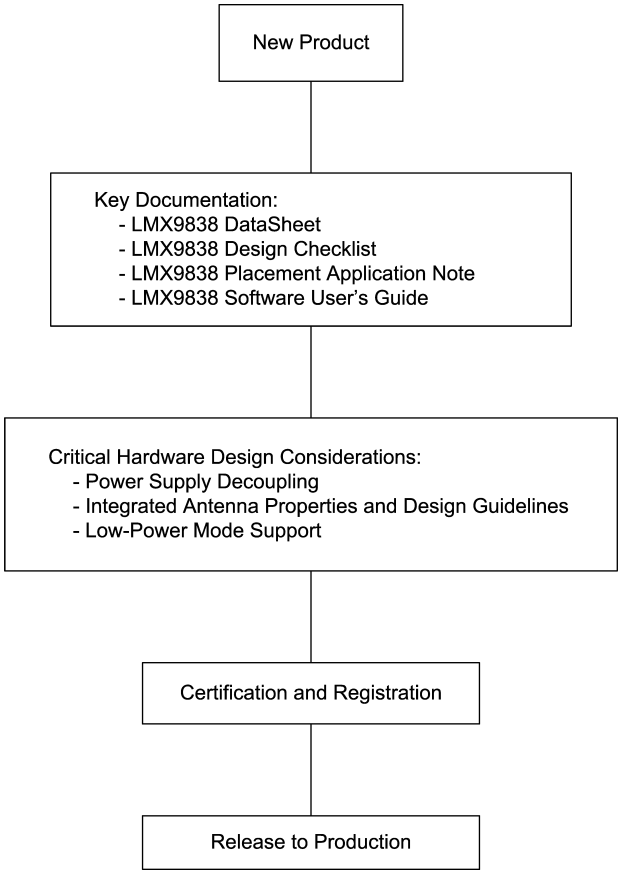
LMX9838 Design Guide

National Semiconductor
Application Note 1587
Sebastien Mathieu
September 2007

1.0 Scope

This application note provides guidelines for designing with the LMX9838 Bluetooth Serial Port Module, particularly for critical aspects of the PCB layout. The guidelines have been organized in a simplified flow that walks you through the first steps on what documentation to review, key design considerations and then the final steps of certification and production.

Since the antenna is integrated in to the module, the PCB layout and placement of the LMX9838 is critical in order to maximize performance (i.e. range and data throughput). The key sections to review before laying out a PCB are Section 5.0 and the document [4].



National Semiconductor ia a registered trademark of National Semiconductor Corporation.
CompactRISC is a trademark of National Semiconductor. Bluetooth is a trademark of Bluetooth SIG, Inc. and is used under license by National Semiconductor.
For a complete listing of National Semiconductor trademarks, please visit www.national.com/trademarks.

Table Of contents

2.0 Device introduction

The National Semiconductor LMX9838 Bluetooth Serial Port module is a fully integrated Bluetooth 2.0 base band controller, 2.4 GHz radio, crystal, antenna, LDO and discreet; combined to form a complete small form factor Bluetooth node.

All hardware and firmware is included to provide a complete solution from antenna through the complete lower and upper layers of the Bluetooth stack, up to the application including the Generic Access Profile (GAP), the Service Discovery Application Profile (SDAP), and the Serial Port Profile (SPP). The module includes a configurable service database to fulfill service requests for additional profiles on the host. Moreover, the LMX9838 is pre-qualified as a Bluetooth subsystem. Conformance testing through the Bluetooth qualification program enables a short time to market after sys-tem integration by insuring a high probability of compliance and interoperability.

Based on National's CompactRISCTM 16-bit processor architecture and Digital Smart Radio technology, the LMX9838 is optimized to handle the data and link management processing requirements of a Bluetooth node.

The firmware supplied in the on-chip ROM memory offers a complete Bluetooth (v2.0) stack including profiles and command interface. This firmware features point-to-point and point-to-multipoint link management supporting data rates up to the theoretical maximum over RFComm of 704 kbps. The internal memory supports up to 7 active Bluetooth data links and one active SCO link.

