



Application Note AN004

Design Guide for the AS1113 / AS1124 PoE Powered Devices

Revision 0.8 May 2007

Confidential

CONTENTS

Contents	2
Introduction	3
About this Design Guide	3
Schematic Considerations	3
Ethernet Line Side	3
DC-DC Converter	3
802.3af/at COMPLIANCE	3
Component Placement Considerations	4
Ethernet Signals	4
DC-DC Converter	4
Layout Considerations	5
Ethernet Signals	5
DC-DC Converter	5
Thermal Pad	6
Ground layers	6
Layout Example	6
Appendix A: Example Schematic	9
Contact Information	11
Important Notices	11
Legal Notice	11
Life and Safety Policy	11

INTRODUCTION

ABOUT THIS DESIGN GUIDE

This design guide provides some general guidelines to be used when you are designing with the AS1113 & AS1124 family of Powered Device (PD) Controllers. It is to be used in conjunction with the datasheet and reference design. The design guide provides guidelines for adapting the reference design to a customer's system. If there are any questions of concerns please contact the Akros applications team through your sales contact or sales@akrossilicon.com.

SCHEMATIC CONSIDERATIONS

ETHERNET LINE SIDE

The Center Tap (CT1, CT2) and Signal Path (SP1, SP2) signals should be directly connected to the center taps on the RJ45 side of the Ethernet line transformer. IEEE 802.3af-compliant system will use either Alternative A or Alternative B power connectivity, with power supplied on CT1 & CT2, or SP1 & SP2 respectively. Although the AS1113 and AS1124 don't differentiate between the pairs, for design flexibility, we recommend that the designer use the IEEE connectivity.

Since the center-tap transformer connections do not carry data traffic, these traces can be wider on the board and increased width is recommended to carry the relatively large current (up to 1A).

DC-DC CONVERTER

A π filter should be employed on the VDD480 pin to reduce the amount of noise on the line. The following filter is recommended between the PD VDD480 pin and transformer primary winding node.

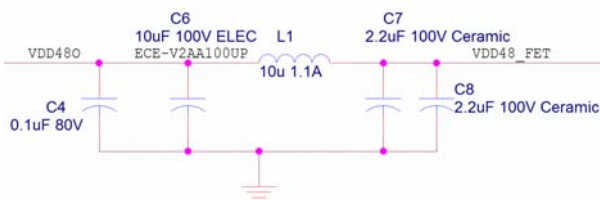


Figure 1: π Filter

Also any device connected to VDD480 should be rated to handle 80V and at least 2A.

The AS1113 and AS1124 support isolated forward, flyback and non-isolated buck DC-DC topologies. Information on the different topologies can be found in the corresponding datasheets.

The designer should also decide whether or not the system needs power isolation. If the system is non-isolated, the high voltage and low voltage grounds should be shorted together.

If the system is isolated, an opto-coupler is needed on the feedback line. This ensures that no DC path connects the two sides of the board. To reduce common-mode noise on the Ethernet lines, a 4700pF, 2KV cap is recommended between the PoE ground and board ground.

802.3af/at COMPLIANCE

Two pins, RCURR and RCLASS are used for programming the PD compliance levels in the AS1113 and AS1124.

AS1124 uses RCURR pin to set the on-board current limit. With the pin open, the nominal AS1124 current limit will be 800mA. With the pin shorted to ground, the nominal current limit will be 400mA. Mapping for intermediate values of RCURR can be found in the datasheet. In AS1113, this pin is connected to ground and can not be used for current limit programming.

User-programmable classification current levels are set by RCLASS pin. Each class represents a power allocation level for a PD, so that PSE can manage power between multiple PDs. The recommended resistance values for the RCLASS pins can be found in Table 1. More information on their use can be found in the datasheet.

For higher power, pre-IEEE802.3at applications, use of Class 4 is recommended. Updated information on the 802.3at standard can be found in the AS1124 datasheet.

