FAIRCHILD™

Is Now Part of

ON Semiconductor®

To learn more about ON Semiconductor, please visit our website at
www.onsemi.com

ON Semiconductor and the ON Semiconductor logo are trademarks of Semiconductor Components Industries, LLC dba ON Semiconductor or its subsidiaries in the United States and/or other countries. ON Semiconductor owns the rights to a number of patents, trademarks, copyrights, trade secrets, and other intellectual property. A listing of ON Semiconductor's product/patent coverage may be accessed at www.onsemi.com/site/pdf/Patent-Marking.pdf. ON Semiconductor reserves the right to make changes without further notice to any products herein. ON Semiconductor makes no warranty, representation or guarantee regarding the suitability of its products for any particular purpose, nor does ON Semiconductor assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation special, consequential or incidental damages. Buyer is responsible for its products and applications using ON Semiconductor products, including compliance with all laws, regulations and safety requirements or standards, regardless of any support or applications information provided by ON Semiconductor. "Typical" parameters which may be provided in ON Semiconductor data sheets and/or specifications can and do vary in different applications and actual performance may vary over time. All operating parameters, including "Typicals" must be validated for each customer application by customer's technical experts. ON Semiconductor does not convey any license under its patent rights nor the rights of others. ON Semiconductor products are not designed, intended, or authorized for use as a critical component in life support systems or any FDA Class 3 medical devices or medical devices with a same or similar classification in a foreign jurisdiction or any devices intended for implantation in the human body. Should Buyer purchase or use ON Semiconductor products for any such unintended or unauthorized application, Buyer shall indemnify and hold ON Semiconductor and its officers, employees, subsidiaries, affiliates, and distributors harmless against all claims, costs, damages, and expenses, and reasonable attorney fees arising out of, directly or indirectly, any claim of personal injury or death associated with such unintended or unauthorized use, even if such claim alleges that ON Semiconductor was negligent regarding the design or manufacture of the part. ON Semiconductor is an Equal Opportunity/Affirmative Action Employer. This literature is subject to all applicable copyright laws and is not for resale in any manner.

AN-4170

Board Assembly Guideline for Fairchild TO-Leadless Package

Summary

This application note is created to guide users of the TO-Leadless (TO-LL) package on how to use it in printed circuit board assembly.

Board Attributes and Design Guide

Pad Design

The board pads are either solder mask defined (SMD) or non-solder mask defined (NSMD). This is illustrated in Figure 1.

In NSMD, the copper pad is etched out to define the land pattern. The overall pattern registration is dependent on the copper artwork. The SMD pad, on the other hand, is defined by a photo-imageable solder mask process. This requires the copper artwork to be larger than the recommended land pattern, in which the solder mask will define the solderable pads.

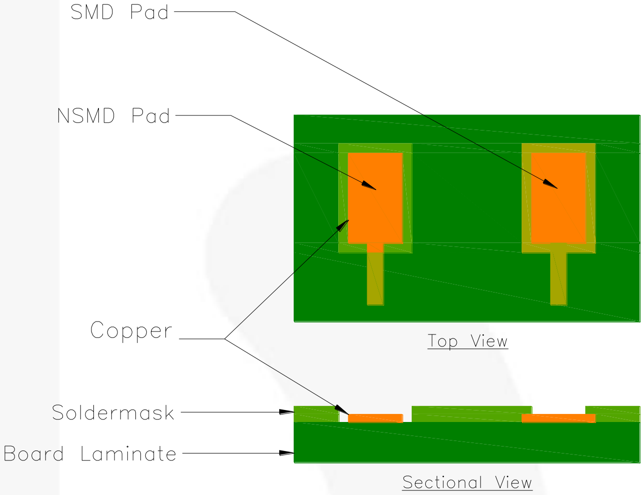


Figure 1. Solder-mask-defined (SMD) vs. Non-Solder-Mask-Defined (NSMD) Board Pads

