

Printed Circuit Board Design Techniques for DS40MB200

National Semiconductor
Application Note 1398
TK Chin
October 2005



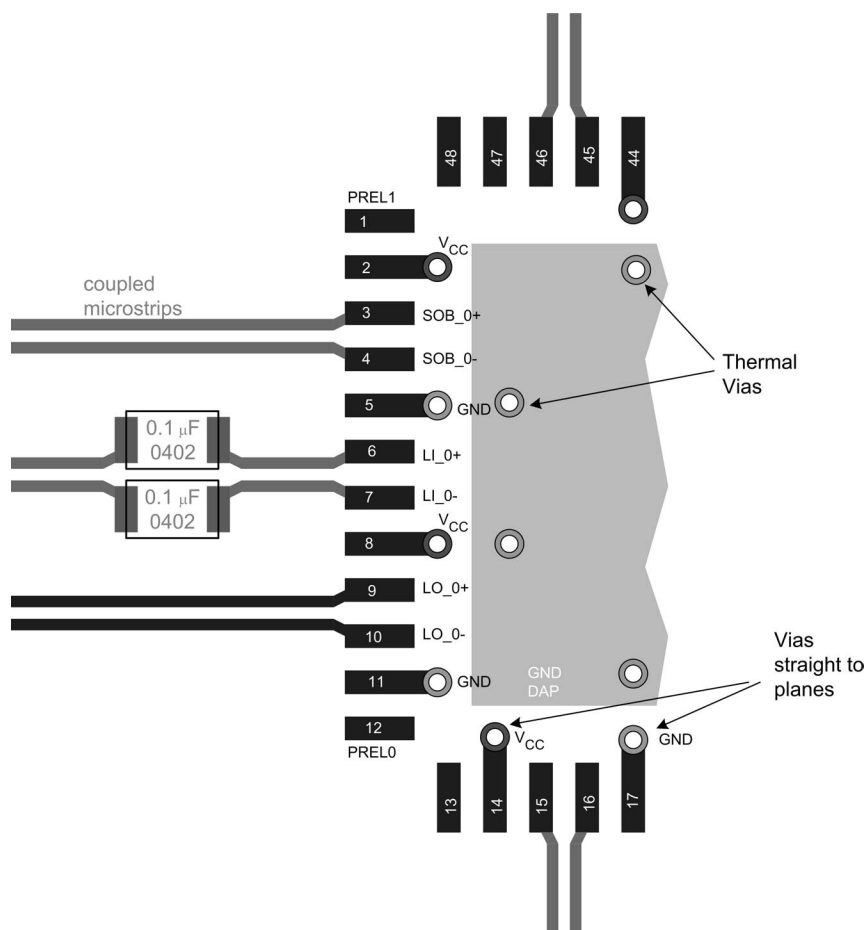
1.0 Introduction

The DS40MB200 is a high speed 2:1 multiplexer and 1:2 fan-out repeater designed to support redundancy and extend copper backplanes up to 4 Gb/s. The output driver features fast edge rates, typically about 80 ps. Each input stage features fixed equalizer, followed by a high-gain limiting amplifier that re-shapes the output waveforms. The DS40MB200 accepts input signal as low as 100 mV (peak-to-peak). The high speed, high gain and fast edge rate attributes of any high-speed circuit require users to use high speed printed circuit board techniques to achieve good performance. This application note outlines the techniques to achieve this goal.

choice for high speed communication designs. The high speed differential IO's are assigned to the center portion of the package such that each differential pair is sandwiched by a power or ground pin, providing isolation and ground shielding. Coupled microstrip board traces are the preferred transmission line structures for best signal fidelity. The optimum trace width is 10 mils, being equal to the 10-mil landing pad of the LLP-48 package. Finer trace width, such as 5 mils, can also be used with slightly higher impedance mismatch caused by changes in trace width between the traces and the pads. The differential board traces should be routed with constant spacing to ensure impedance uniformity along the length of the traces. *Figure 1* shows a partial layout arrangement with 5-mil trace width.

