TECHNICAL NOTE

Assembly instructions of Dual Flat Lead Package (DFL)

TABLE OF CONTENTS

1 Objective

This document provides general guidelines for Printed Circuit Board (PCB) design and assembly of Murata Electronics' 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 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.

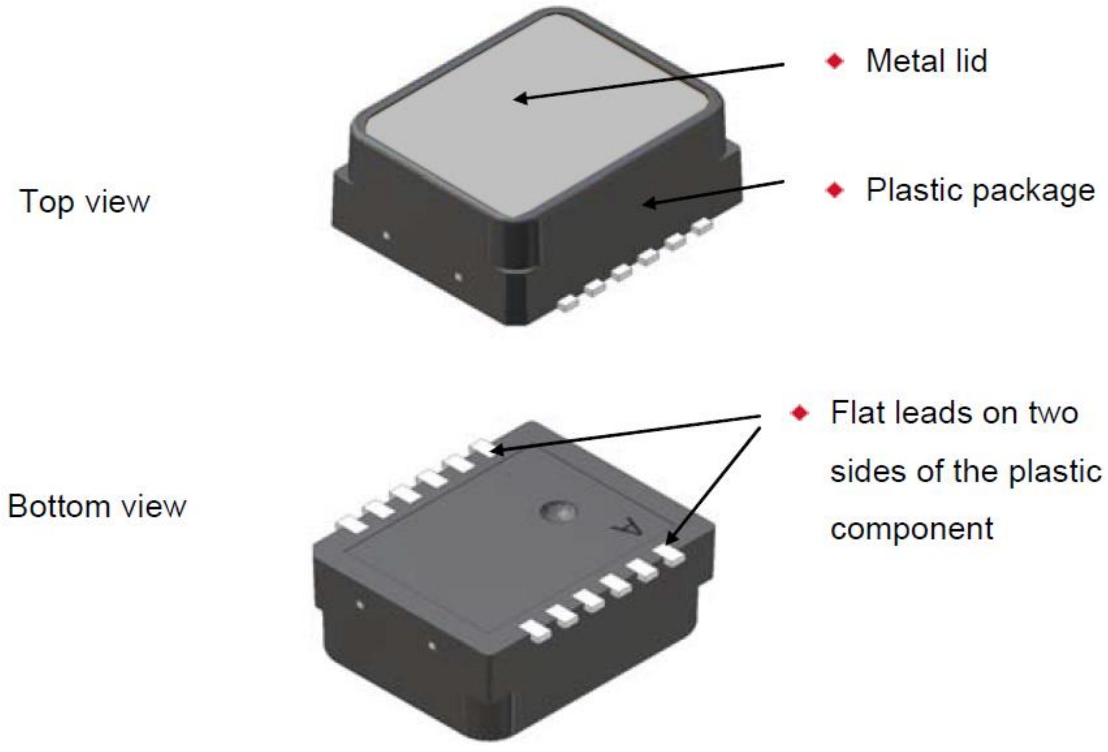


Figure 1 Three dimensional views of the Dual Flat Lead package.