The on-chip Patch RAM provided for lowest cost and risk, allows the flexibility of firmware upgrade.

The module is lead free and RoHS (Restriction of Hazardous Substances) compliant. For more information on those quality standards, please visit our green compliance website at http://www.national.com/quality/green/

3.0 Product design flow

3.1 DOCUMENTATION

- LMX9838 Datasheet [1]
- LMX9838 Design Guide [2]
- LMX9838 Software User's Guide [3]
- LMX9838 Placement Application Note[4]

3.2 KEY DESIGN CONSIDERATIONS

- Power supply decoupling - refer to [1]
- Integrated Antenna Properties and Design Guidelines - refer to [4]
- Low power mode - refer to [1]

3.3 CERTIFICATION AND REGISTRATION

- Certification - refer to Section 7.0

4.0 Design flow details

4.1 DOCUMENTATION STRUCTURE

4.1.1 Design documents

1) LMX9838 Design Guide [2] (this document): contains design key points and recommendations for a system implementing the LMX9838.
2) LMX9838 Datasheet [1]: contains key performance details regarding the device, pin description, functional details, programming details, package information and applications information.
3) LMX9838 Software User's Guide [3]: The document is a reference for implementing the LMX9838 module into a system. A getting started session gives a very detailed entry point for starting software development. The advance usage section describes all features and configuration parameters in detail and gives example for using the LMX9838 as active Bluetooth node. Finally all commands and events are listed and explained in the command section.

5.0 PCB Design details

The following section discusses the layout requirements for the Printed Circuit Board. Figure 2, Figure 3 and Figure 4 are taken from LMX9838 UART Dongle evaluation design.

5.1 MODULE PLACEMENT

The LMX9838 has a built-in antenna which allows easy integration of the module in to a customer's application. There are however some recommended guidelines to follow for optimum performance.

Being small means that the antenna can be easily detuned if it is not placed correctly. The antenna has been optimised to work with FR4 material without metal layers underneath. This means that FR4 material only should be placed under the antenna part of the module as shown on Figure 1. Also we recommend having FR4 material only (without any metal layer) on all layers underneath the antenna part of the module. Figure 1 also shows the optimal module placement on a PCB giving some dimension guidance.