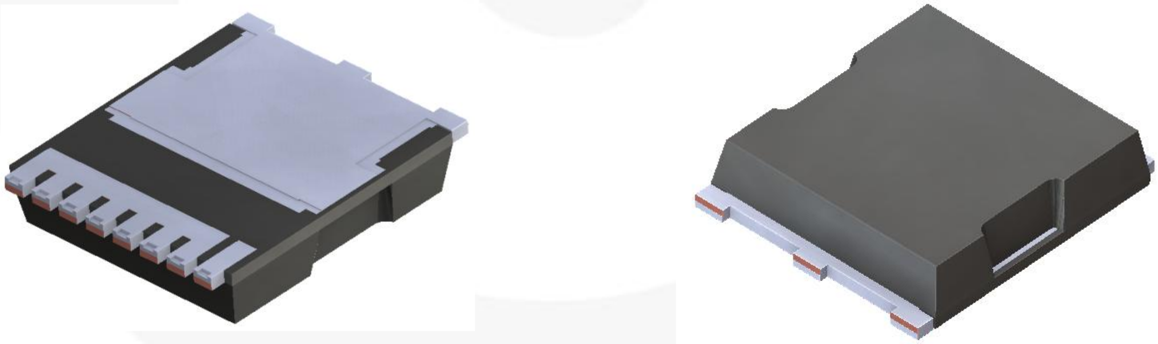


Figure 2. TO-Leadless Package (TO-LL)

© 2014 Fairchild Semiconductor Corporation

One advantage of the NSMD pad is that, because of the smaller area covered, this allows more area for routing traces around the component. For SMD, because of the large Cu coverage, this provides more metal for dissipating heat. The overlap on the copper pad also helps prevent bridging of the paste between the leads and the exposed pad by blocking solder from flowing outside the solderable pads during the reflow process. The solder masks also anchors itself on the copper to give better copper adhesion to the board.

For the TO-LL package, it is recommended to use the SMD pads. The high power application of this package, allows for better dissipation of heat produced during the application.

In routing traces from the solderable pads, it is also recommended that these should not be routed in between the leads and the thermal pad; in high power application, high potential between the terminals can create arcing between the narrow spaces between metals.

Below is the recommended land pattern design for the TO-LL package. In a solder-mask-defined pad, the outline below is defined by the solder mask of the PCB.

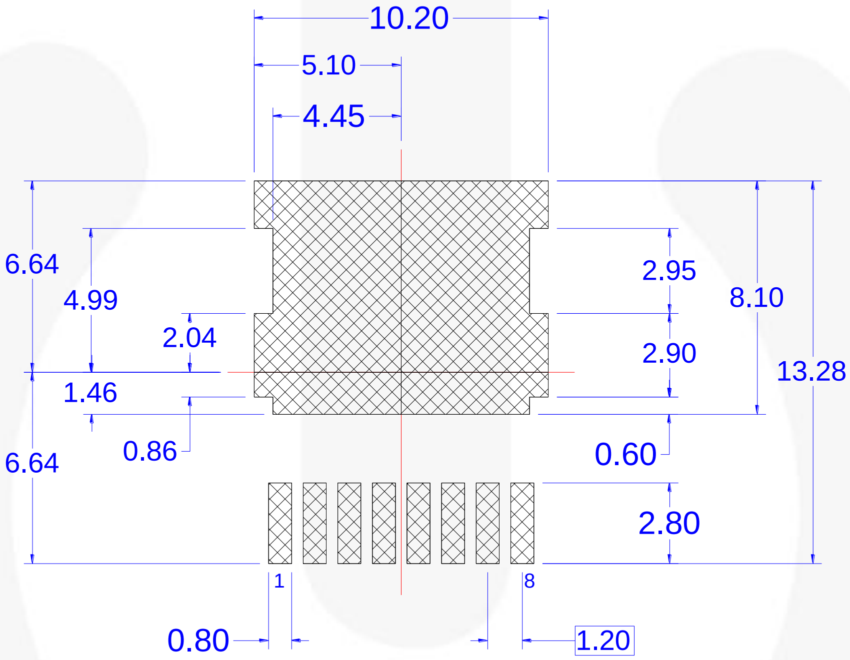


Figure 3. Recommended Land Pattern

Board Surface Finish

The pad surface finishes that are commonly used are Electroless Nickel Immersion Gold (NiAu), Organic Solderability Preservative (OSP) and Hot Air Surface Leveling (HASL). NiAu is preferable in some applications and fine-pitch packages because of its excellent surface solderability and flatness. OSP is excellent for use with fine pitch packages and BGA. It has a very low cost and excellent flatness. HASL is the most readily available surface finish. It has a superior barrel fill and solderability characteristics.