3 DFL Package Outline and Dimensions

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

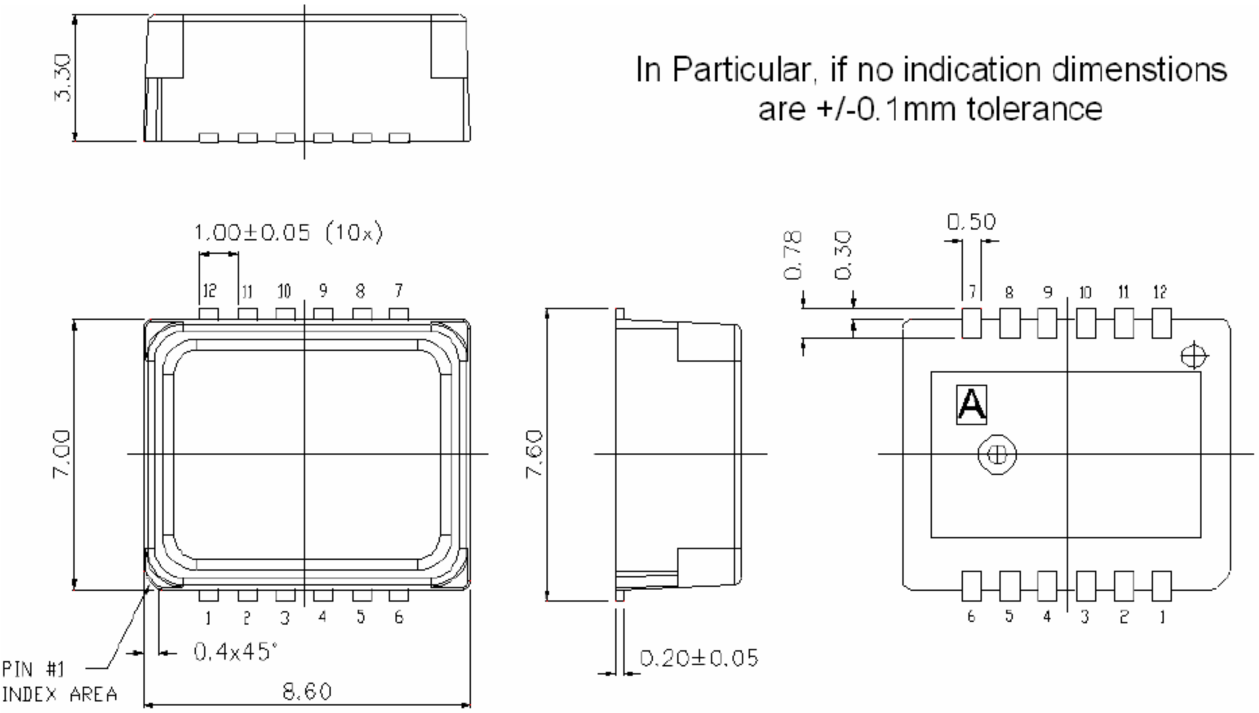


Figure 2 Outline and dimensions for 12 lead DFL package.

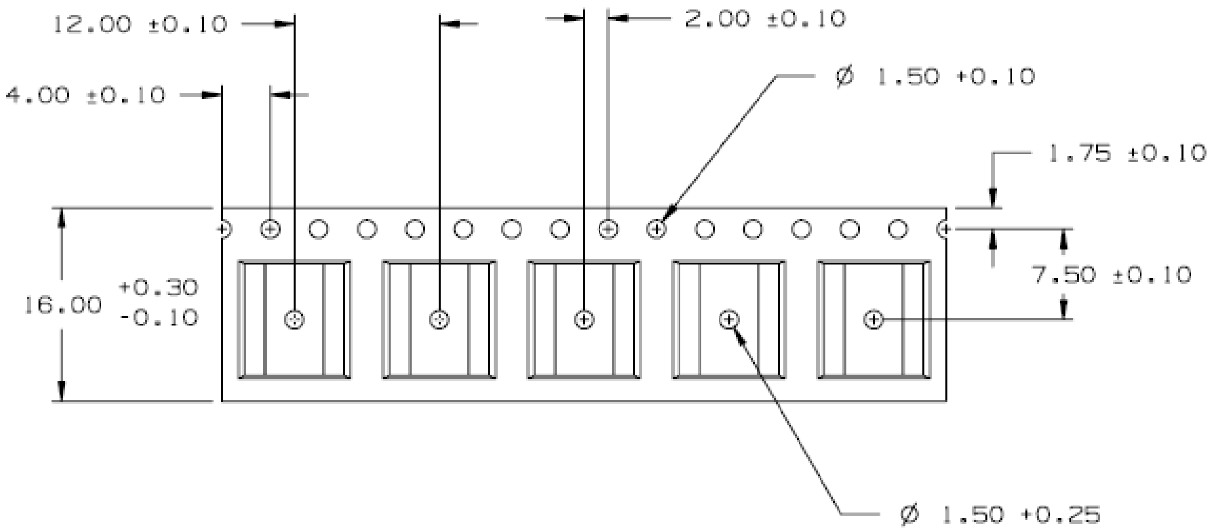
4 Tape and reel specifications

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

4.1 Tape and Reel Materials and Dimensions

- Carrier tape 3M Carrier tape
- Cover Tape 3M Conductive Pressure Sensitive Cover tape 2666, Width 13.3 mm
- Reel Bakeable 125°C SMD-REEL 016-330-4