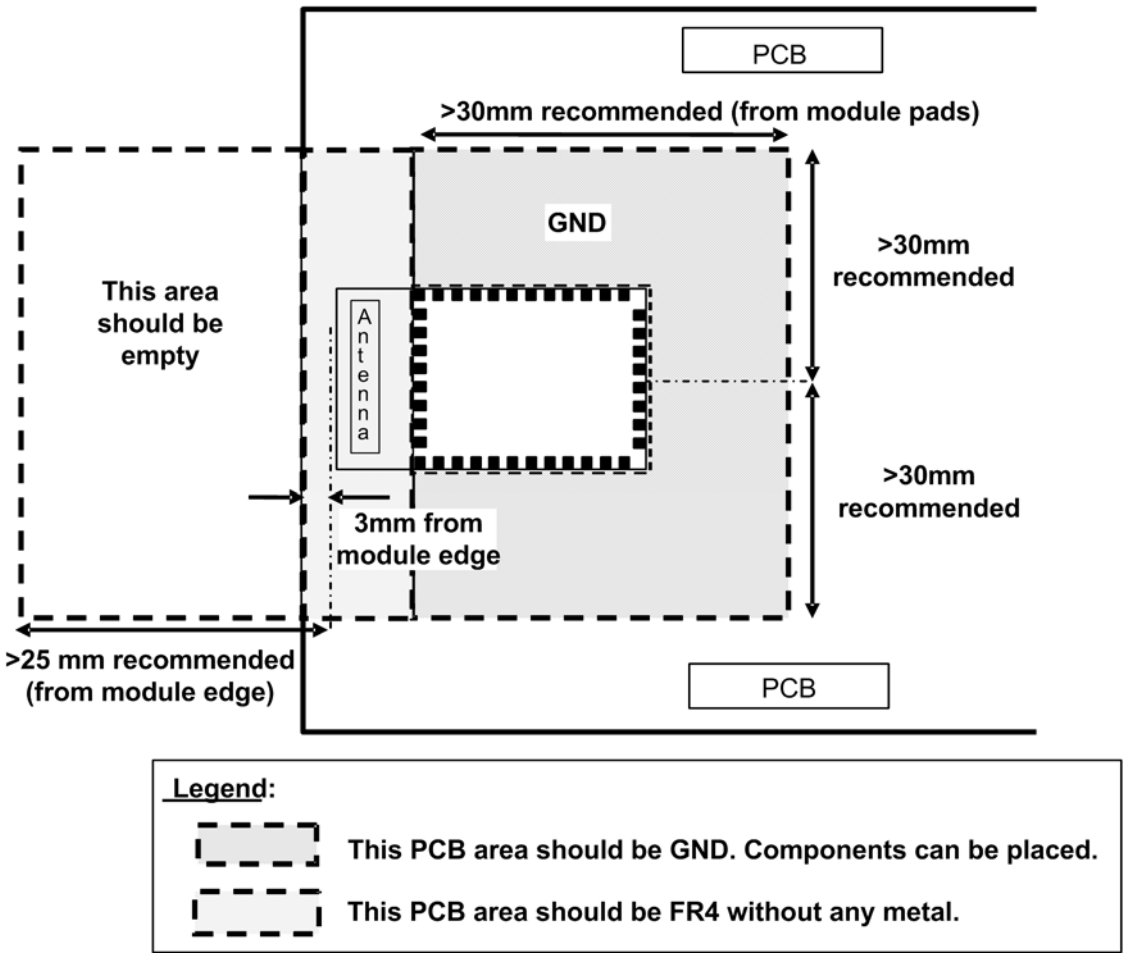


Figure 1. Module placement guideline

For more details on the antenna and its implementation within the LMX9838, refer to [4].

