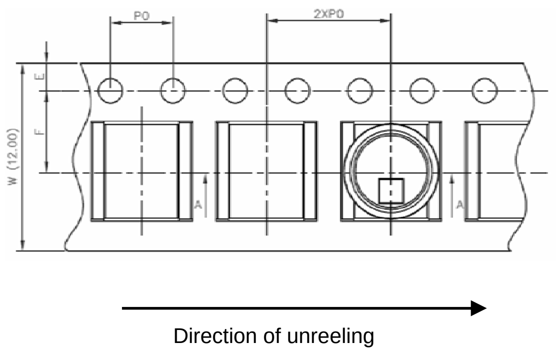


Figure 8. SCP1000's orientation on a tape.

5 Printed Wiring Board (PWB) Level Guidelines

5.1 Land Pad Design

Two types of land patterns are used for surface mount packages:
1) Non-Solder Mask Defined Pads (NSMD)
2) Solder Mask Defined Pads (SMD).

NSMD pads have an opening which is larger than the pad itself, and SMD pads have a solder mask opening that is smaller than the metal pad on PWB. NSMD is preferred since the copper etching process has tighter process control compared to the solder mask process. The specific PWB land pattern for the SCP1000 MID component is presented in Figure 9 by the black lines. In the same figure the outline of the component can be seen by the red lines.

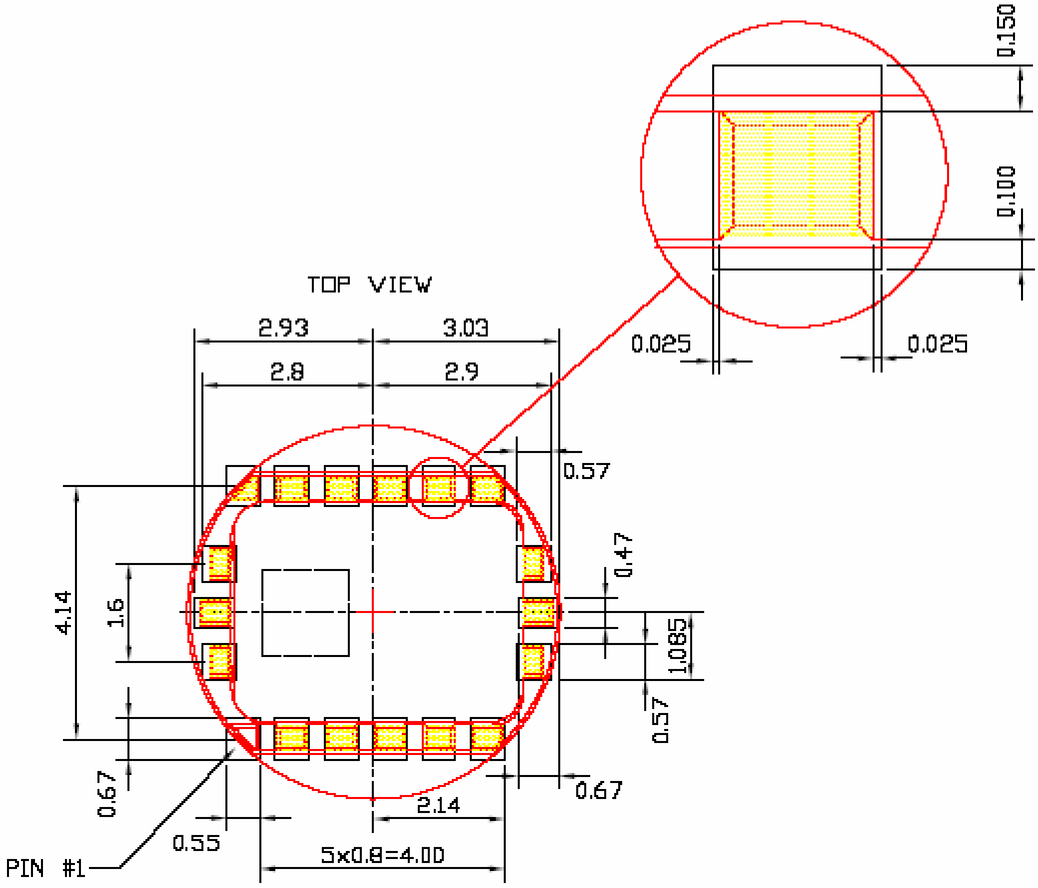


Figure 9. Land pattern for the SCP1000. The yellow colored area with red outer lines illustrates the SCP1000 component pads and the black lines are the land pads on the PWB.