2.0 Differential Pairs

The DS40MB200 uses a 48-lead lead-less LLP package. The LLP offers low package parasitics and is the package of



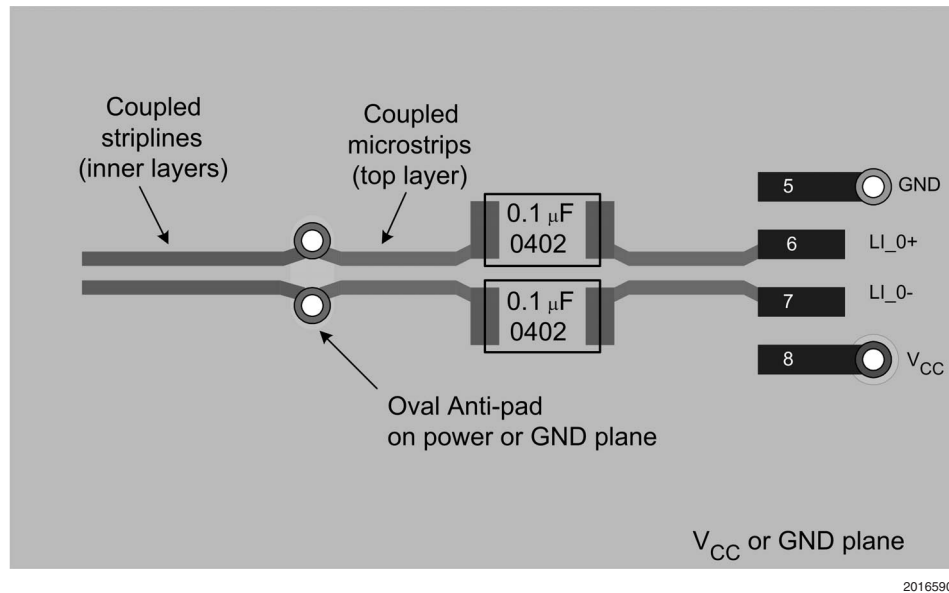
20165901

FIGURE 1. DS40MB200 Routing Example Using Coupled Microstrips

2.0 Differential Pairs (Continued)

In high routing density boards, coupled striplines are commonly used, but requires the use of plated through holes. These plated through holes introduce parasitic capacitance and may degrade signal fidelity. Small via, with less than 8

mils hole size, should be used to minimize mutual capacitive coupling between the via and nearby ground or power planes. Differential via structure with shared oval-shaped anti-pad can further reduce parasitic capacitance. *Figure 2* shows a pair of differential vias with shared oval anti-pad.

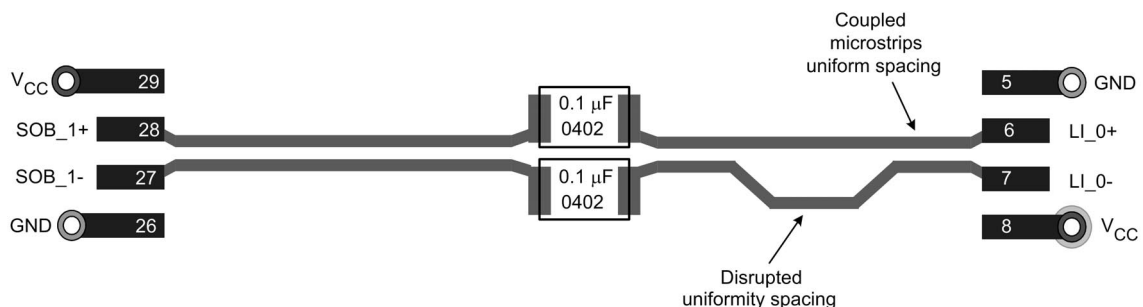


20165902

FIGURE 2. Differential Vias with Oval Anti-Pad

The electrical length of each differential pair should be well matched. As a rule of thumb, the delay skew of the complementary signals are matched within a small fraction of the signal's edge rate. With an edge rate of about 80 ps, it is acceptable to have a delay skew within 10 ps. For a FR4 board, matching trace lengths within 50 mils achieves less than 10 ps skew between the complementary signals of the differential pair. It is important to note that it is the matched electrical length instead of matched physical length that matters. Identical geometries should be used for the complementary signals, and their trace widths in each signal layer should be matched.

In adjusting the trace lengths of a pair of coupled board traces, never disrupt the uniform spacing of the differential pair. *Figure 3* shows a bad example of matching trace lengths. Disrupting the uniform spacing of the coupled differential pair will create impedance discontinuity and impact signal quality. *Figure 4* shows a good example of matching trace lengths. In this example, the lengths of the uncoupled segments of the differential pair are adjusted to achieve the desired match lengths. The coupled board traces maintain their uniform spacing, while the trace width of the uncoupled traces are slightly widened to maintain the same characteristic impedance.



