VTI TECHNOLOGIES

Technical Note 53

Assembly Instructions for Dual Flat Lead Package (DFL)

TABLE OF CONTENTS

1 Objective

This document provides general guidelines for Printed Circuit Board (PCB) design and assembly of VTI's Dual Flat Lead Package (referred as DFL). It should be emphasized that this document serves only as a design guideline to help develop the optimal assembly conditions. It is essential that users also use their own manufacturing practices and experience to be able to fulfill the needs of varying end-use applications.

2 VTI'S Dual Flat Lead Package (DFL)

The DFL package is a surface mounted plastic package, with landpads on two sides of the package. The package consists of a premolded plastic housing, with a copper based lead frame having perimeter land pads on the bottom of the package to provide electrical contact to the PCB. A metal lid is attached to the top of the package. The DFL package is presented in Figure 1.

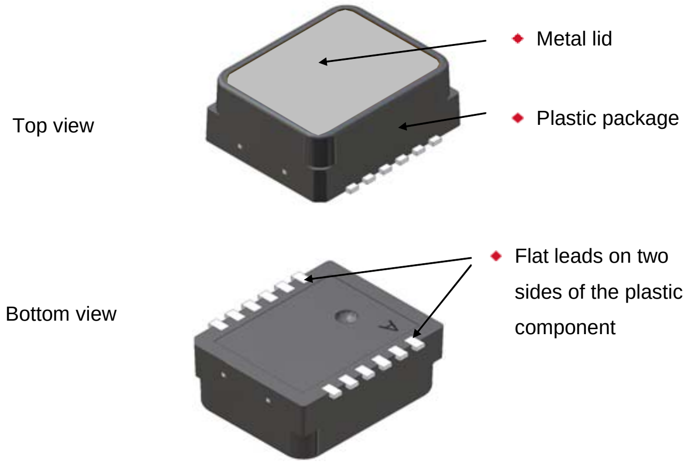


Figure 1: Three dimensional views of the Dual Flat Lead package.

3 DFL Package Outline and Dimensions

The outline and dimensions for the 12 lead DFL package are presented in Figure 2.

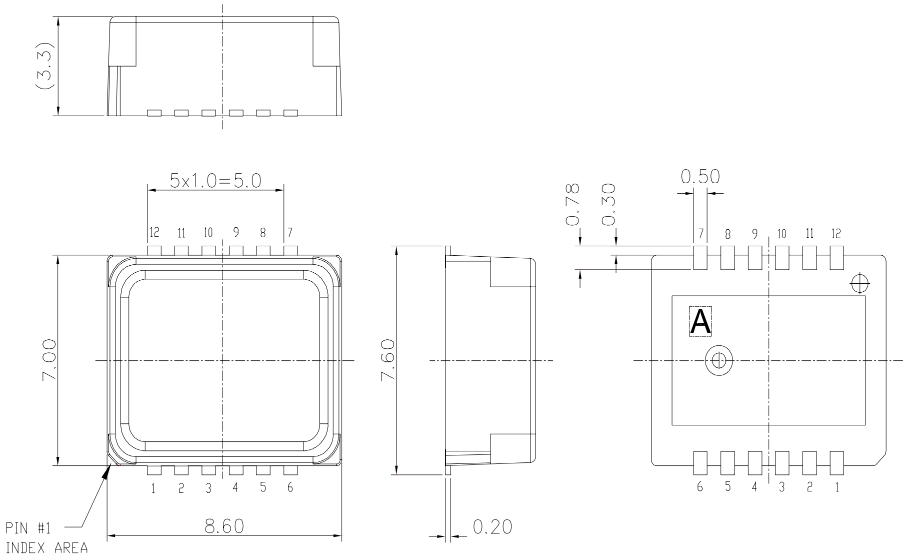


Figure 2: Outline and dimensions for 12 lead DFL package.

4 Tape and reel specifications

Packing tape dimensions are presented in figure 3. The unreeling direction and component polarity on tape are presented in figure 4.