VTI Technologies' pressure sensors can be soldered on commonly used substrates, e.g. FR-4, ceramic, flex-print etc. The pad metallization should be solder wettable in order to assure good quality solder joints. Generally used circuit board finishes for fine pitch SMD soldering are Ag, NiAu, OSP and tin.

5.2 Solder Paste

Lead-free SAC (tin-silver-copper) solder should be used for SCP1000 soldering. The solder paste composition should be near eutectic. The melting point of lead-free SAC solder can vary between 217–221°C, depending on the composition of the solder alloy. A no-clean solder paste is recommended, since cleaning under the SCP1000 MID-package would be difficult. The solder paste must be suitable for printing it through the stencil aperture dimensions. Type 3 paste is recommended (grain size 25-45µm).

5.3 Stencil

The solder paste is applied onto the PWB by using stencil printing. The stencil thickness and aperture determines the precise volume of solder paste deposited onto the land pattern. Stencil alignment accuracy and consistent solder volume transfer are important parameters for achieving uniform reflow soldering results. Too much solder paste can cause bridging and too little solder paste can cause insufficient wetting. Generally the stencil thickness needs to be matched to the needs of all components on the PWB.

For the SCP1000 MID-package the recommended stencil thickness is 5mils (127µm).

Stencil apertures in general should be 1:1 to PWB pad sizes, or stencil apertures could even be reduced by 1mil (25.4µm) from all sides in regard to the PWB land pad size. This reduction of aperture size can reduce bridging between solder joints.

5.4 Paste Printing

The paste printing speed should be adjusted according to the solder paste specifications. It is recommended that care be taken during paste printing in order to ensure correct paste amount, shape, position, and other printing characteristics. Neglecting any of these can cause open solder joints, bridging, solder balling, or other unwanted soldering results.

5.5 Component Picking

Since the SCP1000 is a pressure sensor device and the pressure port is located on the top of the package, there are some limitations to component picking.

The SCP1000 can be picked from the carrier tape using either vacuum assist or mechanical type pick heads. Typically a vacuum nozzle is used. The vacuum nozzle configuration should be designed so as to avoid the vacuum port being located directly over the pressure port on the SCP1000 package. Pick up nozzles are available in various sizes and shapes to suit a variety of different component geometries. It is recommended to that different pick up nozzles are tested to find the best one suited for the purpose. In some cases, it may be necessary to manufacture a specially designed nozzle to achieve the optimal speed for the pick & place operation.

Figure 10 indicates the area where the pressure port is located on the SCP1000 MID component. The nozzle must not touch the marked area to avoid damage to the pressure component and to the pick up nozzle itself.

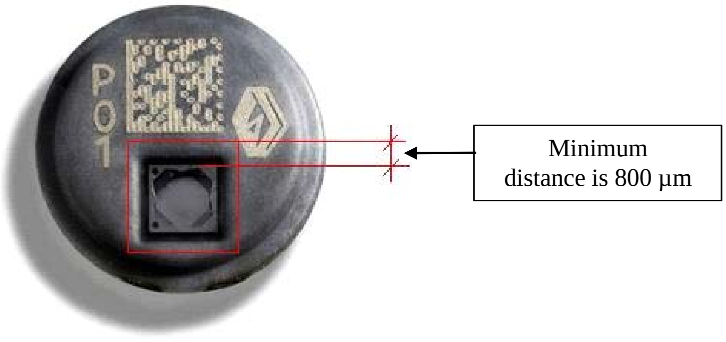


Figure 10. The pick up nozzle must not touch the inner area of the red square. Minimum distances to be kept free is 800 µm on all four sides, measured from the sensor chip edge.

5.6 Component Placement

SCP1000 MID packages must be placed onto PWB accurately according to their geometry. Positioning the packages manually is not recommended. Placement should be done with modern automatic component pick & place machinery using vision systems.

Recognition of the packages automatically by a vision system enables correct centering of packages. Because of the round shape of the SCP1000 component and special shape of solder pads on the other end of the package, there are some recommendations to help in the automatic Pattern Recognition System (PRS).

Figure 11 shows a red rectangle on the back side of the package. That rectangle indicates the solder pads, which are not recommended to be used in pattern recognition process, since their shape might cause some difficulty.