20165903

FIGURE 3. Bad Example Of Matching Trace Length Of Coupled Board Traces

2.0 Differential Pairs (Continued)

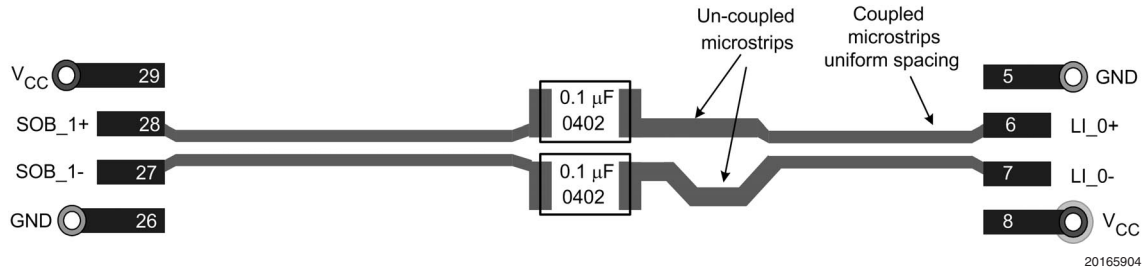


FIGURE 4. Good Example Of Matching Trace Length Of Coupled Board Traces

3.0 Components' Landing Pads

AC coupling capacitors should be placed on the receive inputs of the DS40MB200. The transition time of the bit stream at the receive side is significantly slowed down by interconnect, making it more tolerant to impedance mismatch caused by the AC coupling capacitors and the parasitic capacitance caused by their landing pads. Small physi-

cal size, surface mount capacitors, such as 0402, should be used to reduce impedance mismatch caused by the landing pads of the capacitors.

When fine trace width is used with large components, the landing pads of the components are huge relative to the trace width. *Figure 5* shows an example of 5-mil traces routing to an edge connector with 20 mil pads.