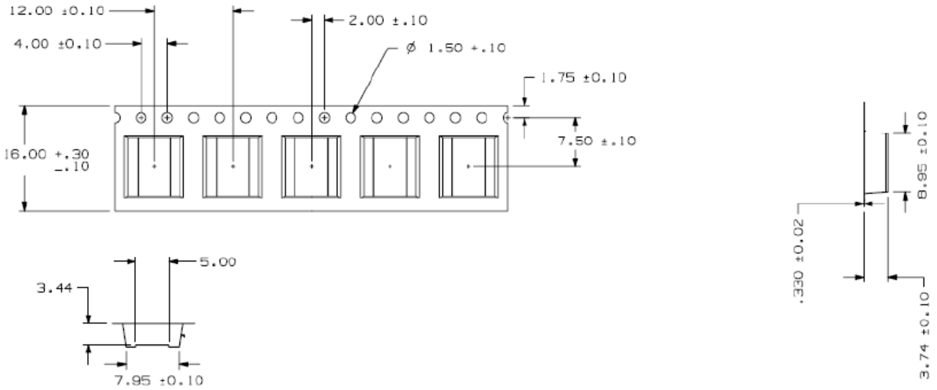


Figure 3: Packing tape dimensions for the 12 lead DFL Package. Dimensions in millimeters [mm].

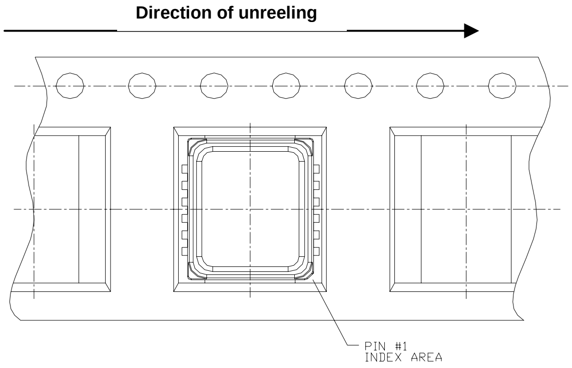


Figure 4: Package polarity and unreeling direction on tape.

The reel dimensions are presented in figure 5 and table 1 below. Dimensions in millimeters [mm].

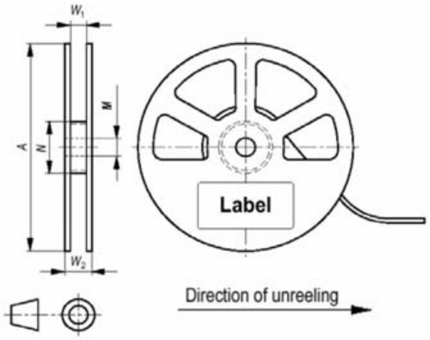


Figure 5: Reel dimensions. All dimensions in millimeters [mm].

Table 1. Packing reel dimensions [mm].

A	N	W1	W_{2max}	M
330	100±1	16.4 (-0/+2.0)	22.4	∅ 13.0 (-0.2/+0.5)

5 Printed Circuit Board (PCB) Level Guidelines

5.1 PCB design recommendations

5.1.1 Land pad design

There are two types of land patterns, which are used for surface mounted 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 PCB. For the DFL package, NSMD pads are preferred, since the copper etching process has tighter process control compared to the solder mask process.

For optimal soldering and solder joint reliability results of VTI's DFL component, the PCB terminal pads should be designed larger than the package leads. The PCB land should be extended 0.055mm on both two *sides* of the component lead. Towards the inside of the package, the PCB land is recommended to be extended by 0.06mm. Towards the outside of the package, the recommended extension for the land pad length is 0.14mm. These recommendations for the land pad design are also presented in Figure 6.