There is no preference for TO-LL on the suitable surface finish; however, reliability tests have been performed on this package using NiAu and OSP surface finishes, wherein the package passed the tests.

Board Assembly Considerations by Solder Paste Printing and Reflow Process

Solder pastes normally used in board assembly have particles sizes of either Type 3 or Type 4, and their flux materials are classified as rosin based, low activity and halide-free (ROL0) or the no-clean solder paste (typically a ROL0 or ROM0). Typical metal loading of the paste ranges from 88 to 90% solder alloy in the paste; this is approximately 50% solder paste by volume. The most common solder alloys for board assembly are the eutectic SnPb solder (63Sn37Pb) and the SAC305 (95.5Sn3.0Ag0.5Cu) for the Pb-free assembly.

Fairchild recommends that solder pastes with a no-clean flux should be used in the board assembly of TO-LL package. Because of the low standoff height of the solder joints; there is difficulty in cleaning the trapped flux residues between the component and the board.

In using the solder paste, it is recommended that proper handling procedure recommended by the solder paste vendor be followed. Proper thawing of the solder paste is needed to ensure minimal moisture uptake during processing.

Stencil for Solder Paste Printing

Shown below are the recommended stencil aperture designs for the TO-LL packages

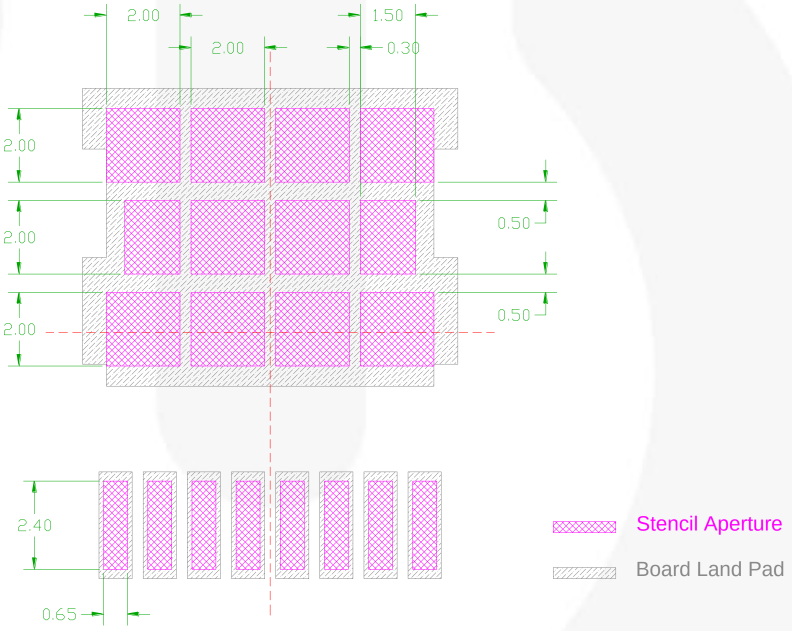


Figure 4. Recommended Stencil Aperture Design

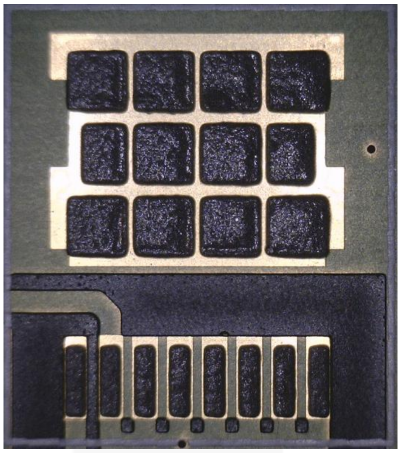


Figure 5. Sample Solder Paste Test Print on an SMD Land Pads

The aperture for leads and exposed die pad are relatively smaller than the board pads. The total aperture size for the exposed thermal pad is about 40 to 70% of the land pad size. The apertures on the leads are smaller than the pads. The recommended stencil thickness is 0.10 mm.

