Application Report
SNLA107A–June 2008–Revised April 2013

AN-1862 Reducing Radiated Emissions in Ethernet 10/100 LAN Applications

ABSTRACT

This application report explains how Texas Instruments' PHYTER products help system designers to reduce radiated emissions in Ethernet 10/100 LAN applications.

Contents

List of Figures

PHYTER is a registered trademark of Texas Instruments.
All other trademarks are the property of their respective owners.

1 Introduction

Ethernet network equipment is required to meet US and International radiated Electromagnetic Interface (EMI) compliance standards, including the US FCC part 15 and IEC/CENELEC/CISPR—22/EN55022 standards. Texas Instruments' 10/100 Ethernet PHYTER® products are designed to help end user applications meet these standards. A number of key factors influence radiated emissions compliance testing performance in networked equipment.

This application note is applicable to the following products:

DP83640	DP83849C	DP83848C
DP83630	DP83849I	DP83848I
DP83620	DP83849ID	DP83848YB
	DP83849IF	DP83848VYB
		DP83848M
		DP83848T
		DP83848H
		DP83848J
		DP83848K
		DP83848Q-Q1

2 Key Recommendations

PHYTER products are designed to limit EMI in system implementations in three ways.

1. Analog and digital power distribution systems are intentionally partitioned, both externally and within the component, to reduce cross functional noise that can result in EMI.
2. Key analog blocks within the component are designed and tested to meet specific ground and power supply noise rejection targets, further reducing the effects of cross functional noise.
3. In some PHYTER products, including DP83849 and DP83640 products, integrated series terminations are provided on digital signal drivers, reducing I/O related EMI.

In addition to these designed in advantages, key recommendations for designing reduced EMI applications include:

- Use high quality CAT5E or better cable in implementing network systems. If possible use shielded cable.
- Use shielded network connectors connected to a decoupled chassis ground plane.
- Use equal length differential MDI (Medium Dependent Interface) signal traces with a strip line impedance of 50 ohms.
- Carefully match the values and placement of MDI termination components.
- Use a common mode choke component in conjunction with the isolation transformer.
- Place local bypass components (including capacitors and optionally ferromagnetic beads) between device supply pins and power sourcing vias on PCB's.
- Reduce the energy of digital signal sources by including series termination resistors in signal paths.

A detailed summary of these and other recommendations is included in the text of this application note.

For demonstration purposes, many of these recommendations were implemented in a test system utilizing a DP83640 family device. Testing was performed using both unshielded and shielded cable. Results from these tests demonstrated FCC and EN55022 class B emissions standards compliance. The full report is available in the DP83640 product page at www.ti.com.

3 Background: Ethernet Signaling Basics

The most important principle to understand when considering Electromagnetic Compatibility (EMC), is that for any signal transmitted from a source to a destination, an equal amount of signal energy returns to the signal source.

With regard to Ethernet physical layer devices, two types of signals are used: single ended and differential signals. Single ended signals utilize single wires or PCB traces as a transmission path for source energy and usually a PCB ground plane or a cable shield for return energy. Differential signals utilize dedicated paths for forward and return energy.

Ethernet physical layer devices provide both a Medium Dependent Interface (MDI), which consists of differential data signals and a Medium Independent Interface (MII), which consists of single ended clock and data signals. In addition to the MDI and MII signals, physical layer devices also utilize internal and external clock signals, and power supply and ground signals. All of these signals are important in considering EMC.

3.1 Differential Signaling

On the MDI side, differential signal energy is usually transferred across networks using twisted pair cable. If the forward and return signals are well balanced and placed in close proximity to each other, the energy fields generated by the signals cancel each other and the signals do not radiate.

However, if non-differential energy (common mode noise) is present on the cable, the most convenient return path for the common mode noise becomes earth ground. The combination of forward common mode energy on the cable coupled with return energy on earth ground results in radiated energy, or EMI.