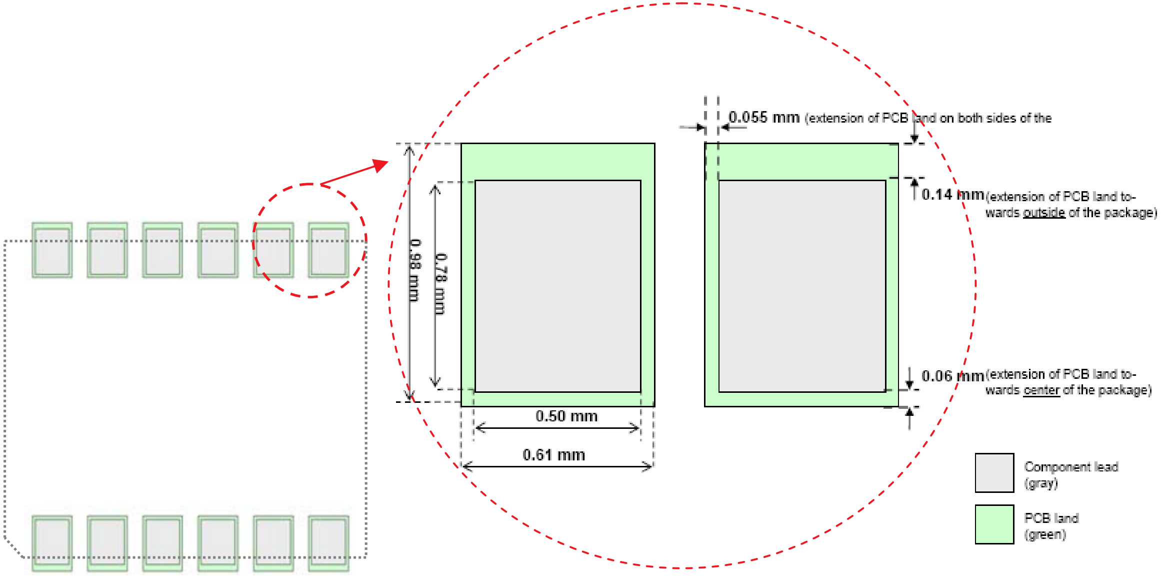


Figure 6: Land pattern design for the 12 lead DFL.

VTI's DFL packages can be soldered on commonly used substrates, e.g. FR-4, ceramic etc. The pad metallization should be solder wettable in order to assure good quality solder joints. For fine pitch assembly, the quality of plating is important. Generally used circuit board finishes for fine pitch SMD soldering are NiAu, OSP, Electroless-Ag and Electroless-Sn.

5.2 Solder paste

The DFL package can be soldered with lead-free SAC (tin-silver-copper) solder. The SAC solder paste composition should be near eutectic. The melting point of lead-free SAC solder can vary between 217–221ºC, depending on the composition of solder alloy. In order to guarantee full RoHS compatibility lead-free solder should be used for the soldering of VTI's DFL component. Also traditional eutectic SnPb solder can be used for soldering the DFL packages if a lead-free process is not required. With the eutectic SnPb solder, the melting point is 183°C.

A no-clean solder paste is recommended, since the DFL package has no stand off and the effectiveness of cleaning process is therefore low. Ultrasonic agitation wash is not allowed for VTI's DFL packaged MEMS components. The solder paste which is used 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 PCB 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 transferring 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 PCB.

For the 12 lead DFL package, the recommended stencil thickness is 150mm.

Stencil apertures in general can be 1:1 to PWB pad sizes, or stencil apertures can be reduced by 5-10% from all sides in regard to the PCB 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 proper care of printing speed is taken during the 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

The DFL package can be picked from the carrier tape using either vacuum assist or mechanical type pick heads. Typically a vacuum nozzle is used. Pick up nozzles are available in various sizes and shapes to suit a variety of different component geometries. It is recommended that different pick up nozzles are tested to find the best one. The polarity of the part must be assured in taping process. The polarity of the part on tape was presented in Figure 4.

5.6 Component placement

DFL packages must be placed onto the PCB 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. Pin #1 is indexed by a faceted corner in the package was shown in figure 2.