5.2 SCHEMATIC

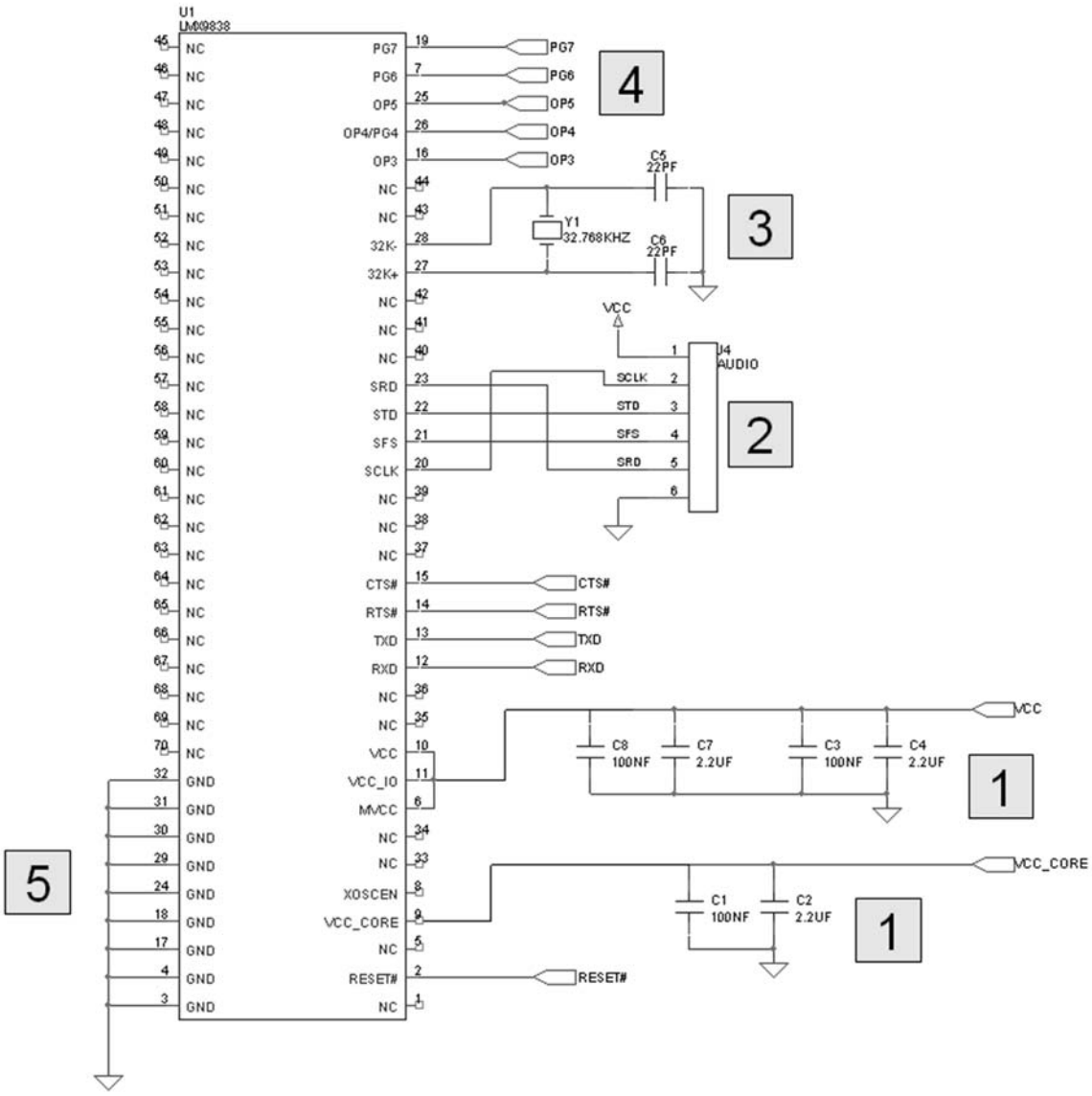


Figure 2. LMX9838 implementation example

5.3 LAYOUT

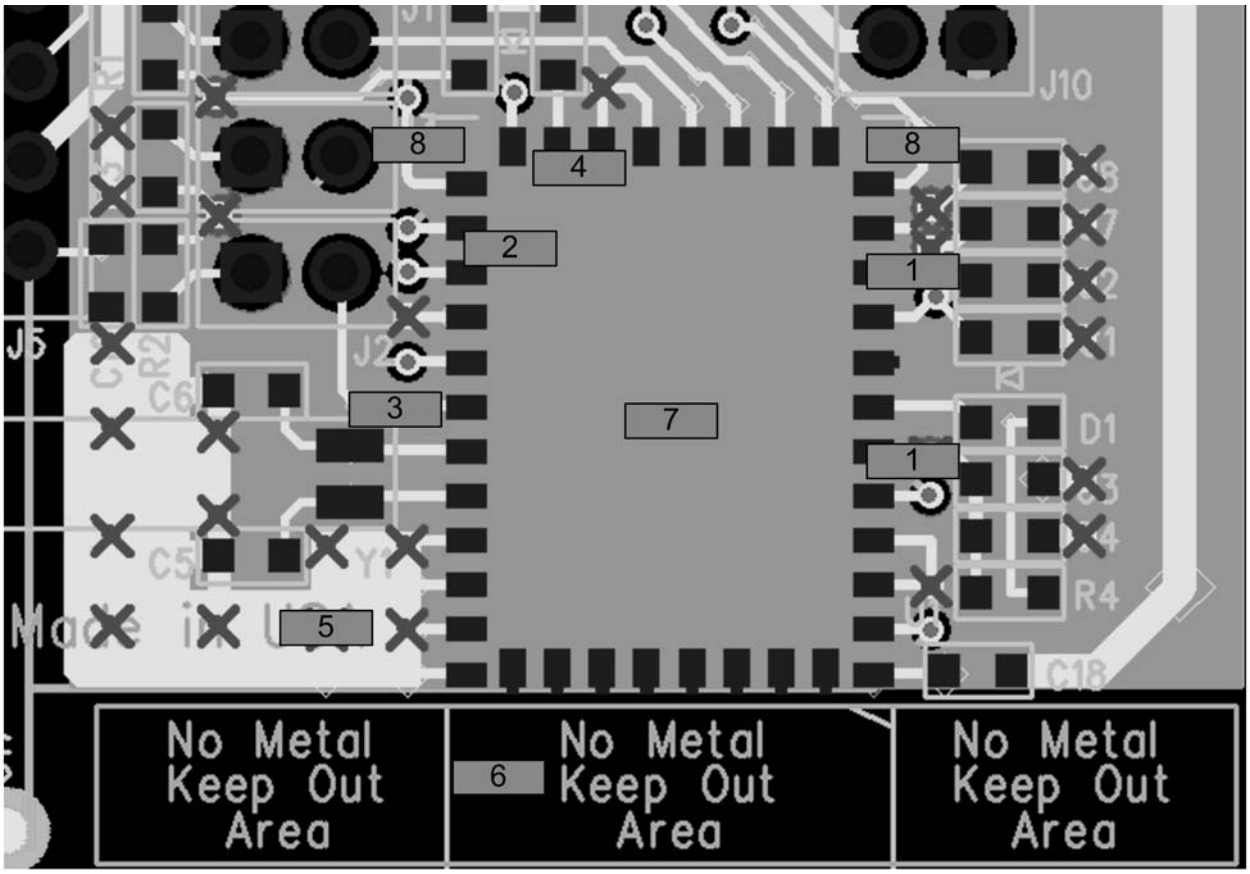


Figure 3. LMX9838 module placement example

Key points to consider for the schematic and layout design are listed below:

Table 1. Design key points

1	Decoupling capacitors
2	Advanced audio interface connect to PCM codec or leave open
3	32Khz crystal for low power mode, see [1]
4	Frequency baud rate selector and GPIO's
5	Ground connections and Ground vias
6	FR4 only area. Keep metal out of this area.

Table 2. Soldering recommendation

7	Inner area of module shall be left empty, do not solder the inner pads of the module
8	Vias should be placed outside the module outline to avoid raising the package corner, which will prevent the package from sitting flat and creating problems for soldering
-	All traces connecting to the soldering pads should be the same width in order to equalize the metal area for soldering. Specifically connections to pads 3, 4, 29, 30, 31 and 32. Tangential connections to the pads should be avoided

Table 2. Soldering recommendation

-	The solder paste stencil artwork should be arranged so that the pad openings are 0.1mm outwards, generating what is called a "stencil pullback". This helps center the package during reflow, creates an external "fillet" and prevents any shorts on the corner pads by increasing the space between paste openings
-	The silkscreen outline for the module should be placed outwards so that the ink does not raise the component up and also so that the ink does not get too close to the soldering pads
-	Designators should not be placed under the module outline

Table 3. FR4 PCB Thickness recommendation

-	It is recommended to use the FR4 PCB with a thickness between 30 mil (0.76mm) and 100 mil (2.54mm). Optimum antenna performance is achieved using 60 mil (1.52mm).

Table 4. Proximity recommendation

-	It is recommended to keep metal objects no closer than 30mm away from the antenna end of the LMX9838 module to minimize field pattern modification and performance degradation.
-	Do not use a shielded RF enclosure. The enclosure must allow the antenna to radiate thru.