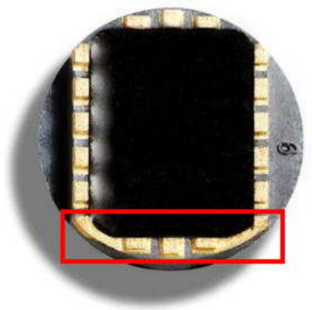


Figure 11. The solder pads, which are inside the red rectangle, are not recommended to be used in pattern recognition of the component.

5.7 Reflow Soldering

A forced convection reflow oven is recommended to be used for soldering SCP1000 MID components. IR-based reflow ovens are not generally suitable for lead-free soldering. **Figure 12** presents a general forced convection reflow solder profile and it also shows the typical phases of a reflow process.

The reflow profile used for soldering the SCP1000 MID package should follow the solder paste manufacturer's specifications and recommended profile. The typical ramp-up rate is 3°C/second max. For lead-free soldering with SnAgCu solder, the pre-heat temperature zone is ~ 150°C-200°C and should last for ~ 60-180 seconds.
Time above the liquidus temperature (~217°C) should be ~ 60-150 seconds. The reflow peak max. temperature should not exceed 260°C. Temperature within actual peak temperature is typically ~20-40 seconds. The ramp down rate should be 6°C/second max.

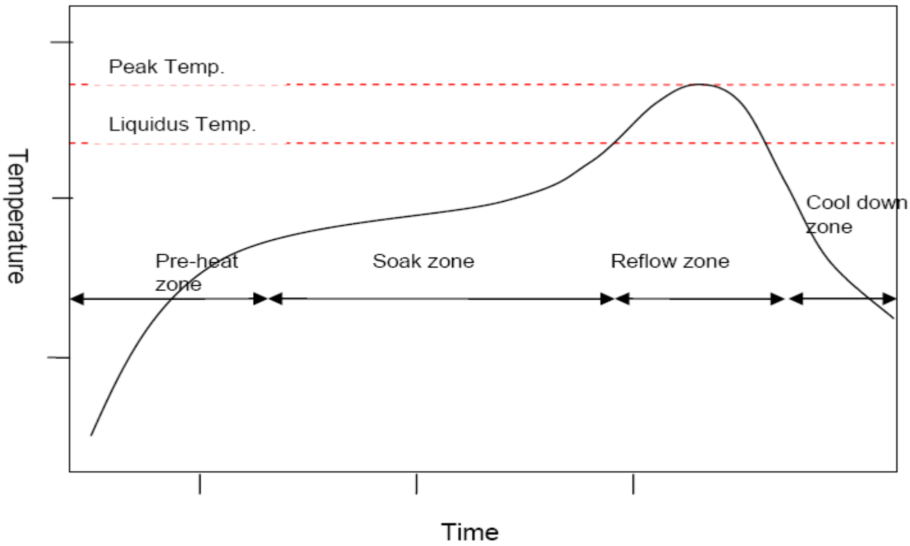


Figure 12. Typical convection reflow soldering phases and profile.

The process window for lead-free soldering is narrower than for traditional eutectic SnPb solders. Thus, caution has to be taken care when adjusting the reflow profiles. The reflow profile should be measured using a thermo-couple measurement system. It is recommended that at least three thermo-couples are used, depending on the application. As a general guide, one thermo-couple should be placed under a component having the largest thermal mass, one next to the smallest component, one should be in contact with pressure sensor's solder joint, and others to the appropriate spots on a circuit board, e.g. corner, center, bottom of the board etc. The reflow profile should be adjusted according to the measured data so that each solder joint experiences an optimal reflow profile. The temperature gradient should be as small as possible across the circuit board. Extreme caution has to be taken if the circuit board contains components with vastly different thermal masses.

5.8 Moisture sensitivity level (MSL) classification

The Moisture Sensitivity Level of the SCP1000 component is Level 3 according to the IPC/JEDEC J-STD-020C. The part is delivered in a dry pack. The manufacturing floor time (out of bag) at the customer’s end is 168 hours. Maximum soldering peak temperature for the SCP1000 is 260°C/40sec, measured from the package body.

5.9 Inspection

If optical and visual inspection of solder pads is done, need to be considered, that the component leads as well as solder pads are located on the bottom of the SCP1000 component. Figure 13 show photos of how the soldered component on PWB looks from different directions.