Figure 1 illustrates the operation of a single differential pair between two network devices. Both differential signal energy and undesirable common mode noise are illustrated.

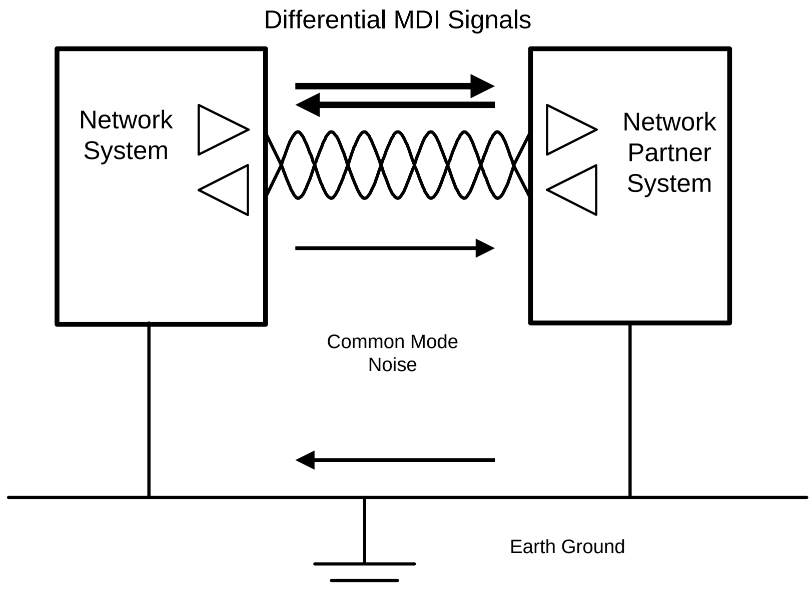

Figure 1. MDI Signal Energy between Networked Systems

3.2 Single Ended Signaling

As mentioned earlier, both clock and MII signals are non–differential, or single ended in nature. Power supply current can be considered a single ended signal as well. As with any single ended signal, energy that is transferred through a power distribution system returns to the power supply source through the ground system.

Single ended signaling is one potential source for unwanted common mode energy. Single ended signals can couple onto a system chassis or network cable through the PCB traces or the power/ground system, producing unwanted EMI. Similarly, power supply current is a potential source for unwanted energy. Power supply and ground currents can couple onto system chassis and network cables as well.

Figure 2 illustrates the operation of a single receive data signal between a physical layer device and a digital system. Both signal energy and power supply energy are illustrated.