Note: Tape&Reel materials are suitable for baking for 24 hours at 85°C



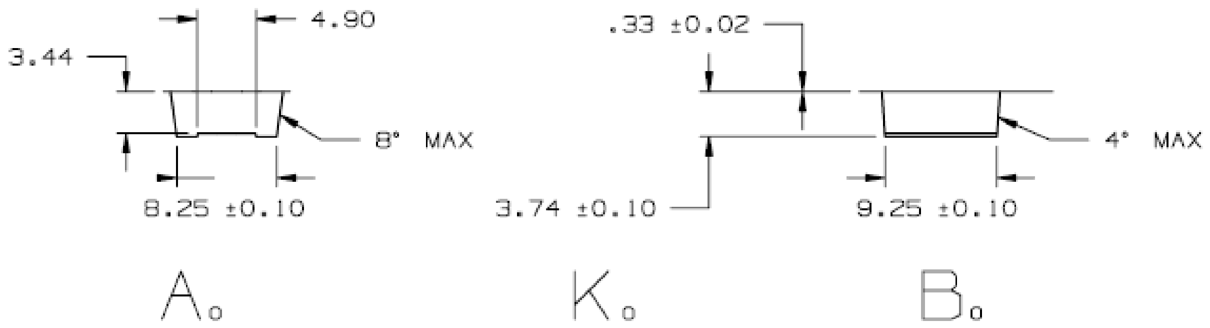


Figure 3 Packing tape dimensions for the 12 lead DFL Package. Dimensions in millimeters [mm].

4.2 Method of Taping/Polarity and Orientation

At the start of the reel shall be minimum of 400mm of cover tape, including at least 100mm of carrier tape with empty compartments sealed by the cover tape. At the end of the reel (near the hub) shall be minimum of 160mm empty carrier tape sealed with cover tape.

Instruction and tips to use Pressure Sensitive Adhesive Cover Tape can be found from 3M Technical Bulletin Prevention of Pressure Sensitive Adhesive Cover Tape Jamming in Feeders with Nip Gear and/or Collection Bins Systems

The polarity of the part must be assured in taping process. The polarity of the part on tape looks as follows:

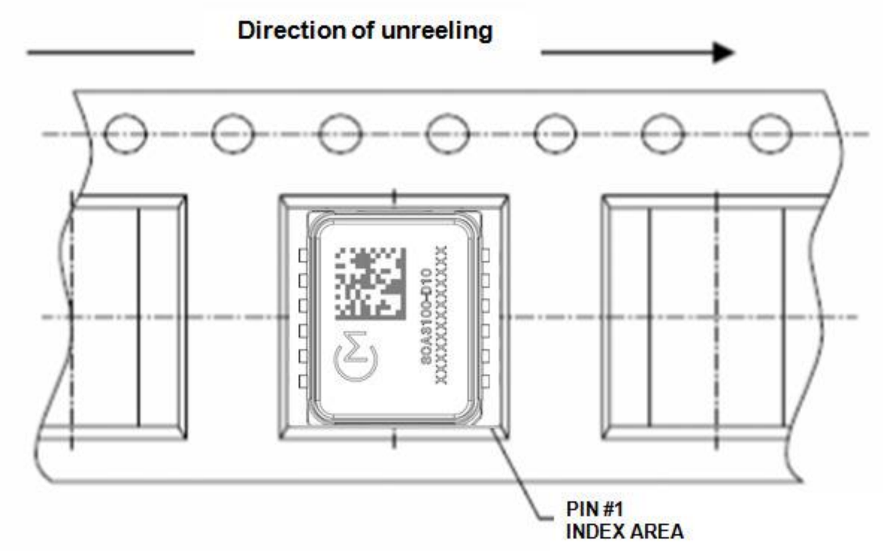


Figure 4 Package polarity and unreeling direction of the tape. Lid markings can be found of corresponding Top Level drawing if applicable.

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

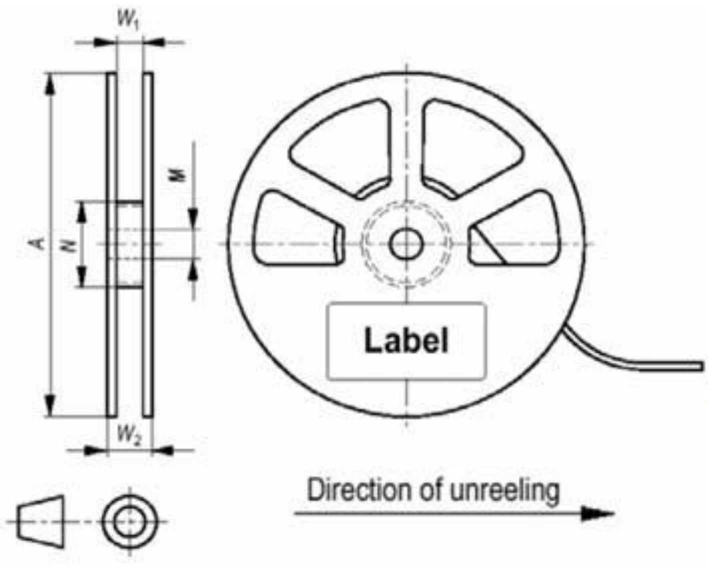


Figure 5 Reel dimensions. All dimensions in millimeters [mm].