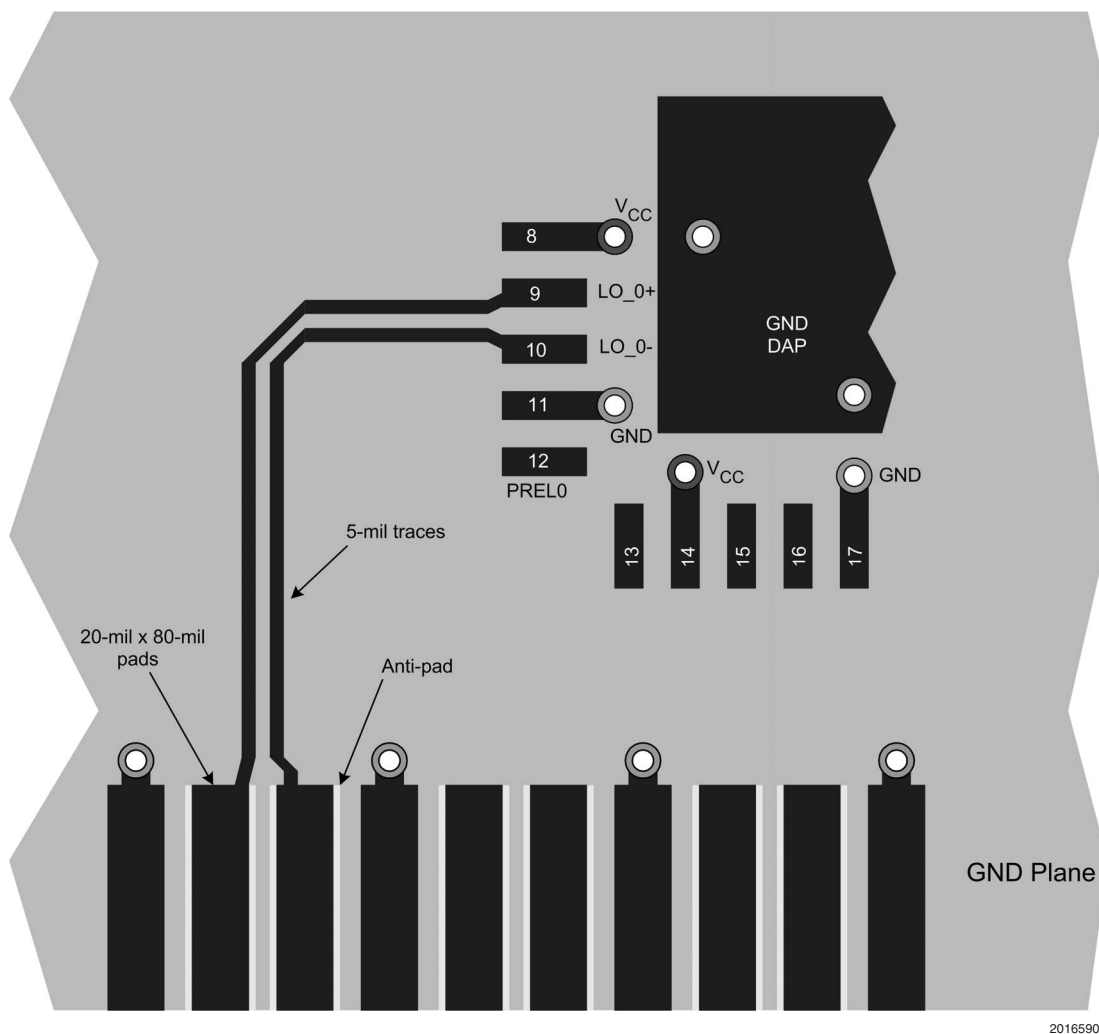


FIGURE 5. Fine Board Traces Used With Large Component Landing Pads

3.0 Components' Landing Pads (Continued)

The huge landing pads of the edge connector effectively form a short segment of wide trace, significantly lowering the impedance and introduce relatively large excess capacitance between the pad and the ground plane immediately

below it. *Figure 6* depicts the cross-sectional view of the board showing the effect of impedance drop from landing pads.