5.4 DESIGN CONSIDERATIONS

- Ideally a decoupling capacitor pair should be used per VCC pin of the device. It is recommended to use a 2.2uF and 100nF together. Single capacitors maybe used in some cases without significantly degrading performance. The capacitors should be placed as close as is physically possible to the module's respective pins.
- One solid Bluetooth ground plane for all ground pins on the module. It is recommended to place this ground plane on the layer immediately under the top layer. The size of the ground plane only has to be as large as the module and the modules supporting components.
- The Bluetooth ground plane is typically separated from the other system ground planes to reduce coupling of noise in to the Bluetooth module or from the Bluetooth module in to sensitive circuitry.

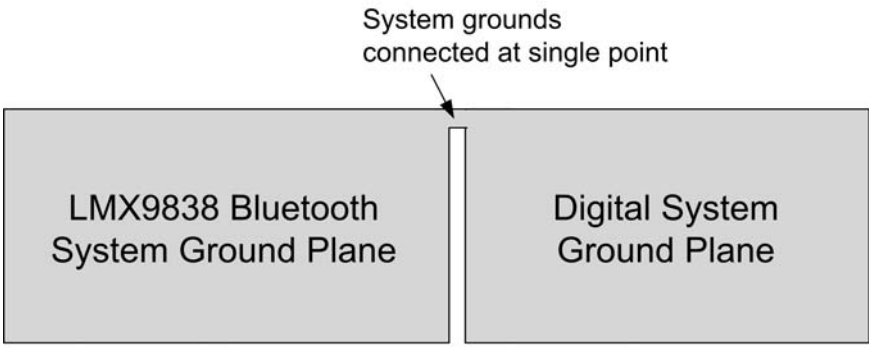


Figure 4. System ground planes

For example, if there is additional digital circuitry sharing the same printed circuit board then it is advisable to create a separate digital ground plane and Bluetooth ground plane as in Figure 4. The ground planes should not overlap and should be positioned as close as possible to each other. They should be connected together at a single point.

- For the internal antenna in the module to operate to its maximum efficiency then the PCB area around the antenna should be cut away. It is recommended to place the end of the module flush to the edge of the PCB and to route the PCB out around the antenna end, see Figure 1.

- Digital signals to/from the module are typically routed on the top layer of the board but should be kept away from the antenna end of the module to avoid cross-talk problems. Digital signals can be routed underneath the module but they must be routed under the ground plane (to reduce cross-talk in to the module) and then come through the ground plane to the top layer. Having too many vias in a close area perforating the ground plane will reduce the effectiveness of the ground plane. Therefore use them sparingly.
- A 32.768KHz oscillator maybe used to support lower power mode for reduced power con-sumption. The modular structure of the LMX9838 allows the firmware to power down unused modules such as the radio LLC and UART interface. Refer to [1] for further information.

6.0 Most common design errors

Since the LMX9838 is a highly integrated module with internal tuned crystal and antenna, there is little room for error in terms of PCB layout. However, here are mentioned some common customer errors that have occurred in the past.

Non Compact Design

When there is more than enough space on the PCB which must be of a certain shape and size to conform to the application, customers tend to spread the external components over a larger area than is required. This is a mistake, though there maybe space available, the external components such as decoupling capacitors, etc must be placed as close to the LMX9838 as possible otherwise this will result in degraded performance due to unnecessary line parasitic. Long lines also cause cross-coupling and spurious emissions.

Poor grounding

Not using enough ground vias or ground lines that are too thin and long is a common error. Long tracks are inductive causing the grounding point to sit at a higher voltage (at 2.4GHz) than the ground plane.

Not enough clearance around internal antenna

The internal antenna must be kept clear of any components and especially copper ground plane that can detune it (see Section 5.0). A detuned antenna will result in bad return loss into it and ultimately decreased range of operation.

7.0 Certification and registration

7.1 BLUETOOTH CONFORMANCE TESTING

The LMX9838 has been fully certified to Bluetooth core specification 2.0 through SPP (Serial Port Profile) as an end-product. If SPP is used then no further testing is required, however, if the customer decides to implement another profile then the customer should contact the BQB (Bluetooth Qualification Body) to see if any additional interoperability testing is required.