A	N	W1	W_{2max}	M
330	100±1	16.4 (-0/+2.0)	22.4	ø13.0 (-0.2/+0.5)

4.3 Dry packing

ADP component is a reflow/moisture sensitive component and it requires a dry pack. Moisture Sensitive Level (MSL) is 3.

Once the devices are removed from the dry pack bag, the total exposure time to the factory environment, prior to mounting the parts onto a circuit board, should not exceed the recommended time specified on the dry pack caution label. The out-of-bag time is 168h for MSL3.

4.3.1 References

- JEDEC-STD-020 Moisture/Reflow Sensitivity Classification for Plastic Integrated Circuit Surface Mount Devices
- JEDEC-STD-033 Standard for Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices
- EIA – 583 Packaging Material Standards for Moisture Sensitive Items
- EIA/JEP 113-B Symbol and Labels for Moisture-Sensitive Devices
- MIL-B-81705 C Barrier Materials
- MIL-D-3464 Desiccants

5 Printed Circuit Board (PCB) Level Guidelines

5.1 PCB design recommendations

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 Murata Electronics' 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.

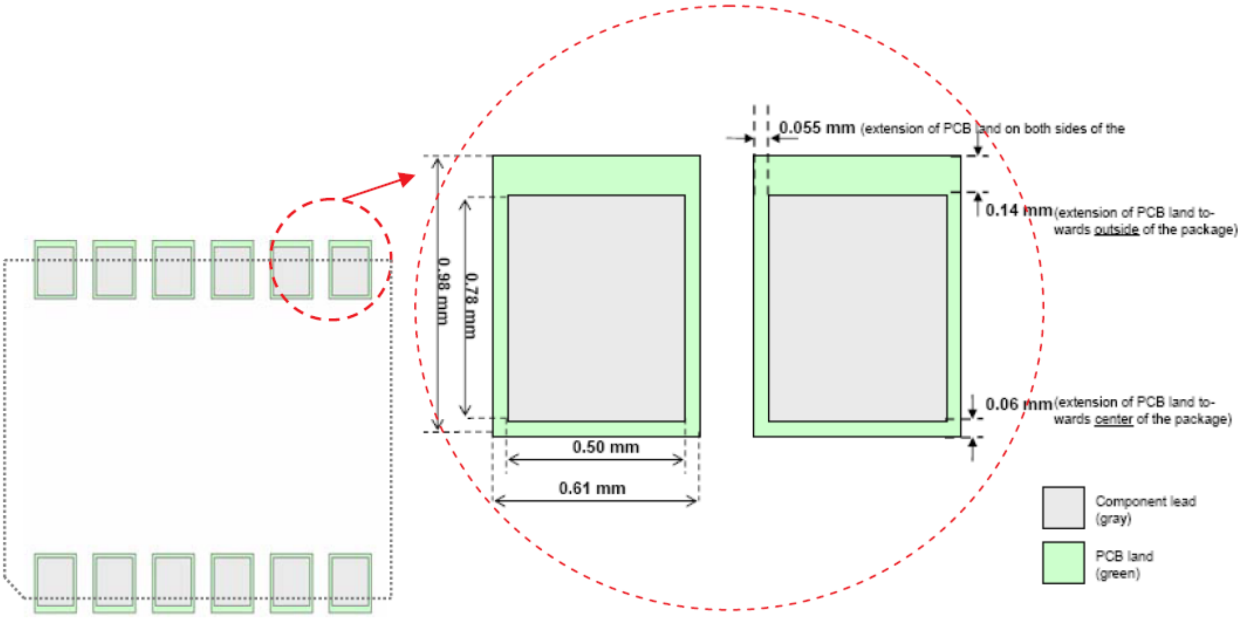


Figure 6 Land pattern design for the 12 lead DFL

Murata Electronics' 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.

For the vibration sensitivity, it is recommended that PCB or Unit resonance modes are analyzed to verify that there are no critical modes close to sensor resonance frequencies or excessive amplifications of hundreds of Gs.

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 Murata Electronics' 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 Murata Electronics' 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 25-45mm).

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 150µm. 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.

Cover tape of reel is pressure forming type tape with adhesive. Caution has to be taken care when rolling off the cover tape to prevent the tape sticking and machine jamming. Extensive baking should be avoided, because it can increase the sticking of the cover tape.