Table 1. Classification Resistors

Class	Power (Watts)	Iclass	Rclass
0	0.44-12.95	0-4 mA	Pull-up
1	0.44-3.84	9-12 mA	280k Ω
2	3.84-6.49	17-20 mA	143k Ω
3	6.49-12.95	26-30 mA	90.9k Ω
4	Reserved	36-44 mA	63.4k Ω

COMPONENT PLACEMENT CONSIDERATIONS

Appendix A shows example evaluation board schematics for a Power Splitter design with Isolated Forward DC-DC Converter. Example layouts from this board are used to illustrate desirable design and layout practices through rest of this document.

ETHERNET SIGNALS

No special placement consideration is required for the Ethernet line components using the AS1113 or AS1124. In general the PD device should be placed as close to the center tap of the transformer as possible to limit IR drop.

DC-DC CONVERTER

If using a Forward or Flyback Topology, the NDRV, PDRV FETs, and their associated protection diodes should be placed as close to the PD Controller as possible. If possible, place them on the reverse side of the board. The DC-DC transformer should be placed on the top of the board to limit the distance between the switching FETs and the transformer.

When using in a buck topology, the FET and the inductor should be placed as close to the PD Controller as possible.

Figures 2 and 3 shows an example component placement of the top and bottom of a board.