5.7 Reflow soldering

A forced convection reflow oven is recommended to be used for soldering DFL components. IR-based reflow ovens are not generally suitable for lead-free soldering. Figure 7 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 DFL package should always follow the solder paste manufacturer's specifications and recommended profile. The typical ramp-up rate is 3°C/second max. The reflow max. peak temperature (measured from the component body) should not exceed 260°C. The ramp down rate should be 6°C/second max. If washing process is done after the soldering process, it must be noted that ultrasonic agitation wash after reflow is not allowed for

VTI's DFL packaged MEMS components. As mentioned before (section 5.2) a no-clean paste is recommended.

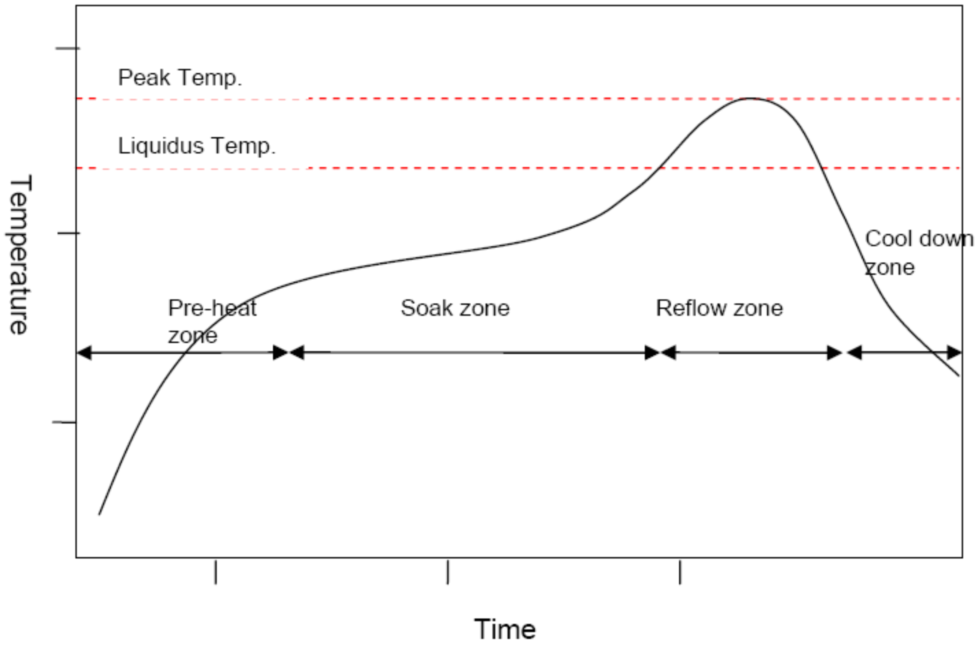


Figure 7: Typical convection reflow soldering phases and profile.

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 to use at least three thermo-couples, 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 DFL component'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 highly different thermal masses.

5.8 Moisture sensitivity level (MSL) classification

The Moisture Sensitivity Level of the DFL 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 DFL package is 260°C/40sec, measured from the package body.

5.9 Inspection

Optical and visual inspection of solder joints can be done only partly, since the solder joints are partly located underneath the DFL component itself. A visual inspection of the solder joints with conventional AOI (automatic optical inspection) system is limited to the outer surfaces of solder joints. Figure 8 shows photos of the soldered DFL component on PCB.

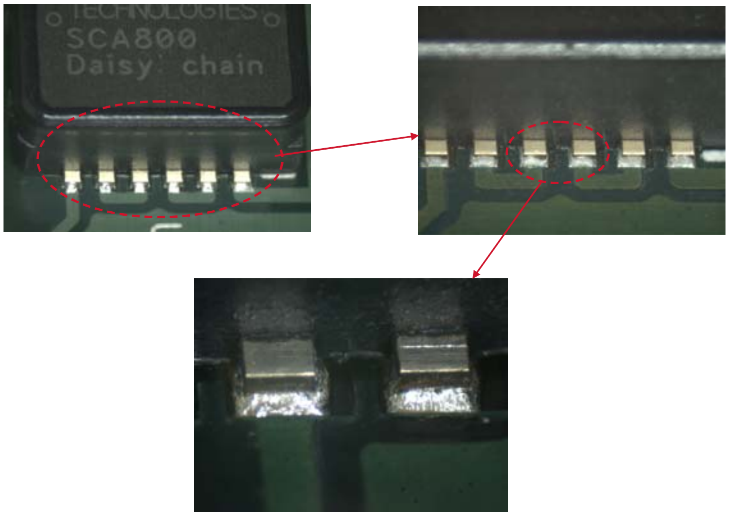


Figure 8: Soldered DFL component on PCB. Solder joints are partly located underneath the DFL component itself.