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 250°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 Murata Electronics' DFL packaged MEMS components. As mentioned before (section 5.2) a no-clean paste is recommended.

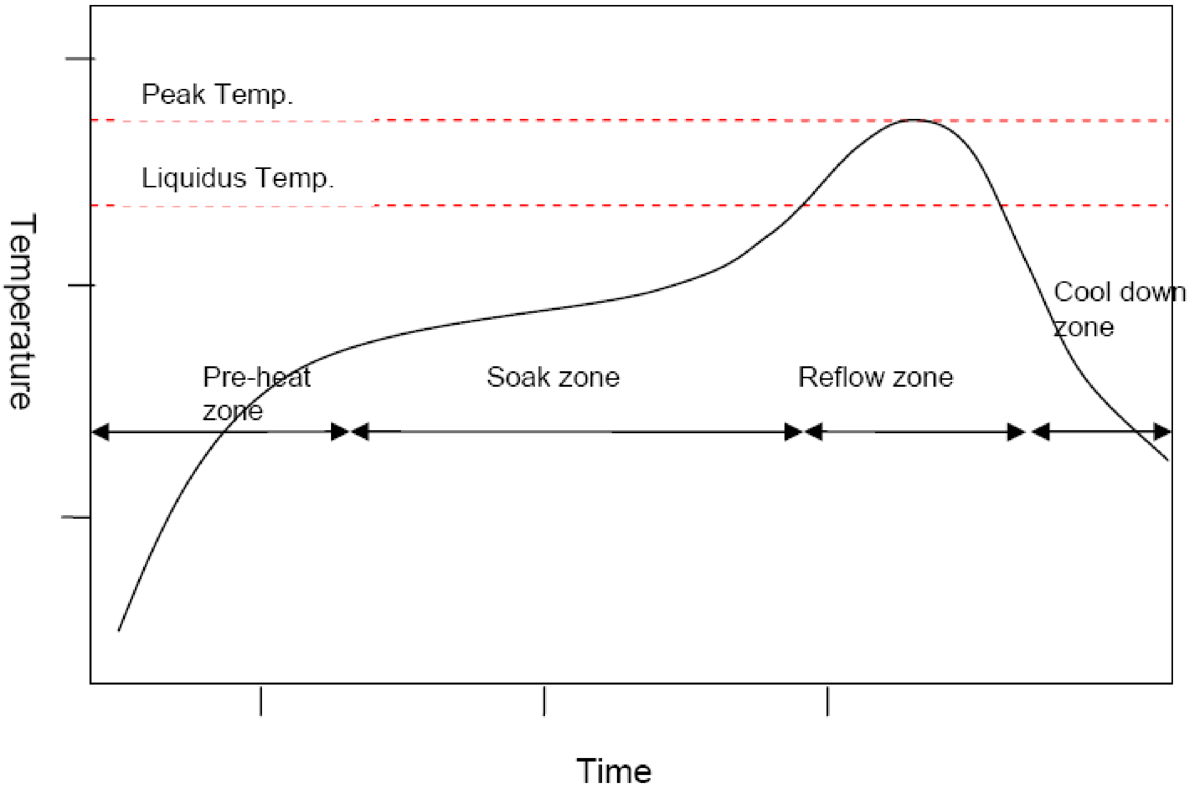


Figure 7 Typical convection reflow soldering phases and profile.

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.

Reflow soldering with vacuum condition is prohibited as it may have effect for sensor performance.

5.8 Moisture sensitivity level (MSL) classification

The Moisture Sensitivity Level of the DFL component is Level 3 according to the IPC/JEDEC JSTD-020. 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 250°C/30sec, measured from the package body.

Following instruction shall be followed:

1. Calculated shelf life in sealed bag: 12 months at < 40 °C and < 90% relative humidity (RH).

2. Maximum soldering peak temperature for the package is 250°C/30sec, measured from the package body.

3. After bag is opened, devices that will be subjected to reflow solder or other high temperature process must be
   a) Stored at <10%RH or
   b) Mounted within 168 hours of factor conditions ≤30 °C/60%RH.
   Note: Do not re-store devices that have exposed >10% RH conditions.

4. Devices require bake, before mounting, if:

   a) Humidity Indicator Card is > 10% when read at 23 ± 5 °C
   b) 3a or 3b not met.

5. If baking is required, devices may be baked for 24 hours at 85°C.

Note: Also Tape&Reel materials are applicable for baking at 85°C.
Note: Packing materials and procedures according to IPC/JEDEC J-STD-033
Note: Level and body temperature defined by IPC/JEDEC J-STD-020

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.