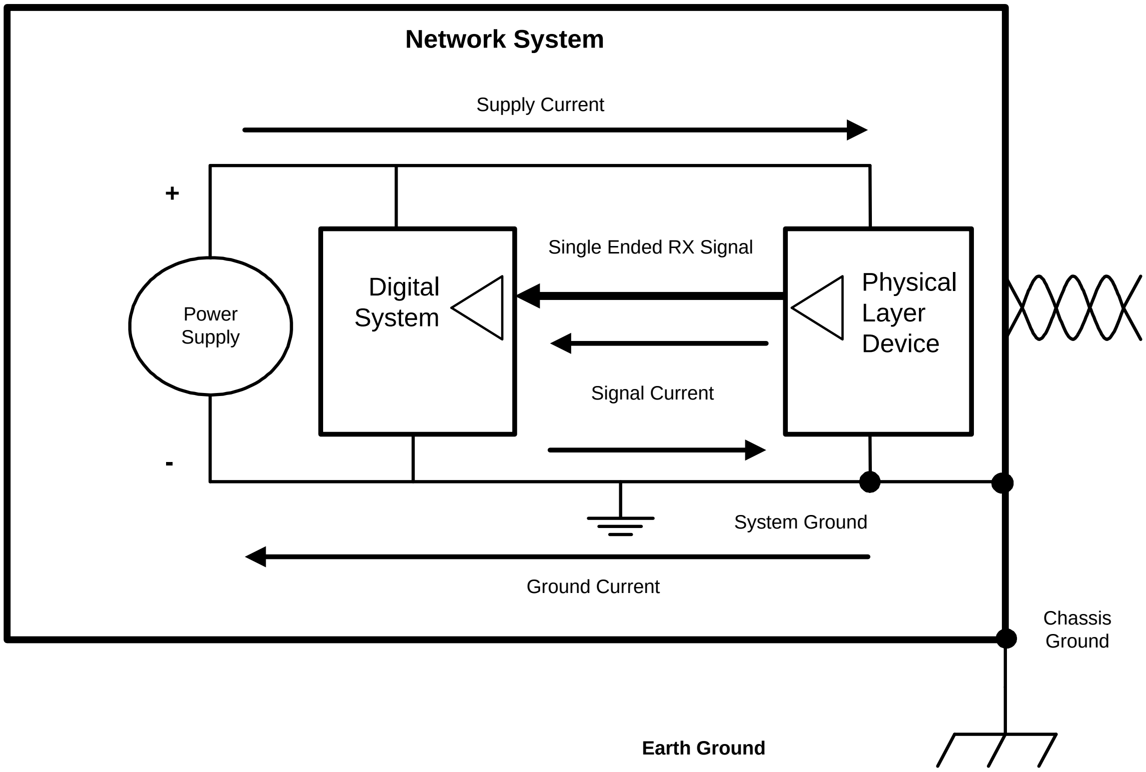

Figure 2. MII Signal Energy

4 Sources of EMI in Ethernet Applications

Ideally, if differential signals are perfectly balanced, no common mode energy exists in the system. In single ended signal systems, ideally all forward energy is contained within the signal wire or trace and return energy is contained within a ground wire or plane in close proximity to the signal.

The source of unwanted emissions in network applications is common mode energy, radiating either from differential signal wires, or directly from the system chassis.

This common mode energy can originate from any of three sources:

1. Imbalance in the differential signal path
2. Noise coupled to or from the system chassis or power supply system
3. Noise coupled to or from the network interface cable

4.1 Differential Signal Path Imbalance

Signal path imbalance can occur in two ways: across the differential signal pair or between the signal source and destination.

Imbalance that occurs across a twisted pair can result from the cable medium itself being unbalanced, or from signal termination imbalance. Imbalance across a signal pair results in one signal having a larger magnitude than the opposite signal, which manifests itself as common mode noise.

End to end or longitudinal path imbalance can occur if the source impedance, transmission line impedance, and destination impedance are not exactly matched in a system. This form of mismatch causes energy reflections across the cable from end to end, which also results in common mode noise.

Recommendations for preventing differential signal path imbalance include:

- Use high quality symmetrically and tightly wound cable. ISO CAT5E or better quality cable is recommended for 10/100 applications.
- Use equal length differential MDI signal traces with a strip line impedance of 50 ohms.
- Closely match the values and physical placement of signal termination components.

4.2 Chassis Coupled Noise

The network cable connector in a system can be a source for radiated noise. At this critical point, any noise that originates from inside the system can couple through the connector to the chassis and to the cable.

Ground loops form when a low impedance path is made available across a chassis for power supply and common mode signal return energy. Energy on the chassis can be a source for common mode radiation, as can chassis energy coupled onto the network cable.

Recommendations for limiting chassis coupled noise include:

- Use a shielded connector on the network interface PCB. The shielded connector should be connected to a PCB chassis ground plane that is decoupled from the PCB system ground.
- Connect the chassis ground plane to the system ground plane using size 1206 zero ohm resistors symmetrically placed on either side of the RJ45 connector. These resistors can be removed or replaced with alternative components (i.e. capacitors or EMI beads) if necessary during certification testing. See Figure 3.
- Use common mode choke transmission components in the network interface PCB design. These devices are commonly available in discrete form, integrated into network transformers, and integrated with transformers in network connectors.
- If possible, the use of shielded cable can reduce emission levels by 6 to 10 dBµV or more.