X-ray inspection is a recommended inspection method for being able to check the complete solder joint area for solder bridges or short-circuits between solder pads. The X-ray inspection systems vary from manual to fully automated optical inspection systems. X-ray can be used for sample based process control, but it can also be implemented as an automatic in-line control. X-ray inspection can also be used to establish and optimize the component assembly process parameters.

Cross-sectional analysis is also an approved method to inspect how well solder has wetted the pads of component. Cross-sectional analysis is not used for production inspection, but if required, it can be used to establish and optimize the component assembly process parameters. Cross-sectioning is a destructive inspection method. An example of a DFL solder joint cross-section in presented in Figure 9.

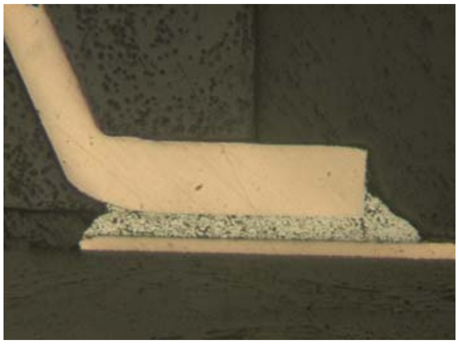


Figure 9: Cross-section of the DFL package lead's solder joint.

6 Hand Soldering Guidelines

For hand soldering of the DFL component, VTI recommends eutectic tin-lead solder due to the lower melting point compared to lead-free solders. Generally the hand soldering of the DFL component can be done in two different ways:

Method A: Soldering of components with tin wire and soldering iron

Method B: Soldering by applying solder paste onto PCB land pads and *then* using reflow heating or soldering iron to melt the paste and achieve the soldered joint.

To make the manual hand soldering with soldering iron easier, VTI recommends increasing the PCB land pads length toward the outside of the package by up to 1mm, as shown in Figure 10 below. This extra area is helpful when soldering iron is being used. The two methods for hand soldering are presented in more detail below.

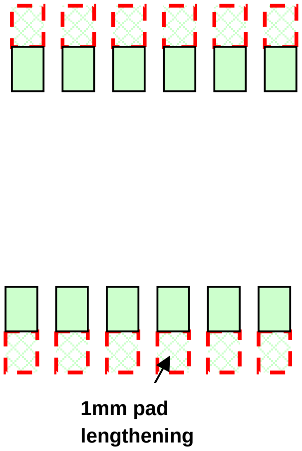
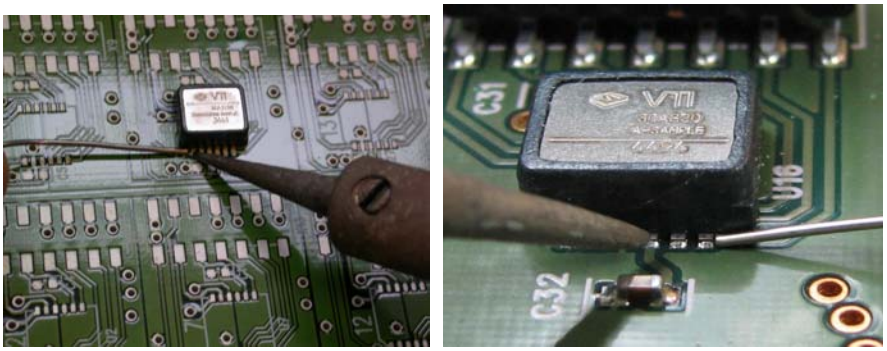


Figure 10: Recommended PWB land pad lengthening for hand soldering.