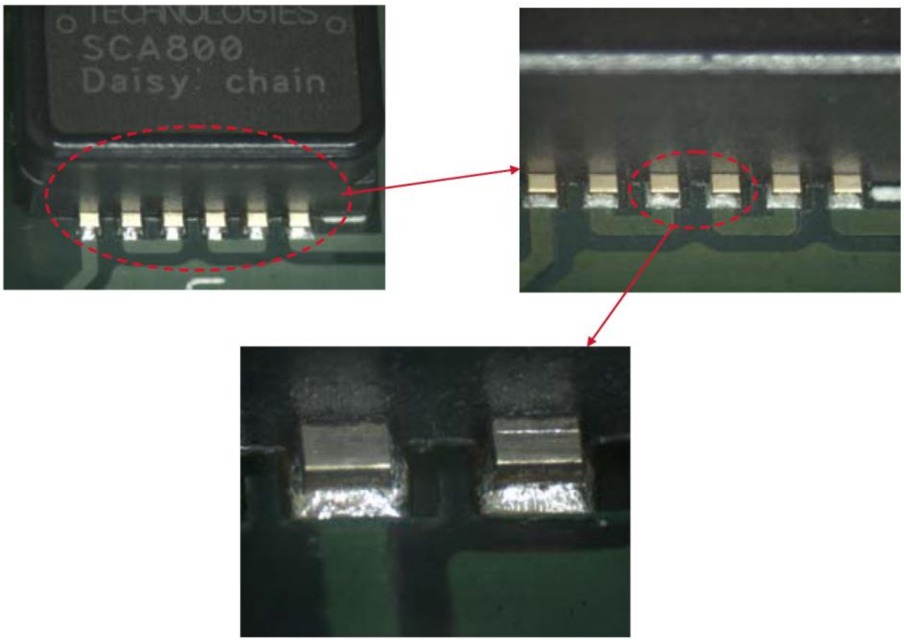


Figure 8 Soldered DFL component on PCB. Solder joints are partly located underneath theDFL component itself

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.

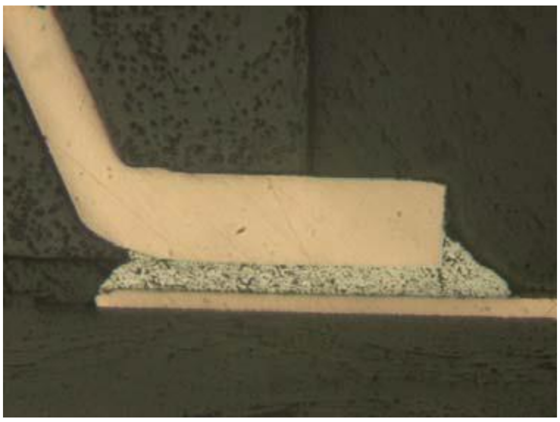


Figure 9 Cross-section of the DFL package lead's solder joint.

5.10 Precautions

MEMS sensors are mechanically and electrically sensitive components. Following sections describe typical processes or treatments, but are not limited to, which may be harmful for sensor component. Exceeding these limits or neglecting these guidelines may lead to malfunction of sensor component.

The reliability requirements for the devices are applied and validated according to AEC-Q100 Rev. G.

5.10.1 Mechanical shocks

Shocks may cause mechanical damage to the internal structures of MEMS sensor, causing malfunction of sensor, therefore mechanical shocks should be avoided. The level depends heavily on the pulse width and shape and should be evaluated case by case. As a general guideline, the lighter assembly or part, the higher shock levels will be generated on sensor component. Dropped components shall not be used and shall be scrapped.

Reference tests:

1. Mechanical Shock JEDEC JESD22-B104, 5 pulses, 0.5 ms, 3000g for 5 axes.
2. Package Drop, 10 drops, random orientation, 0.8m drop.

5.10.2 Vibration

Sensor components are mechanical devices and especially sensitive to repetitive vibrations and shocks, therefore vibration of the device should be avoided both prior to or during assembly. Many assembly processes can induce vibration, typical ones being PCB singulation, mechanical shocks, transportation, friction welding and ultra-sonic cleaning.