For thicker stencils, it is recommended that the aperture size be reduced proportional to the thickness added, such that the total paste volume is approximately the same. However, as the stencil thickness goes up, the aperture aspect ratio and area ratio are lower, these should be checked to be above the typical values of 1.5 and 0.67 respectively; this prevent clogging of the stencil and have good solder paste release during printing. The stencil aspect and area ratio are calculated as follows:

$$\text{Aspect Ratio} = \frac{\text{Width of Aperture}}{\text{Thickness of Stencil Foil}} \qquad \text{Area Ratio} = \frac{\text{Area of Pad}}{\text{Area of Aperture Wall}}$$

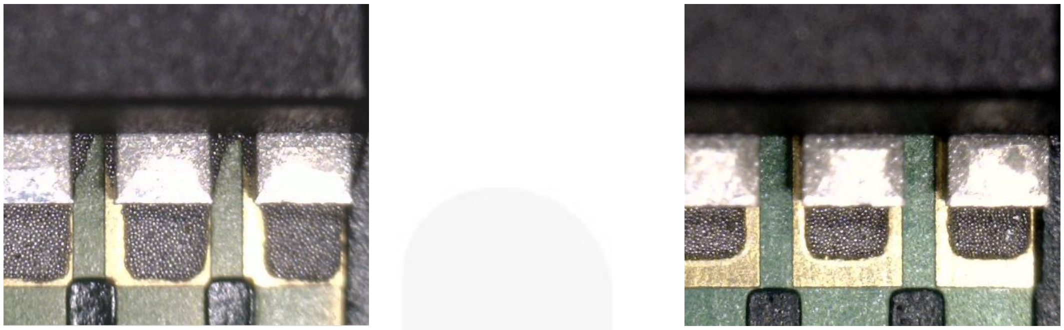
Tapering or trapezoidal opening of stencil holes are recommended, because this improves the solder paste released during paste printing. Electro-polishing a laser cut stencil is also recommended to have a better solder paste release. Use of electroformed stencil is also recommended because of its excellent release performance and its long useful life, but it may not be necessary for this type of package. The drawback for the electroformed stencil is its cost; this is significantly more expensive than the laser cut stencil.

Component Placement

Depending on the placement accuracy of the pick-and-place machine, the TO-LL packages can tolerate up to a certain amount of placement offset. Simulations done showed that the TO-LL package can allow placement offset up to 0.10 mm, which is relatively loose in comparison to the capability of different pick-and-place machine available in the market, that can do of up to ± 0.05 mm placement accuracy. Simulation done using offset component placement showed that the component self-align with the board pads after reflow; no bridging or related anomaly observed.

Placement height or pressure should be taken into consideration during component placement on the board. Compressing the printed paste result to solder paste being spread out on the land pad area, this can result to bridging of solder paste in the leads, as shown below. Bridging of solder paste can create an imbalance in solder volume between leads during reflow. More solder may flow to one lead and, in turn, it reduces the volume on the other.

Compressing the printed paste, on the other hand, may reduce or close the gap between apertures for the thermal pads, this limit the flow of outgas from the solder paste. Simulation showed that this can lead to solder beading.



(a) (b)

Figure 6. ***(a) Compressed Solder Paste, Paste Bridging is Observed, (b) Good Unit Placement***

It is recommended that the placement height be controlled. Most component mounter has the capability to control the height or the amount of bonding force during component placement.

Reflow Process

The temperature profile to use during reflow of the TO-LL package should be based on the recommended temperature profile of the solder paste vendor. The reflow temperature profile typically consists of heating, soaking, soldering and cooling zones. At the heating zone, the temperature is slowly being ramp to meet the target soaking temperature. Heating rate is being kept at a minimum to minimize temperature gradient across the board and to avoid subjecting the components to a thermal stress. Typical heating rate is at 1°C/second. Solder flux is being activated in the soaking zone; this will facilitate wetting of the solder once this goes through the soldering or reflow zone. In the soldering zone, the solder liquefies, and this is where solder wetting and soldering occurs. This is the temperature where the solder liquefies (liquidus temperature). At this zone, the time above liquidus temperature (TAL) and the peak temperature is being controlled. Typically, the time above the liquidus temperature is between 30 to 120 seconds, and the peak temperature is normally 20 to 30°C above the liquidus temperature. At this zone, intermetallics are formed between the interfaces of the board, liquefied solder and component; this creates the bond between these interfaces. Long exposure above the liquidus temperature can result to excessive formation of intermetallic compounds, which can reduce the solder joint reliability, short exposure above the liquidus temperature may result to insufficient solder wetting, which will be manifested by incomplete solder wetting coverage or voids. Figure 7 shows a typical reflow temperature profile for a Pb-free solder.