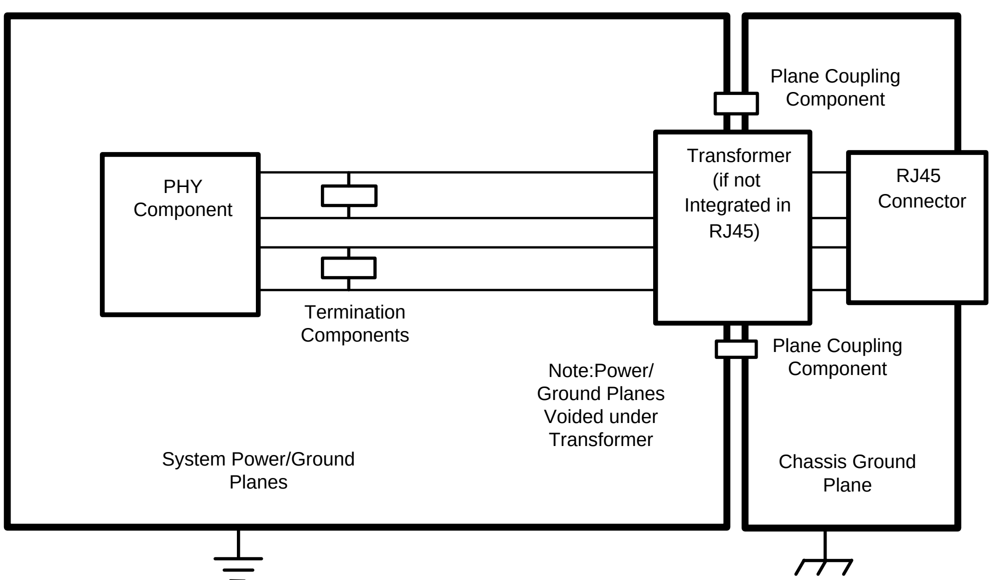

Figure 3. Chassis and Ground Plane Layout

4.3 PCB Coupled Noise

PCB coupled noise originates from signals within a chassis, including the network interface PCB. This noise can couple to the differential signal transmission signals, and then propagate to the outside wire.

There are four ways PCB noise can couple to the network signal transmission path on the PCB:

1. Inductive or magnetic coupling
2. Direct capacitive signal coupling
3. Power/ground plane noise coupling
4. Digital I/O noise coupling

4.3.1 Magnetic Coupling

Magnetic devices, including transformers and common mode chokes, are used for isolation in network applications. These devices can be vulnerable to inductive or magnetic coupling of noise that resides on the system PCB ground plane. Recommendations for limiting magnetically coupled noise include:

- Void power and ground planes under discrete magnetic components.
- Provide a chassis ground plane under network connectors utilizing integrated magnetic components.

4.3.2 Direct Capacitive Signal Coupling

Direct capacitive signal coupling can occur if single ended signals are run in parallel or in near proximity to network signal traces. The recommendation for preventing direct signal noise coupling is:

- Keep differential MDI network signals physically separated from single ended digital signals.

4.3.3 Power/Ground Plane Coupling

With regard to power/ground plane induced noise coupling, TI's PHYTER products are designed such that digital and analog supply sources are physically isolated within the component.

Component network driver and receiver circuits are also designed specifically to prevent the effects of on chip common mode and power supply noise from coupling onto network signals.

Even though isolation is provided within the physical layer component, care must also be taken to limit power and ground noise external to the component. This is accomplished through adequate capacitive bypassing of component power pins. The following recommendations are provided for reducing power and ground plane noise sources. (See Figure 4):

- Bypass components must be placed as physically close to the individual power supply pins of the component as possible, preferably between the component and the via connecting the trace to the power plane. Low impedance ceramic 0.1 µF bypass capacitors are recommended for all PHYTER family products.
- Ground vias must be placed as close to the ground pins as possible.
- In systems where noise is especially prevalent, the use of ferromagnetic bead components in series with device supply pins may be desirable. When using ferromagnetic beads, usually an impedance of between 100 and 2000 ohms at 100 MHz is adequate.
- It is also important that the physical size of bead components be chosen to accommodate the current necessary to supply the physical layer device supply pins. See individual component data sheets for component current requirements.