Reference test:

1. Variable Frequency Vibration, JEDEC JESD22-B103, 20 Hz to 2 kHz to 20 Hz, 4x in each orientation, 30g peak acceleration depending on freq (2-30g at 20-100Hz)

5.10.3 Chemicals

Sensor components shall not be exposed to:

- Chemicals which are known to react with silicones, such as solvents. These are for example used in various cleaning processes
- Chemicals with high impurity levels, such as Cl-, Na+, NO3-, SO4-, NH4+
- Pressurized low molecular gas such as He or H2, used for example in leak test for hermetical sealing
- Materials with high amount of volatile content, like solvents

Materials containing halogens (F, Br, I or Cl), halides, their compounds or phosphorus containing materials (such as in flame retardants, thermal stabilizers in plastics, Bromine in PCB material or halogens/halides in soldering paste) shall be avoided in close vicinity of sensor component.

If heat stabilized polymers are used in application, user should check that iodine, or other halogen, containing additives are not used. Iodide compounds are known to cause issues with gold aluminum interconnects. User should also check that excessive amount of I, Br or other halogen containing additive are not used. Quantitative value is depending on number of factors, including the total volume of such halogen material, stability of halogen within material, hermeticity level of application and temperature, among others.

Life-time reliability tests should always be performed at application level to validate the end product against life-time requirements/mission profile, as corrosion effects heavily depend on the final construction of application, temperature, time and other application specific factors.

5.10.4 Coatings

Coatings on sensor component are not generally recommended. Coating penetration inside the component is high risk for performance change. If coating is required, the effect of coating on sensor component reliability and electrical performance has to be evaluated case by case. As a general guideline, lower the viscosity or higher the modulus of coating, higher the probability of adverse effects on sensor component.

Sealing of the component lid is prohibited.

5.10.5 Vacuum level

Vacuum levels lower than 0.4 bar shall not be applied on sensor component. Fast pressure changes of over 0.5 bar/min should be avoided. Example processes can be vacuum sealing or pressure testing of the device.

5.10.6 Air blowing

Heavy compressed air blowing directly onto sensor component shall be avoided. Air blowing could be used for example during cleaning of the PCB.

5.10.7 ESD

Sensor components are electrical devices. Sensor components should be handled under good ESD practices and ESD discharges should be avoided. The following numbers are absolute maximum ratings:

- ±500 V charged device model
- ±2 kV human body model

5.10.8 Moisture

Sensor components are moisture sensitive devices, classified as MSL3 level. Guidelines defined by IPC/JEDEC J-STD-020 shall be followed.

5.10.9 Mechanical stress

Mechanical stress due to PCB bending, molding and potting may affect the sensor performance. Excessive stress due to PCB bending shall be avoided. If molding or potting material is hard material, contact to sensor shall be avoided.

5.11 Cleaning

Items listed in section "5.10 Precautions" are applied to cleaning also. Examples of cleaning procedures which may have a negative effect on the sensor component performance or functionality are:

- Wet cleaning with solvent
- Ultrasonic
- Heavy air blowing
- Plasma cleaning with vacuum

6 Hand Soldering Guidelines

Hand soldering on sensor component are not generally recommended. Hand soldering may have an effect on the component performance. If required, the effect of hand soldering process on sensor component reliability and electrical performance has to be evaluated case by case.

For hand soldering of the DFL component, Murata Electronics 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, Murata Electronics 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.

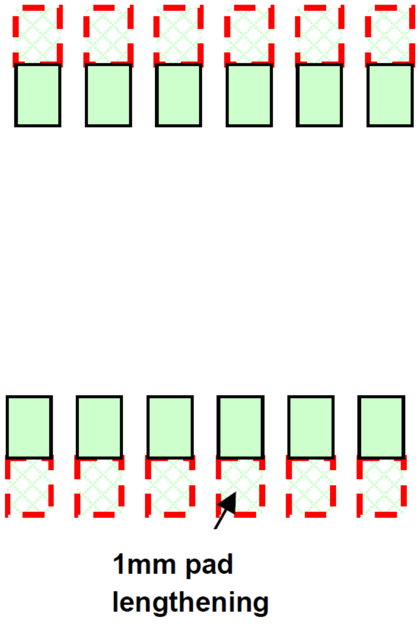


Figure 10 Recommended PWB land pad lengthening for hand soldering.

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.

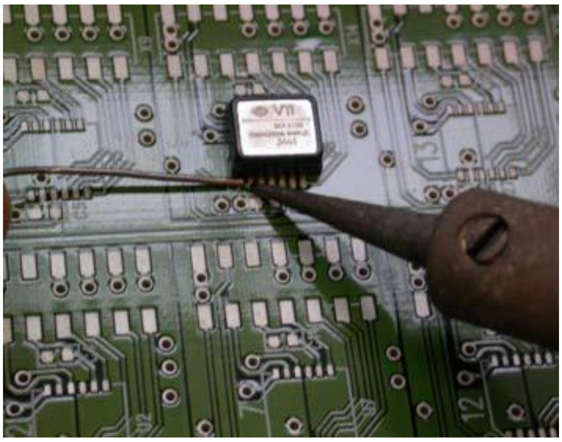


Figure 11 Using tin wire and soldering iron for hand soldering.