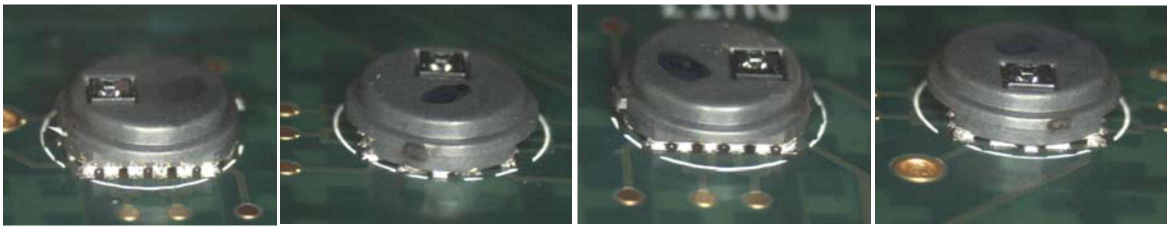


Figure 13. Solder pads, component leads and the solder joints are located under the component itself.

X-ray inspection is the recommended inspection method for checking solder bridges and short-circuits between solder pads. The X-ray inspection systems vary from manual to fully automated optical inspection systems. Typically X- ray is used for sample based process control. X-ray inspection is also used to establish and optimize the component assembly process parameters.

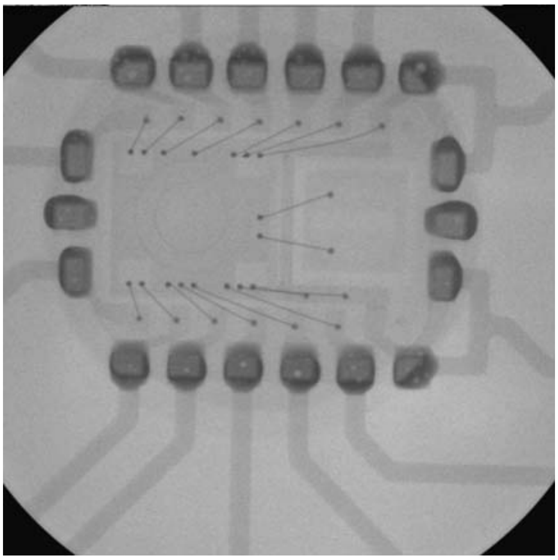


Figure 14. X-ray image of a well soldered SCP1000 component.

Cross-sectional analysis is also an approved method to inspect how well solder has wetted the pads of component. It is important to notice that wetting of solder is different on the inner and outer surfaces of the solder pad of the SCP1000 component. The solder wetting is different, because the inner surface of solder pad is covered with coating material, which overlaps the pad metallization. Cross-sectional analysis is not used for production inspection, but if desired, it can be used to establish and optimize the component assembly process parameters. Cross-sectioning is a destructive inspection method.