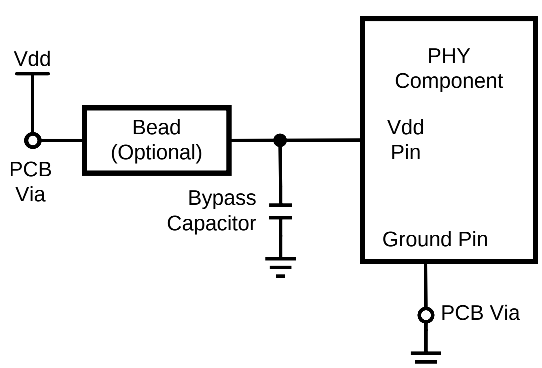

Figure 4. Bypassing and Isolating Power and Ground Pins

4.3.4 Digital I/O Noise Coupling

With regard to noise originating from digital I/Os, signals that have fast edge rates or high frequency content, and signals with long trace lengths can be sources of EMI. Recommendations for reducing the effects of digital I/O noise coupling include:

- Keep digital single ended signals as short as possible.
- In the event that long (greater than 10 cm) trace lengths are necessary, noise can be reduced (and signal integrity improved) by utilizing balanced termination of the signal. This is accomplished by providing equal value termination resistors in series at the source of the signal, and in parallel (to ground) at the destination of the signal. Additionally, the signal trace impedance should be matched to the termination resistor value.
- Use series termination resistors at output drivers to reduce the amount of energy delivered to a signal, and also to reduce the supply current demand for I/O transitions. A value of 50 ohms is recommended for series termination of MII signals. Some PHYTER products, including the DP83849 and DP83640 families of products have integrated output terminations that alleviate the need for on board signal termination resistors.

5 System Test Data

As mentioned in the introduction of this document, a test system utilizing a DP83640 family device was developed for demonstration purposes. FCC and EN55022 EMC certification data was recorded at AHD LC, a National Voluntary Lab Accreditation Program (NVLAP) certified EMI test site lab (www.ahde.com).

The test system consisted of an aluminum enclosure that housed a DP83640 family physical layer component PCB, and a Programmable Logic Device (PLD) based packet generator PCB. Data was recorded using both shielded and unshielded CAT5 cables. 100MB per second data was generated in standard MII mode by the PLD, and looped back through the PHY device and the cable back to the PLD, fully exercising the PHY component MDI and MII transmit and receive signals. See Figure 5.