METHOD A: Soldering of components with soldering iron

1. Place the DFL component onto the PCB

2. Use the tin wire and soldering iron to solder the component onto PCB. Do not touch the package plastic body with the soldering iron, soldering iron should touch only the PCB pad and through that the heat should be conducted to the tin wire and component lead. This is presented in Figures 11 & 12. Flux can be used to ease the soldering process.



Figures 11 & 12: Using tin wire and soldering iron for hand soldering.

METHOD B: Soldering by applying solder paste and then using reflow heating or soldering iron

1. Apply the solder paste onto the PCB land pads. The paste can be applied by two different methods.
    - Manual solder paste printing through a stencil with normal openings designed for the DFL component. Printed solder paste on the PCB pads is presented in Figure 13.
    - Needle dispensing of solder paste manually onto the PCB pads. VTI recommends use of a microscope in manual solder paste dispensing. Dispensing needle tip size can be 0.12"-0.16".

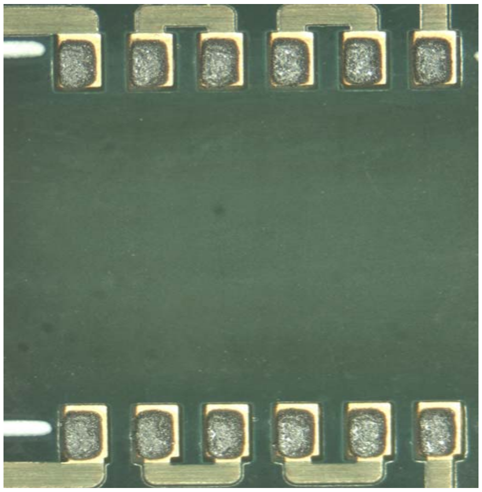


Figure 13: Solder paste on PCB land pads (Note: In this photo the solder pads are not extended).

2. Place the component gently on top of the solder paste. To avoid solder bridging, push only very gently on top of the component.

3. **i)** Melt the solder paste by putting the part through reflow oven *or* **ii)** by using soldering iron as mentioned in the previous part. When using soldering iron, do not touch the package plastic body, soldering iron should touch only the extension in the PCB's land pad through that the heat should be conducted to melt the solder paste and then form the solder joint to the component lead. See figure 14 for details.

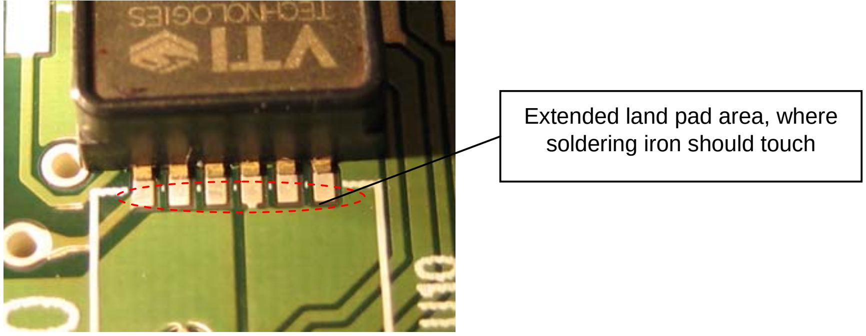


Figure 14: Solder paste on PCB land pads.

7 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 DFL 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, PCBs 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 PCB 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 PCB is recommended in order to reduce the PCB warpage during the rework operation.

Once the solder joints have been fully melted, the component can be carefully removed from the PCB. 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 PCB and the component.

After the component is removed, the pad areas of the PCB 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 PCB 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 PCB, a solder paste should be applied on the cleaned PCB 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 PCB. 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 PCBs should be allowed to cool to the room temperature. The PCBs 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.

8 Environmental Aspects

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

9 References

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

10 Document Change Control

Version	Date	Change Description
1.0	19.06.2006	First release
1.1	25.9.2006	Tape&Reel, MSL classification and Rework chapters added.
1.2	13.2.2007	Minor changes in chapter 5.7. regarding reflow soldering details