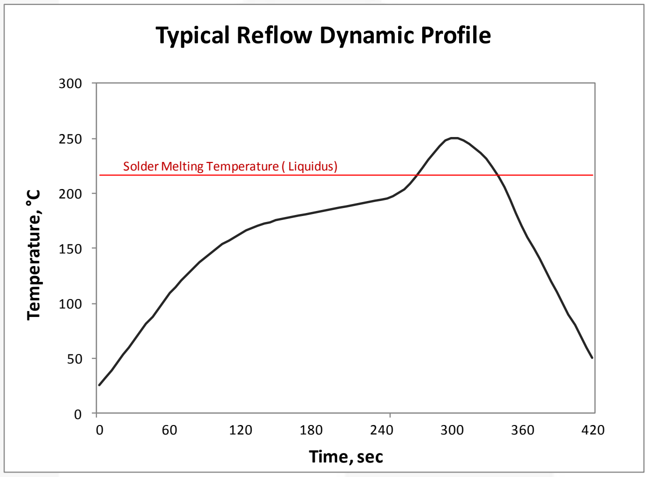


Figure 7. Sample Reflow Temperature Profile

Reflow temperature profiling is being done to check the temperature of the board and component once it goes inside the reflow oven. This is done by running the board through the reflow oven with thermocouples attached to the board. Temperature measurements are recorded using a temperature profiler. The resulting reflow temperature profile is then being compared with the recommendation of the solder paste vendor. Adjustment in the temperature settings and conveyor speed may be needed to meet the target temperature profile.

Thermocouple position is critical in conducting the reflow temperature profiling. This may be placed on areas where temperature sensitive components are present, on areas where the board is densely populated by components or on areas where there are fewer components. Placing thermocouples on these locations of the board will help determine whether there are temperature gradients within the board, and whether all the components on the board are being subjected to the right temperature conditions. It is advised to check the handling requirements of all the components on the board, some components may be rated to certain maximum temperatures only. In the case of the TO-LL, this has been tested to meet moisture sensitivity level 1 at 260°C peak reflow temperature per IPC/JEDEC J-STD-

020, thus, no special handling such as baking of this component prior to the assembly process is not required.

Different board design, thickness and material may require different sets of reflow parameters, with this; reflow temperature profiling must be performed for each.

There are a few different reflow techniques for board assembly. These are listed on Fairchild's Application Note, AN-7528, these are infrared reflow, forced convection reflow, vapor phase reflow and wave-soldering. The first three techniques may be used for the TO-LL; wave soldering cannot be used for this component.

Inspection

Inspection of the mounted component should be done with the use of 10-20x magnification scope and transmission or laminograph x-ray.

A well-reflowed solder joint shows evidence of wetting and adherence wherein the solder merges to the soldered surface forming a contact angle of ≤ 90°. The solder joints should normally have a smooth appearance. On certain occasion, a matte, dull or grainy solder joints may appear, this can be due to the solder alloy used, the component termination or board pad surface finish, or the soldering process used. IPC-A-610 provides the inspection methodology and acceptance criteria for this package.

Figure 8 shows the side view of the leads that has already been soldered on board. A cross-section of this lead is shown on the right.

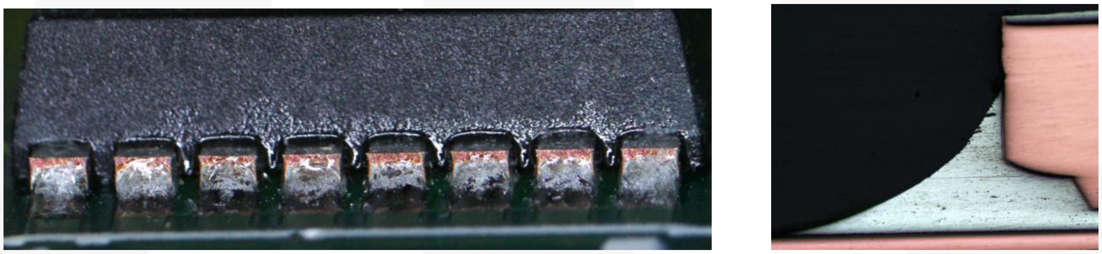

(a) (b)

Figure 8. (a) Side View Showing Leads Solder Joint, (b) Cross-Sectional View of Leads

The extended lead land pad and the notch on the tip of lead allow solder fillet to be formed on the tip of the leads. This will allow the AOI to be used in auto inspection of the leads' solder joints.