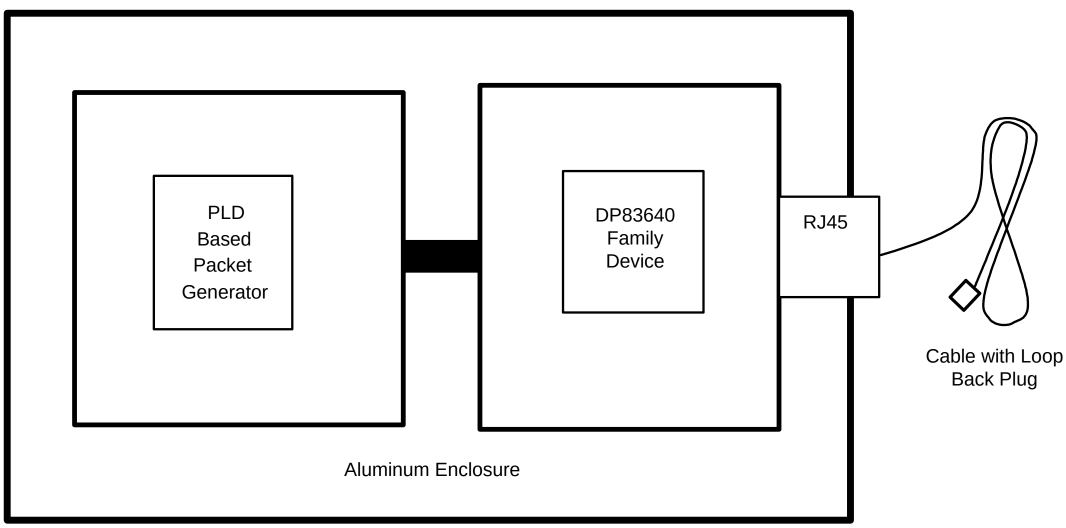

Figure 5. Test System Configuration

Testing was performed using a broadband antenna located 3 meters from the device under test and calibrated to FCC and IEC standards. The test results revealed FCC / EN55022 class B compliance for both unshielded and shielded cables.

The full report is available in the DP83640 product folder at www.ti.com.

6 Summary

This application note described key factors which influence radiated EMI performance in Ethernet network systems. Basic Ethernet signaling was described, including differential and single ended signaling, and the potential issues associated with each signal type. Specific recommendations were also provided for designing Ethernet network interface systems to meet US and International radiated EMI standards.

For convenience, all recommendations described in the text are included here in table form:

Noise Source	Recommendation
Differential Signal Path	Use high quality symmetrically and tightly wound cable. ISO CAT5E or better quality cable is recommended for 10/100 applications.
	Use equal length differential MDI (Medium Dependent Interface) signal traces with a strip line impedance of 50 ohms.
	Closely match the values and physical placement of signal termination components.
Chassis Coupled Noise	Use a shielded connector on the network interface PCB. The shielded connector should be connected to a PCB chassis ground plane that is decoupled from the PCB system ground.
	Connect the chassis ground plane to the system ground plane using size 1206 zero ohm resistors symmetrically placed on either side of the RJ45 connector. These resistors can be removed or replaced with alternative components (i.e. capacitors or EMI beads) if necessary during certification testing. See Figure 3.
	Use common mode choke transmission components in the network interface PCB design. These devices are commonly available in discrete form, integrated into network transformers, and integrated with transformers in network connectors.
	If possible, the use of shielded cable can reduce emission levels by 6 to 10 dB or more.
PCB Coupled Noise	Void power and ground planes under discrete magnetic components.
	Provide a chassis ground plane under network connectors utilizing integrated magnetic components.
	Keep differential MDI network signals physically separated from single ended digital signals.
	Bypass components must be placed as physically close to the individual power supply pins of the component as possible, preferably between the component and the via connecting the trace to the power plane. Low impedance ceramic 0.1 µF bypass capacitors are recommended for all PHYTER family products.
	Ground vias must be placed as close to the ground pins as possible.
	On occasion, where system noise is especially prevalent, the use of ferromagnetic bead components in series with device supply pins may be desirable. When using ferromagnetic beads, usually an impedance of between 100 and 2000 ohms at 100 MHz is adequate.
	It is also important that the physical size of bead components be chosen to accommodate the current necessary to supply the physical layer device supply pins. See individual component datasheets for component current requirements.
Digital I/O Noise Coupling	Keep digital single ended signals as short as possible.
	In the event that long (greater than 10 cm) trace lengths are necessary, noise can be reduced (and signal integrity improved) by utilizing balanced termination of the signal. This is accomplished by providing equal value termination resistors in series at the source of the signal, and in parallel (to ground) at the destination of the signal. Additionally, the signal trace impedance should be matched to the termination resistor value.
	Use 50 ohm series termination resistors at output drivers to reduce the amount of energy delivered to a signal, and also to reduce the supply current demand for I/O transitions. Some PHYTER products, including the DP83849 and DP83640 families of products have integrated output terminations that alleviate the need for on board signal termination resistors.