For example, if a customer uses the LMX9838 as pure slave device, with no additional profile on top, there will be no additional qualification required and therefore no testing costs AND no listing fee. If the customer uses the LMX9838 as active master, it might be necessary to at least do some interoperability tests, to make sure the interface is used correctly - this decision will depend on the BQB. Also, if additional profiles are used on the host, it needs to be qualified as well.

For more detailed information on certification and regulatory testing refer to [5].

8.0 Bibliography

[1] LMX9838 Datasheet: Technical document explaining the LMX9838 features, pinout, caracteristics, and soldering informations.

[2] AN-1587: LMX9838 Design Guide: This document.

[3] AN-1699: LMX9838 Software User's Guide: Document exploring in details the LMX9838 firmware functionalities, the embedded language interface, and a few interesting scenarios such as incoming link, multiple links, audio links, DUN profile, Headset and Handsfree profiles.

[4] AN-1701: LMX9838 Placement Application Note: Document concentrating on the antenna backgrounds demonstrating the best physical placement of the LMX9838 on a printed circuit board.

[5] AN-1709: LMX9838 Qualification Document: Gives a brief overview of the qualification steps required for a converting a LMX9838 design to an end product.

NOTES

THE CONTENTS OF THIS DOCUMENT ARE PROVIDED IN CONNECTION WITH NATIONAL SEMICONDUCTOR CORPORATION ("NATIONAL") PRODUCTS. NATIONAL MAKES NO REPRESENTATIONS OR WARRANTIES WITH RESPECT TO THE ACCURACY OR COMPLETENESS OF THE CONTENTS OF THIS PUBLICATION AND RESERVES THE RIGHT TO MAKE CHANGES TO SPECIFICATIONS AND PRODUCT DESCRIPTIONS AT ANY TIME WITHOUT NOTICE. NO LICENSE, WHETHER EXPRESS, IMPLIED, ARISING BY ESTOPPEL OR OTHERWISE, TO ANY INTELLECTUAL PROPERTY RIGHTS IS GRANTED BY THIS DOCUMENT.

TESTING AND OTHER QUALITY CONTROLS ARE USED TO THE EXTENT NATIONAL DEEMS NECESSARY TO SUPPORT NATIONAL'S PRODUCT WARRANTY. EXCEPT WHERE MANDATED BY GOVERNMENT REQUIREMENTS, TESTING OF ALL PARAMETERS OF EACH PRODUCT IS NOT NECESSARILY PERFORMED. NATIONAL ASSUMES NO LIABILITY FOR APPLICATIONS ASSISTANCE OR BUYER PRODUCT DESIGN. BUYERS ARE RESPONSIBLE FOR THEIR PRODUCTS AND APPLICATIONS USING NATIONAL COMPONENTS. PRIOR TO USING OR DISTRIBUTING ANY PRODUCTS THAT INCLUDE NATIONAL COMPONENTS, BUYERS SHOULD PROVIDE ADEQUATE DESIGN, TESTING AND OPERATING SAFEGUARDS.

EXCEPT AS PROVIDED IN NATIONAL'S TERMS AND CONDITIONS OF SALE FOR SUCH PRODUCTS, NATIONAL ASSUMES NO LIABILITY WHATSOEVER, AND NATIONAL DISCLAIMS ANY EXPRESS OR IMPLIED WARRANTY RELATING TO THE SALE AND/OR USE OF NATIONAL PRODUCTS INCLUDING LIABILITY OR WARRANTIES RELATING TO FITNESS FOR A PARTICULAR PUR

POSE, MERCHANTABILITY, OR INFRINGEMENT OF ANY PATENT, COPYRIGHT OR OTHER INTELLECTUAL PROPERTY RIGHT.

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:

Life support devices or systems are devices 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. A critical component is any component in 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.

National Semiconductor and the National Semiconductor logo are registered trademarks of National Semiconductor Corporation. All other brand or product names may be trademarks or registered trademarks of their respective holders.

Copyright© 2007 National Semiconductor Corporation

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

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: +49 (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