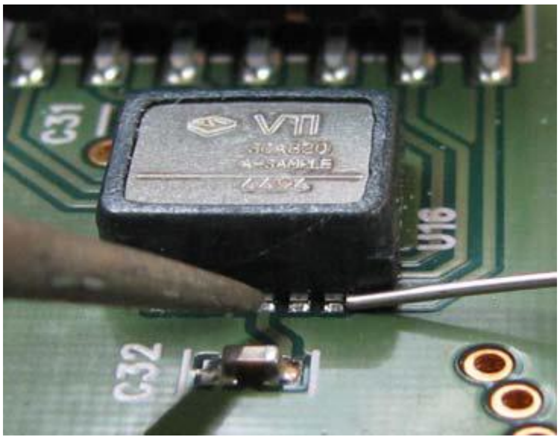


Figure 12 Using tin wire and soldering iron for hand soldering.

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. Murata Electronics recommends use of a microscope in manual solder paste dispensing. Dispensing needle tip size can be 0.12"-0.16".

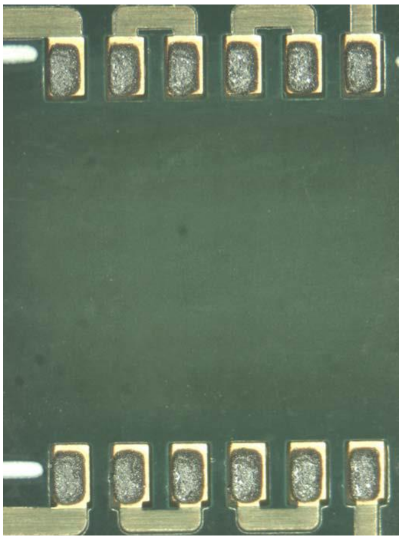


Figure 13 Solder paste on PCB land pads (Note: In this photo the solder pads are not extended).

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.

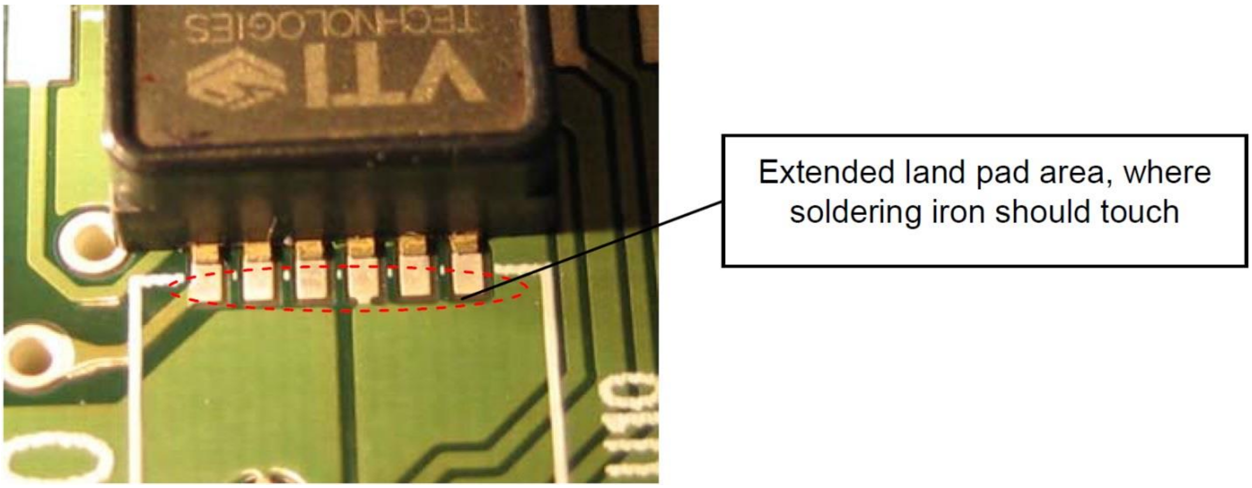


Figure 14 Solder paste on PCB land pads.

7 Rework Guidelines

Rework on sensor component are not generally recommended. Rework may have an effect on the component performance. If required, the effect of rework process on sensor component reliability and electrical performance has to be evaluated case by case.

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 Murata Electronics'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

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

9 References

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