7 References

For further information, the following references are provided:

- *DP83848C PHYTER Comm Temp Single Port 10/100Mb/s Ethernet Phy Layer Transceiver* (SNOSAT2)
- *DP83848I Ind Temp Single Port 10/100 Mb/s Ethernet Phy Layer Transceiver* (SNLS207)
- *DP83848YB Extreme Temp Single Port 10/100 Mb/s Ethernet Phy Layer Transceiver* (SNLS208)
- *DP83848M PHYTER Mini - Commercial Temperature Single 10/100 Ethernet Transciver* (SNLS227)
- *DP83848T PHYTER Mini - Industrial Temp Single 10/100 Ethernet Transceiver* (SNLS228)
- *DP83848VYB PHYTER - Extended Temperature Single Port 10/100 Mb/s Ethernet Physical Layer X-ceiver* (SNLS266)
- *DP83848J PHYTER Mini LS Commercial Temperature Single Port 10/100 Mb/s Ethernet Transceiver* (SNLS250)
- *DP83848K PHYTER Mini LS Industrial Temperature Single Port 10/100 Ethernet Transceiver* (SNLS251)
- *DP83848C/I/YB Schematic* (SNLR019)
- *DP83848C/I/YB Bill of Materials* (SNLR020)
- *DP83848M/T/H Schematic* (SNLR015)
- *DP83848M/T/H Bill of Materials* (SNLR016)
- *DP83848J/K Schematic* (SNLR011)
- *DP83848J/K Bill of Materials* (SNLR012)
- *DP83849C PHYTER DUAL Commercial Temperature Dual Port 10/100 Mb/s Ethernet Physical Layer* (SNOSAX0)
- *DP83849I PHYTER DUAL Industrial Temperature with Flexible Port Switching Dual Port* (SNOSAX1)
- *DP83849ID PHYTER DUAL Industrial Temperature with Fiber Support (FX), Dual Port 10/100 Mb/s Ethernet PHY X-ceiver* (SNOSAX2)
- *DP83849IF PHYTER® DUAL Industrial Temperature with Fiber Support (FX) and FI* (SNOSAX8)
- *DP83640 Precision PHYTER - IEEE® 1588 Precision Time Protocol Transceiver* (SNOSAY8)
- *AN-1469: PHYTER Design & Layout Guide* (SNLA079)
- Montrose, Mark I. and Nakauchi, Edward W. 2004. Testing for EMC Compliance. Piscataway, NJ: IEEE
- FCC Part 15 Regulations, available at www.fcc.gov/oet/info/rules/
- IEC EN61000-6-3 2007. Emissions standards for residential, commercial, and light-industrial environments
- IEC EN61000-6-4 2007. Emission standard for industrial environments
- CISPR-22 Limits and Methods of Measurements of Radio Interference Characteristics of Information Technology Equipment.
- EN 55022: 2002. Limits and Methods of Measurements of Radio Interference Characteristics of Information Technology Equipment.
- www.ahde.com.

Products		**Applications**	
Audio	www.ti.com/audio	Automotive and Transportation	www.ti.com/automotive
Amplifiers	amplifier.ti.com	Communications and Telecom	www.ti.com/communications
Data Converters	dataconverter.ti.com	Computers and Peripherals	www.ti.com/computers
DLP® Products	www.dlp.com	Consumer Electronics	www.ti.com/consumer-apps
DSP	dsp.ti.com	Energy and Lighting	www.ti.com/energy
Clocks and Timers	www.ti.com/clocks	Industrial	www.ti.com/industrial
Interface	interface.ti.com	Medical	www.ti.com/medical
Logic	logic.ti.com	Security	www.ti.com/security
Power Mgmt	power.ti.com	Space, Avionics and Defense	www.ti.com/space-avionics-defense
Microcontrollers	microcontroller.ti.com	Video and Imaging	www.ti.com/video
RFID	www.ti-rfid.com		
OMAP Applications Processors	www.ti.com/omap	**TI E2E Community**	e2e.ti.com
Wireless Connectivity	www.ti.com/wirelessconnectivity		

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265