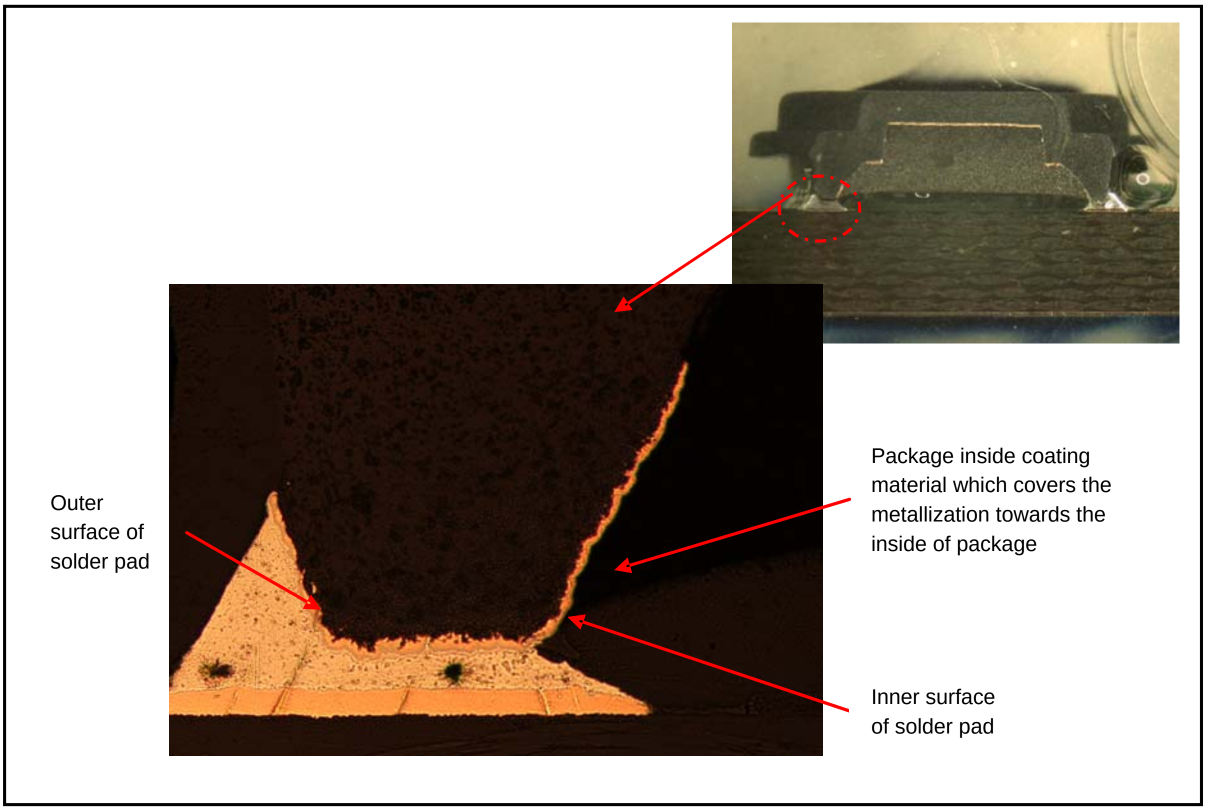


Figure 15. Wetting of solder is different on the inner and outer surface of the solder pad.

6 Rework Guidelines

There are several rework systems on the market. Some will heat solder joints directly from the sides of component package, while others will direct heat on the top of component. Occasionally, very rough rework methods are used such as hand placement and heating with a soldering iron. However, these rough methods are not suitable for VTI's MID components. The heat flow of hot-air convection should be directed at the edge and under the component body directly to the solder joints and component pad areas. The package has a limited thermal conductivity and thus, a horizontal flow method, a conduction method, or a conduction heating from the top of the package to the solder joints should not be used.

Prior to the rework process, PWBs and components should be free from moisture. If necessary, baking and drying processes should be performed.

The reflow profile for the component removal should be similar to the initial reflow. A carefully adjusted reflow profile is essential for the successful rework operation. A proper reflow profile should be measured with thermo-couples. During the reflow profiling, at least the solder joints, the top of the component, and the bottom of the PWB should be monitored. If there are other components near the component to be reworked, the temperatures of those should also be monitored. The bottom side heating of the PWB is recommended in order to reduce the PWB warpage during the rework operation.

Once the solder joints have been fully melted, the component can be carefully removed from the PWB. The common rework methods for the component removal can be used, i.e. a vacuum nozzle etc. It is absolutely necessary to ensure the complete melting of the solder joints before the component lifting. If the solder joints are not fully melted before the lifting operation, pad damage may occur on the PWB and the component.

After the component is removed, the pad areas of the PWB should be cleaned using common rework methods. These include the applying of a rework flux, an excess solder and flux removal using a vacuum solder removal tool or a solder wick and a soldering iron with a wide chisel tip, and the cleaning of solder pads with alcohol and brush. The PWB cleaning process should be performed gently as too high force or a scrubbing motion can cause pad lifting and trace damage.

If the component reuse is desired, it should be carefully inspected for a potential damage, solder residues should be removed, and the pads should be cleaned. Prior to the placement of a replacement component on to the reworked PWB, a solder paste should be applied on the cleaned PWB pads. Suitable methods for applying the solder paste are the usage of a micro-stencil or the dispensing of the solder paste. If the micro-stencil is used, it should be cleaned after each paste application to prevent clogging. For the solder paste and the stencil, the same guidelines as for the initial reflow process should be used.

The accurate alignment of the component is an important process step thought the surface tension of the solder during the reflow step will help with the self-alignment. The use of a split-vision alignment system is recommended to ensure the precise alignment of the component to the PWB. The "Z" placement force should be carefully controlled in order to prevent the solder bridging.