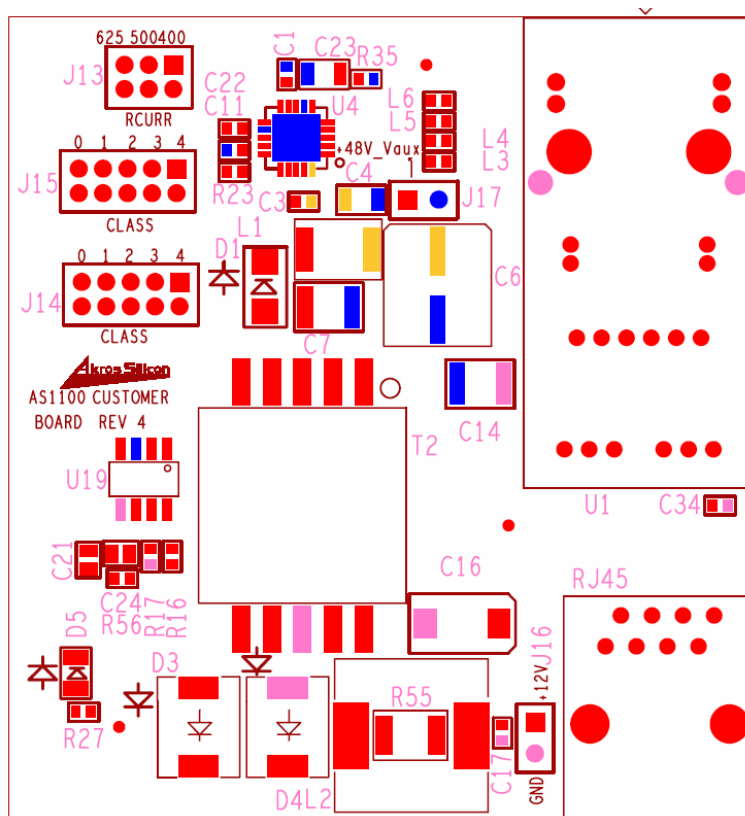


Figure 2: Top Side Component Placement

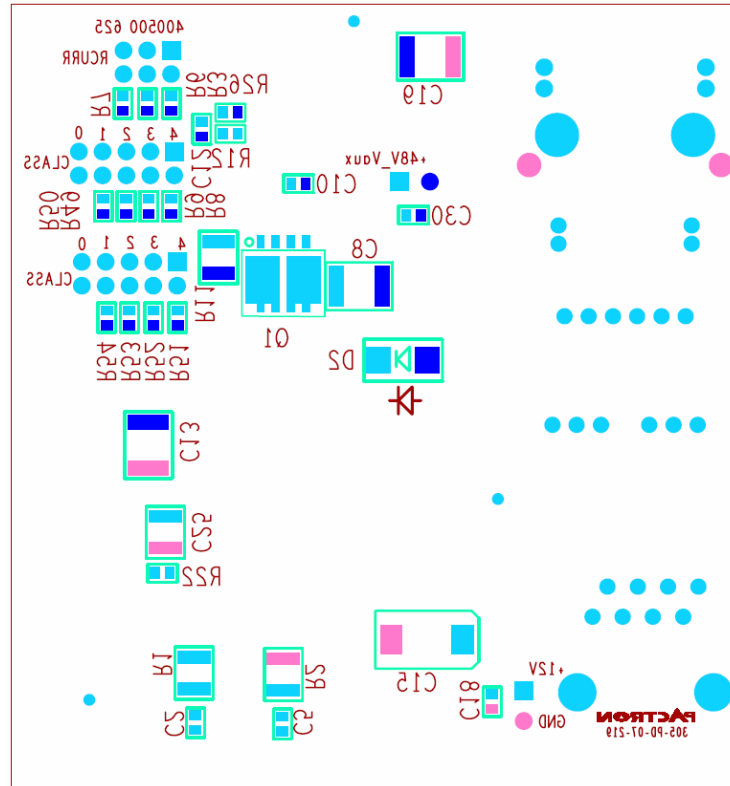


Figure 3: Bottom Side Component Placement

LAYOUT CONSIDERATIONS

ETHERNET SIGNALS

Ethernet signal layout on the secondary side of the input transformer should follow recommendation from the PHY supplier. The AS1113 and AS1124 do not require additional considerations.

In general, Ethernet data signals are high frequency differential signaling channels. Their routing should reflect 100Ω differential impedance on both sides of the input transformer. They should avoid sharp bends and sudden width changes as this affects the impedance and current carrying capacity of the lines. They should have appropriate ground planes underneath them for both common-mode signal return and to shield against coupled noise from the DCDC converter.

The grounds should be solid without islands or choke points to ensure low impedance return path.

High speed signals should be routed away from the DC-DC converter to avoid the converter switching noise appearing on

the data lines.

In general the center tap and transformer input signals (CT1, CT2, SP1 and SP2) should be 50mils wide to minimize the IR drop for the relatively large current to & from the line.

DC-DC CONVERTER

VDD480 signal routing should be at least 200 mils wide and all components directly connected to it should also be 200 mils wide including routing to the drain and gates of the FETs between NDRV, PDRV and the primary side of the transformer.

Traces attached to the secondary side of the converter should be appropriate for the output current of the DCDC converter. For max power delivery, it is recommended that a line-width of >200mils be used. Any layer transitions for these high-power signals require multiple vias to ensure low inductance and resistance.

The length of all high impedance small signal traces should be kept to a minimum and shielded from any switching power traces.

Ceramic bypass capacitors C3 and C10 should be placed as close as possible to the PD Controller for optimum sourcing of the switching current associated with the gate drive of the high side and low side power MOSFETs. This will assist in minimizing the high frequency ringing and ripple at VBP and VBN of the AS1113 and AS1124.

The AS11xx paddle should tie directly to the return of C7 and C8 in a path that does not share the high power current associated with the DC-DC converter.

Minimizing the loop area for all high frequency switching currents helps to reduce radiated EMI. Ceramic capacitors C7 and C8 supply the high frequency switching current for the primary side of the integrated converter block in the PD Controller. The loop of C7 and C8 connects to Q1-S_P and the cathode of D1 with the return path tying to the current sense resistor R11. This loop area should be minimized.

THERMAL PAD

The Akros-recommended 5x5mm 20-LD QFN footprint for the AS1100 package should be followed. It is essential that the bottom-side pad be used for both thermal and electrical contact. A good solder connection to it is essential.

This pad should be filled with as many vias as practical to ensure good thermal conductivity to the board ground plane. It is also recommended that a metal pad is created on the bottom-side of the board, and it is not recommended that signals be

routed underneath the part. Stack-up and General layout guidelines

For a multilayer board, ground and signal layers should be internal to the board and should separate active signals to ensure good capacitance between layers and ensure there is a good reference or return for the active signals.

GROUND LAYERS

The DC-DC converter creates isolated voltages from the high voltage signals coming from the Ethernet line. Because of this there are two grounds in the system: a high voltage and a low voltage ground. The grounds should be separated by a gap on all power and ground planes to reduce DC-DC noise coupling to the Ethernet signals. Akros recommends a minimum gap of 0.050". Ground planes should use 1oz copper and should be as continuous as possible with no isolated islands and no narrow choke points.