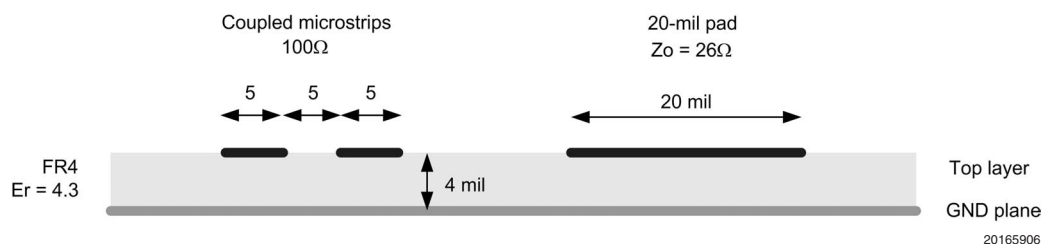


FIGURE 6. Cross-Section Of Board Trace And Component Pad

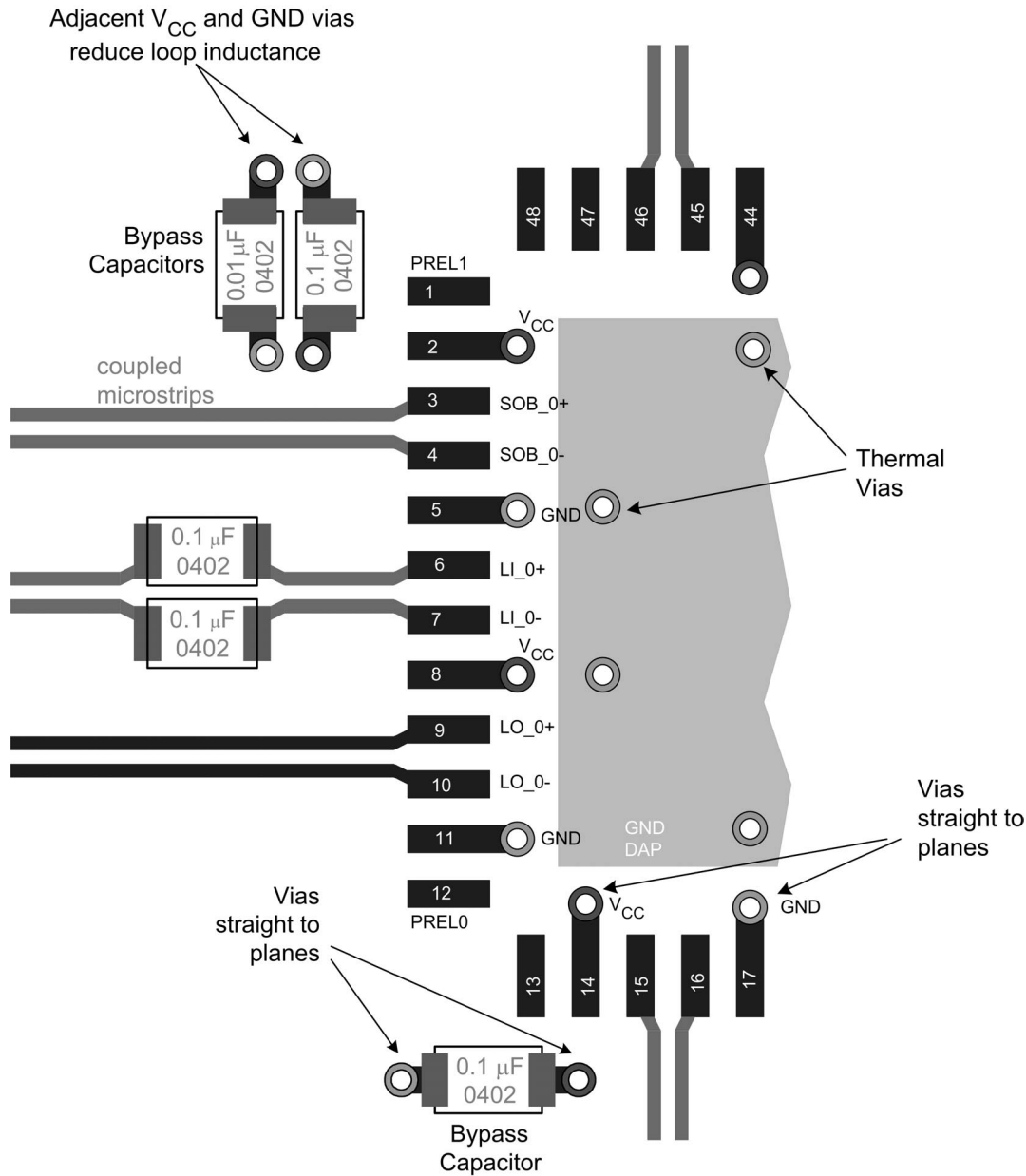
We can use anti-pads on the power and ground planes below the landing pads. This technique is used to reduce the parasitic capacitance formed between the pad and the nearby power or ground plane. *Figure 5* illustrates anti-pads used with an edge connector to minimize excess capacitance. A 3-D electromagnetic field solver is usually used to determine the size of the anti-pad to optimize compensation.

be used to connect a power or ground pin to nearby power or ground plane. Ideally, vias are placed tangent to the IC pads to avoid adding trace inductance. Placing power planes closer to the top of the board reduces the length and inductance of the vias.

4.0 Power Pins

Each power or ground lead of the DS40MB200 should be connected to the power or ground plane through a low impedance path. For best results, one or more vias should

4.0 Power Pins (Continued)



20165907

FIGURE 7. V_{CC} and GND Hook-Up For The DS40MB200

Bypass capacitors should be placed close to VCC pins. They can be conveniently placed at four corners of the LLP package. Small physical size capacitors, such as 0402, X7R, surface mount capacitors should be used to minimize body inductance of capacitors. Each bypass capacitor is connected to the power and ground plane through vias tangent to the pads of the capacitor. Figure 7 shows V_{CC} and GND hook-up for the DS40MB200 and placement of bypass capacitors.

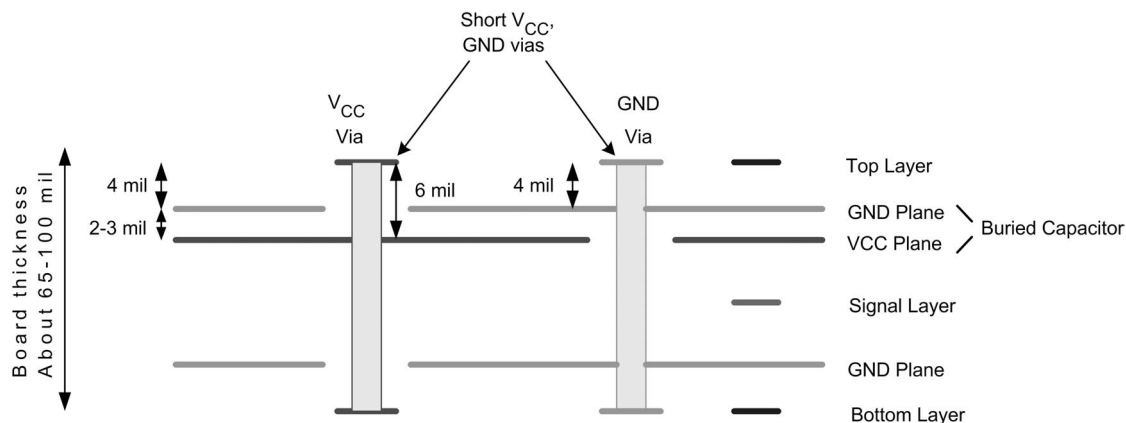
An X7R surface mount capacitor of size 0402 has about 0.5 nH of body inductance. At frequencies above 20–30 MHz, X7R capacitors behave as low impedance inductors. To extend the operating frequency range of these capacitors to