The same reflow profile, as for the component removal, can be used for the re-attachment of the replacement component as long as all solder joints will fully melt and properly wet the contact areas. Otherwise, a new and proper reflow profile must be developed and measured from the solder joints. The reworked PWBs should be allowed to cool to the room temperature. The PWBs and the components should be inspected after the rework process for possible defects. The use of X-ray inspection techniques can be used to verify the success of the rework process.

It should be noted, that the performance and the reliability of the reworked component may have decreased due to the rework operation.

7 Hand Soldering Guidelines

Hand soldering of the SCP1000 MID package is not recommended. For proto-uses of the SCP1000 component, VTI can provide pre-assembled component on a PWB. For more information on this, please find the document *TN38_SCP1000_Test_PWB_Pin_Order*. This can be found from the Internet at www.vti.fi.

8 Environmental Aspects

VTI Technologies respects environmental values and thus its MID packaged pressure sensors are lead-free and RoHS compatible. VTI Technologies' sensors should be soldered with lead-free solders in order to guarantee full RoHS compatibility.

9 Effect of reflow soldering

Reflow soldering may cause reversible pressure offset shift due to thermal stress induced to the component. The amount of shift is proportional to the temperature peak in soldering. Typical graph when using recommended maximum reflow temperature is presented in Figure 16 below.

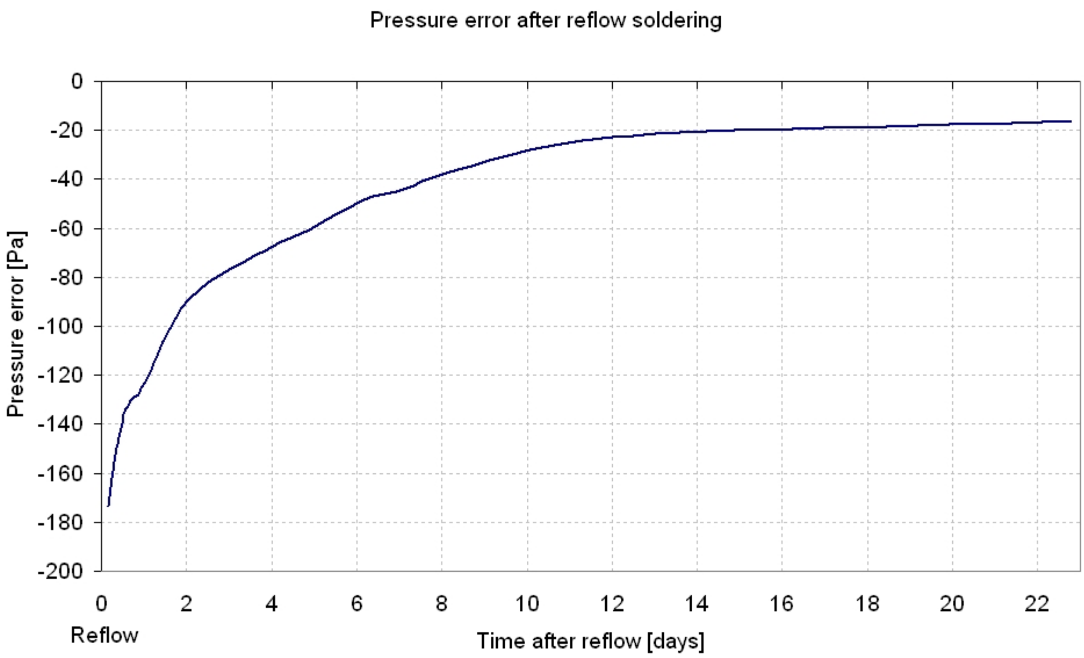


Figure 16. Typical reflow effect on SCP1000 pressure offset.

10 References

JEDEC / Electronic Industries Alliance, Inc. Moisture/ReflowSensitivity Classification for Non-Hermetic Solid State Surface Mount Devices (J-STD-020C).

11 Document Revision History

Version	Date	Change Description
0.01	08.05.2006	This document replaces the following SCP1000 document: - "TN37_SCP1000-D01_D02_Solder_Instructions"
0.02	22.05.2006	Land pattern for the SCP1000 picture update All mm units -> µm units Reflow Soldering / Inspection: text refining
0.03	04.07.2006	Tape&Reel and MSL classification chapters added
0.04	27.09.2006	Rework guidelines added
0.05	10.11.2006	Effect of reflow soldering added