To maintain ground separation, secondary-side components and routing should be placed over the low-voltage ground and primary-side components and routing should be placed over the high-voltage ground.

LAYOUT EXAMPLE

Figure 4 is an example layout that applies to both the AS1113 and AS1124.

Appendix A has the corresponding schematics the implementation of a forward isolated converter.

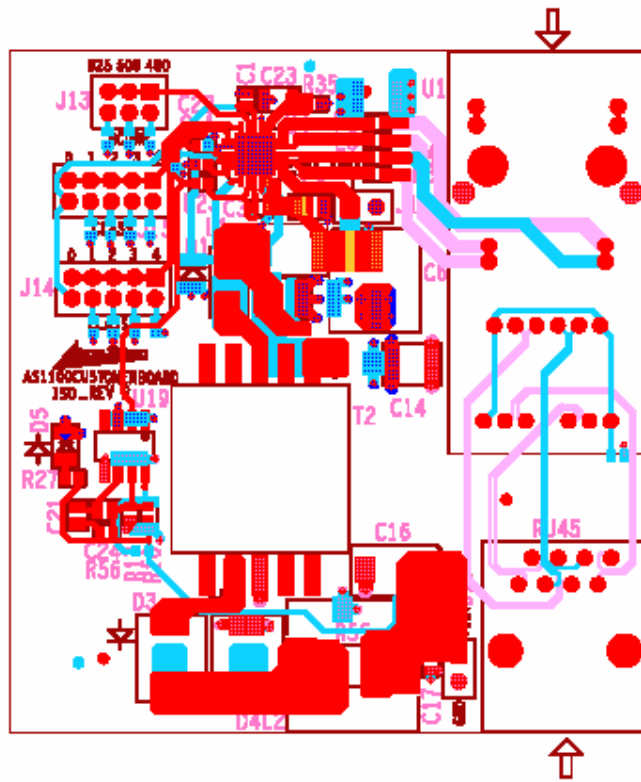


Figure 4: Layout Example

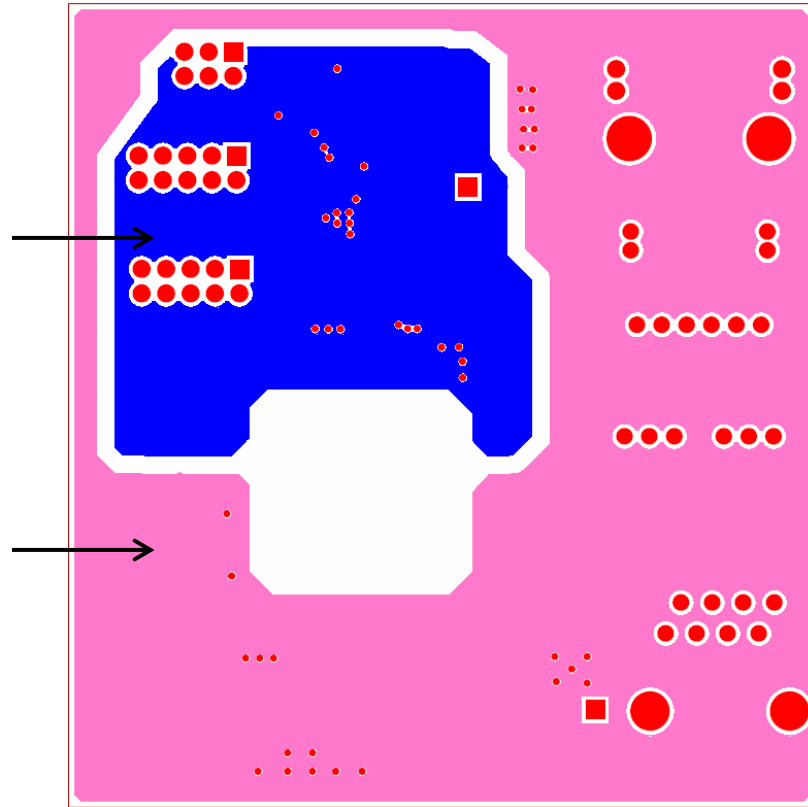
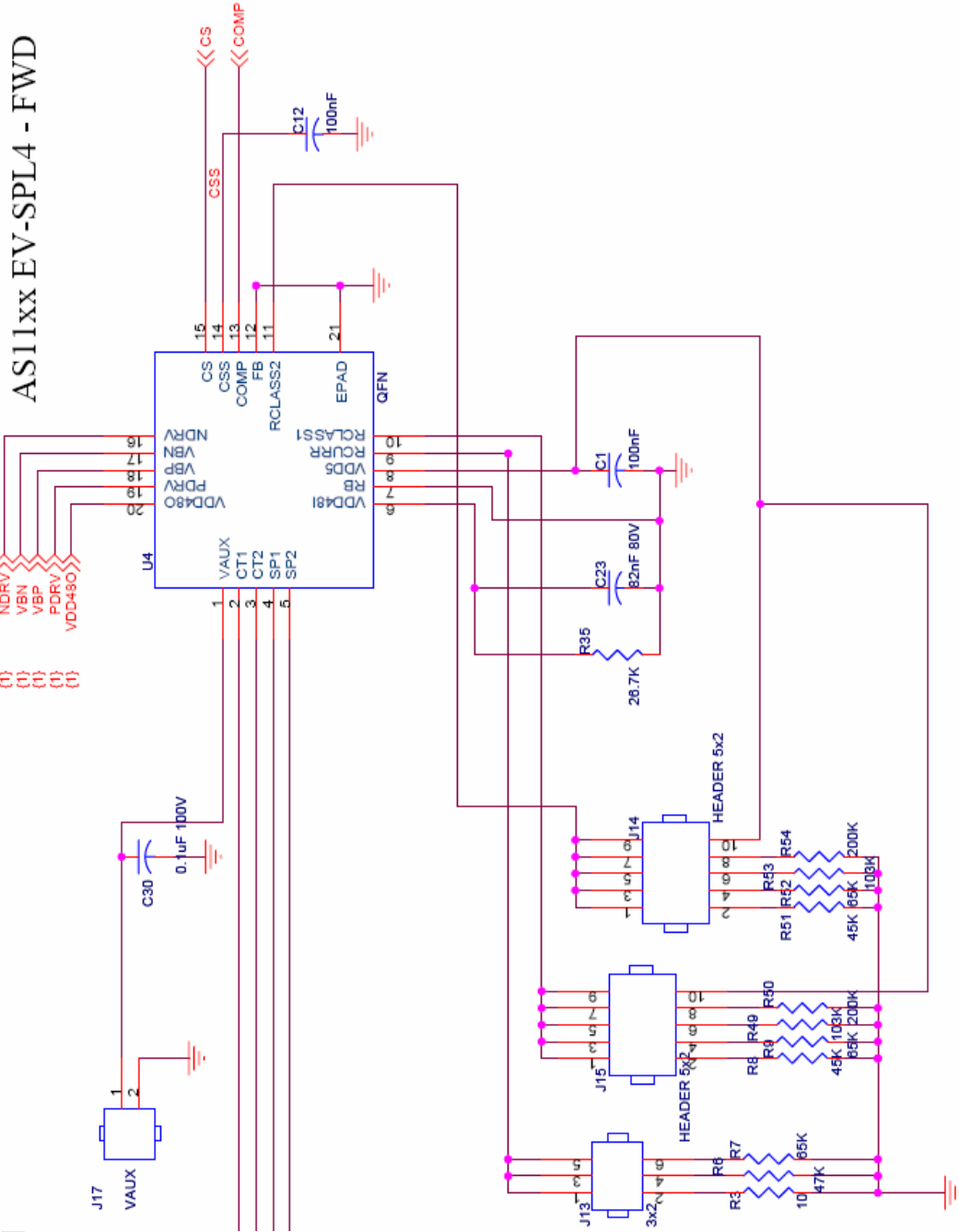