The solder coverage at the drain soldered unit can't be inspected visually since it's not exposed. The appropriate control for this is through x-ray inspection of the solder coverage between the land pad and the solderable surfaces at the bottom of the component.

The most common anomaly observed during board assembly simulation done on the TO-LL is solder beading. An x-ray image below shows that beads can either be around the periphery of the component or underneath.

Evaluation done showed that this solder bead maybe due to excessive solder paste printed on the board or too much force applied during component placement. Wrong handling of the solder paste may also cause this, such as insufficient thawing time of the paste.

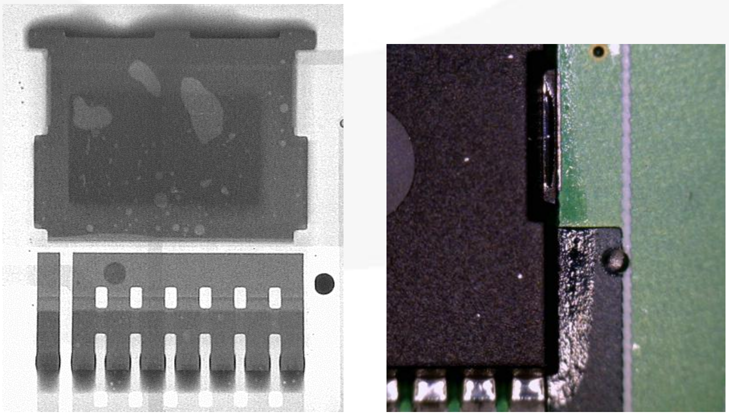

(a) (b)

Figure 9. (a) X-Ray Image of Board Mounted TO-LL with Solder Beads Under the Package, (b) Solder Bead

General Rework Guideline

The recommended board rework methodology for this package is as follows.

STEP	TOOLS/EQUIPMENT	GUIDELINE / PROCEDURE
Baking	Oven	Baking of the board assembly maybe necessary depending on the moisture sensitivity of the board and the other surrounding components. The purpose is to eliminate absorbed moisture on these parts, and prevent the damaging effects when subjected to sudden ramp of temperature during removal of the component.
Component Removal	Rework Station Heat Gun	Secure the board on the rework station. Preheat the whole board to minimize warpage of the board when high temperature is applied on the component to be removed. Apply heat into the component using a heat gun, once the solder joint has melted; remove the component immediately using a vacuum nozzle. It is important to take note that applying too much heat on the board can also affect the surrounding components, thus it is important to do this process quickly. It is also important not to expose the TO-LL package to too much heat so as not to damage the component. This will help us understand failure mechanism should this component be returned to us for analysis.
Land Preparation	Solder Wick Soldering Iron Continuous Vacuum DesolderingSystem Sesoldering Tip	After the component is removed, remove excess solder left on the lands using a continuous vacuum desoldering system and soldering tip. Soldering iron and solder wicking material can also be used to do this.
Component Installation	Mini Stencil Solder Paste Dispenser Low Magnification Microscope Pick and Place Machine Reflow Oven	After preparing the lands, install a new component into the board. The old component should not be used. Installation of the new component follow these steps: Solder paste printing/dispensing – screen print solder paste on the lands. Use a mini stencil to do this. The stencil should have the same aperture size and thickness with the one used in the whole board assembly. A solder paste dispensing system can also be used to put paste on the lands. Inspection – perform inspection to check if sufficient paste is printed on the board. Component placement – place the component onto the board either manually or the work station. Reflow – reflow the board using the standard reflow profile established for the whole board assembly.
Inspection	X-Ray Machine Low Magnification Microscope	Inspect the component after reflow using an X-ray machine to check solder joint anomalies like solder bridging, beading and voids.