a few hundred MHz, an array of different capacitance ranging from 100 pF, 1 nF, 0.01 μF , 0.1 μF are commonly used in parallel. A more effective bypass capacitor can be built using sandwiched layers of power and ground planes at a separation of 2–3 mils. An ideal high frequency bypass capacitor is formed. With a 2-mil FR4 dielectric, there is about 500 pF capacitance per square inch. Figure 8 depicts a PCB stack-up with V_{CC} -GND planes placed on the top side of the board. This stack-up arrangement provides a high frequency “buried” capacitor, and lowers the inductance of power and ground via by reducing the lengths of the vias.

The center dap of the package should be connected to ground plane through an array of vias. The array of via

4.0 Power Pins (Continued)

reduces the effective inductance to ground, and also offer enhanced thermal performance for the package.



20165908

FIGURE 8. PCB Stack-Up With "Buried" Capacitor And Short V_{CC} /GND Vias

5.0 Conclusion

High speed printed circuit boards require designers to go beyond connectivity. Multi-gigabit boards require attention to board geometry details. These details include dimensions of trace width, component pads, plated-through holes, anti-pads, board thickness and board stack-up. Board material properties such as dielectric constant and loss tangent are important considerations in choosing proper board structures. The techniques described in this application notes apply to multi-gigabit board design, with examples illustrated for DS40MB200 mux/buffer. In a multi-gigabit board design, board designers have to find the unwanted L's and C's and devise geometries to overcome their impacts. Very often, a 2-D electromagnetic field solver is used to predict the elec-

trical behavior of board traces, and a 3-D field solver is needed to deal with 3-dimensional geometries such as via and anti-pad. With a little extra time spent on attention to details, the reward is a high performance board running at multi-gigabit and beyond.

Reference

Datasheet of DS40MB200 Dual 4Gb/s mux/buffer, www.national.com

AN-1389—Setting Pre-Emphasis Level for DS40MB200 Dual 4 Gb/s Mux/Buffer

AN-1187—Leadless Leadframe Package (LLP)

National does not assume any responsibility for use of any circuitry described, no circuit patent licenses are implied and National reserves the right at any time without notice to change said circuitry and specifications.

For the most current product information visit us at www.national.com.

LIFE SUPPORT POLICY

NATIONAL'S PRODUCTS ARE NOT AUTHORIZED FOR USE AS CRITICAL COMPONENTS IN LIFE SUPPORT DEVICES OR SYSTEMS WITHOUT THE EXPRESS WRITTEN APPROVAL OF THE PRESIDENT AND GENERAL COUNSEL OF NATIONAL 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, and whose failure to perform when properly used in accordance with instructions for use provided in the labeling, can be reasonably expected to result in a 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.

BANNED SUBSTANCE COMPLIANCE

National Semiconductor manufactures products and uses packing materials that meet the provisions of the Customer Products Stewardship Specification (CSP-9-111C2) and the Banned Substances and Materials of Interest Specification (CSP-9-111S2) and contain no "Banned Substances" as defined in CSP-9-111S2.

Leadfree products are RoHS compliant.



National Semiconductor
Americas Customer
Support Center
Email: new.feedback@nsc.com
Tel: 1-800-272-9959

National Semiconductor
Europe Customer Support Center
Fax: +49 (0) 180-530 85 86
Email: europe.support@nsc.com
Deutsch Tel: +49 (0) 69 9508 6208
English Tel: +44 (0) 870 24 0 2171
Français Tel: +33 (0) 1 41 91 8790

National Semiconductor
Asia Pacific Customer
Support Center
Email: ap.support@nsc.com

National Semiconductor
Japan Customer Support Center
Fax: 81-3-5639-7507
Email: jpn.feedback@nsc.com
Tel: 81-3-5639-7560

www.national.com