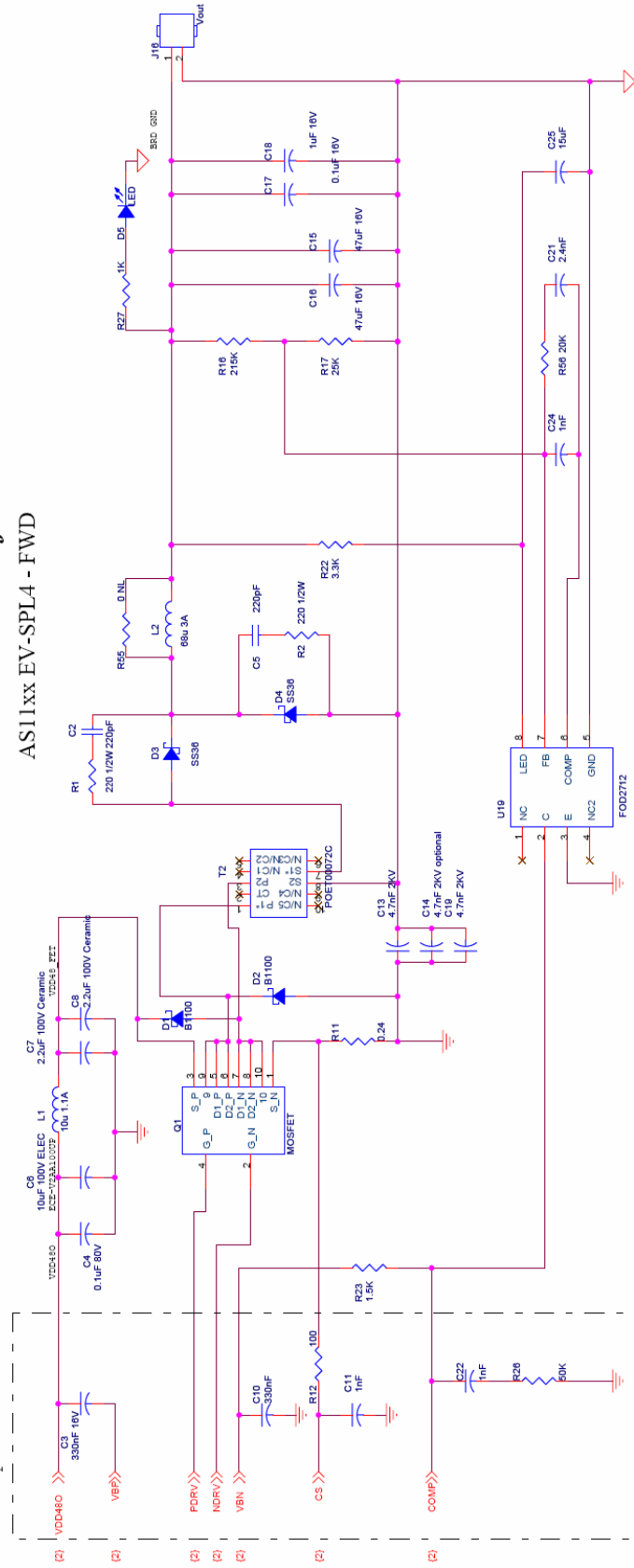
Figure 5: Recommended Ground Plane Split on the board using Mag-jack for POE port

APPENDIX A: EXAMPLE SCHEMATIC

Akros Silicon - Confidential



Layout Note: Place these components closer to AS1100 device pins.



Akros Silicon - Confidential
AS11xx EV-SPL4 - FWD

CONTACT INFORMATION

Akros Silicon Inc.

6399 San Ignacio
Ave, San Jose , CA
95119 USA

Tel: (408) 746 9000 ext. 100

Fax: (916) 351 8102

Email inquiries: marcom@akrossilicon.com

Website: www.akrossilicon.com

IMPORTANT NOTICES

LEGAL NOTICE

Copyright © 2006 Akros Silicon™ . All rights reserved.

Other names, brands, and trademarks are the property of others.

Akros Silicon™ assumes no responsibility or liability for information contained in this document. Akros reserves the right to make corrections, modifications, enhancements, improvements, and other changes to its products and services at any time and to discontinue any product or services without notice. The information contained herein is believed to be accurate and reliable at the time of printing.

LIFE AND SAFETY POLICY

Akros' products are not authorized for use as critical components in life support systems for surgical implant into the body, or other applications intended to support or sustain life or any other applications whereby a failure of the Akros product could create a situation where personal injury, death or damage to persons, systems, data or business may occur.