DISCLAIMER
FAIRCHILD SEMICONDUCTOR RESERVES THE RIGHT TO MAKE CHANGES WITHOUT FURTHER NOTICE TO ANY PRODUCTS HEREIN TO IMPROVE RELIABILITY, FUNCTION, OR DESIGN. FAIRCHILD DOES NOT ASSUME ANY LIABILITY ARISING OUT OF THE APPLICATION OR USE OF ANY PRODUCT OR CIRCUIT DESCRIBED HEREIN; NEITHER DOES IT CONVEY ANY LICENSE UNDER ITS PATENT RIGHTS, NOR THE RIGHTS OF OTHERS.

LIFE SUPPORT POLICY
FAIRCHILD'S PRODUCTS ARE NOT AUTHORIZED FOR USE AS CRITICAL COMPONENTS IN LIFE SUPPORT DEVICES OR SYSTEMS WITHOUT THE EXPRESS WRITTEN APPROVAL OF THE PRESIDENT OF FAIRCHILD SEMICONDUCTOR CORPORATION.
As used herein:

1. Life support devices or systems are devices or systems which, (a) are intended for surgical implant into the body, or (b) support or sustain life, or (c) whose failure to perform when properly used in accordance with instructions for use provided in the labeling, can be reasonably expected to result in significant injury to the user.
2. A critical component is any component of a life support device or system whose failure to perform can be reasonably expected to cause the failure of the life support device or system, or to affect its safety or effectiveness.

ON Semiconductor and ON are trademarks of Semiconductor Components Industries, LLC dba ON Semiconductor or its subsidiaries in the United States and/or other countries. ON Semiconductor owns the rights to a number of patents, trademarks, copyrights, trade secrets, and other intellectual property. A listing of ON Semiconductor's product/patent coverage may be accessed at www.onsemi.com/site/pdf/Patent−Marking.pdf. ON Semiconductor reserves the right to make changes without further notice to any products herein. ON Semiconductor makes no warranty, representation or guarantee regarding the suitability of its products for any particular purpose, nor does ON Semiconductor assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation special, consequential or incidental damages. Buyer is responsible for its products and applications using ON Semiconductor products, including compliance with all laws, regulations and safety requirements or standards, regardless of any support or applications information provided by ON Semiconductor. "Typical" parameters which may be provided in ON Semiconductor data sheets and/or specifications can and do vary in different applications and actual performance may vary over time. All operating parameters, including "Typicals" must be validated for each customer application by customer's technical experts. ON Semiconductor does not convey any license under its patent rights nor the rights of others. ON Semiconductor products are not designed, intended, or authorized for use as a critical component in life support systems or any FDA Class 3 medical devices or medical devices with a same or similar classification in a foreign jurisdiction or any devices intended for implantation in the human body. Should Buyer purchase or use ON Semiconductor products for any such unintended or unauthorized application, Buyer shall indemnify and hold ON Semiconductor and its officers, employees, subsidiaries, affiliates, and distributors harmless against all claims, costs, damages, and expenses, and reasonable attorney fees arising out of, directly or indirectly, any claim of personal injury or death associated with such unintended or unauthorized use, even if such claim alleges that ON Semiconductor was negligent regarding the design or manufacture of the part. ON Semiconductor is an Equal Opportunity/Affirmative Action Employer. This literature is subject to all applicable copyright laws and is not for resale in any manner.

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:
Literature Distribution Center for ON Semiconductor
19521 E. 32nd Pkwy, Aurora, Colorado 80011 USA
Phone: 303−675−2175 or 800−344−3860 Toll Free USA/Canada
Fax: 303−675−2176 or 800−344−3867 Toll Free USA/Canada
Email: orderlit@onsemi.com

N. American Technical Support: 800−282−9855 Toll Free USA/Canada
Europe, Middle East and Africa Technical Support:
Phone: 421 33 790 2910
Japan Customer Focus Center
Phone: 81−3−5817−1050

ON Semiconductor Website: **www.onsemi.com**

Order Literature: http://www.onsemi.com/orderlit

For additional information, please contact your local Sales Representative

© Semiconductor Components Industries, LLC