LXT971A/972A 3.3V PHY Transceivers Design and Layout Guide

Application Note

November 2001

Order Number: 249016-003

Contents

Figures

Tables

Revision History

Date	Revision	Page	Description
November 1, 2001	003	7	Added last bullet.
		23	Added Appendix A: LXT971A Design and Layout Checklist
January 2001	002		Added new language for system clock requirements.
			Change “6 mm” to “6 mils” and “8 mm” to “8 mils”.
			Add crystal/crystal oscillator table.

1.0 General Description

This application note provides detailed design and layout guidelines for achieving optimum performance using Intel's LXT971A or LXT972A 3.3V Dual-Speed PHY Transceiver. Adherence to these guidelines helps ensure a successful design that meets IEEE requirements.

Note: This application note uses the singular designation "LXT971A" to refer to both the LXT971A and LXT972A devices, unless otherwise specified.

This document also supports the LXT971 and LXT972 devices.

The following topics are discussed in this document:

Design Guidelines: Good design practices prevent most common signal and noise issues. General guidelines listed in this section should be followed throughout the entire design.

Power and Ground: This section covers layout of the power and ground planes and internal routing of power and ground signals. Also included are some tips to avoid creating loop antenna effect.

MII Interface: This section discusses the Media Independent Interface (MII).

Network Interfaces: This section provides termination circuitry for the twisted-pair interface. Ideal biasing networks that attach to an external fiber optic transceiver are also shown for the fiber interface.

Magnetic Requirements: This section details the magnetic specifications. Before committing to a specific component, designers should contact the manufacturer for current product specifications and validate components for each application.

1.1 Features

The LXT971A is a 3.3V single-port PHY transceiver supporting both 100BASE-TX and 10BASE-T applications. The LXT971A also supports 100BASE-FX operation via a Pseudo-ECL (PECL) interface (LXT971A only).

The LXT971A incorporates Intel's Optimal Signal Processing (OSP) architecture for low-power consumption and requires only a single 3.3V power supply.

Other features of the LXT971A include:

- Low-power "Sleep" mode (LXT971A only)
- Support for auto-negotiation and parallel detection
- MII interface with extended register capability
- Robust baseline wander correction performance
- 100BASE-FX fiber-optic capable
- Standard CSMA/CD or full-duplex operation
- Configurable via MDIO serial port or hardware control pins
- Integrated programmable LED drivers
- Integrated transmitter termination resistors

2.0 General Design Guidelines

2.1 Introduction

Meeting EMI and ESD requirements and achieving maximum line performance depends on good design practices. These practices minimize high-speed digital switching noise, common-mode noise, and provide shielding between internal circuits and the environment. Good design practices apply throughout the entire design, not just to the LXT971A device, and include the following:

2.2 General Recommendations

- Verify all components meet application requirements. Use component listings only for reference.
- Design in filters for the analog power circuits. The filters may be removed if performance testing proves they are unnecessary.
- Follow the guidelines for designing and laying out the twisted-pair and/or fiber interfaces, including standard practices for differential signals and guidelines for optimizing return loss performance.
- Provide termination on all high-speed switching signals and clock lines.
- Provide impedance matching on long traces to prevent reflections.
- Attach RBIAS to a 22.1 k Ω 1% resistor to ground for internal reference current setup. Place the resistor close to the LXT971A.

2.3 Power and Ground Filtering

- Follow good design practices to minimize noise from digital switching and power supply circuits.
- Ensure the power supply is rated for the load.
- Keep power and ground noise levels below 50 mV.
- Filter the analog power circuits. The filters may be removed if performance testing proves they are unnecessary.
- Filter and shield DC-DC converters, oscillators, etc.

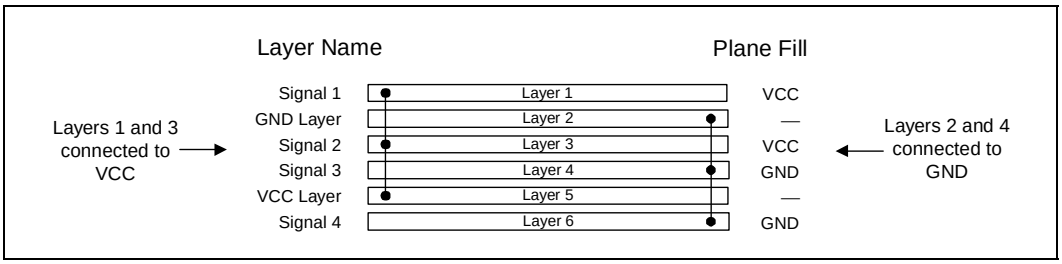
2.4 Decoupling and Bulk Caps

- Use bulk capacitors (4.7 - 10 µF) between the power and ground planes to minimize power-supply switching noise.
- Use an ample supply of .01 µF decoupling capacitors to reduce high-frequency noise on the power and ground planes.

2.5 Power and Ground Planes

- Provide ample power and ground planes.
- Avoid breaks in the ground plane, especially in areas where it is shielding high-frequency signals.
- Route high-speed signals above a continuous, unbroken ground plane.
- When possible, fill in unused areas of the signal planes with solid copper and attach them with vias to a VCC or ground plane that is not located adjacent to the signal layer. This technique is referred to as signal layer filling and can improve capacitive coupling of the power planes (refer to Figure 1).

Figure 1. Signal Layer Filling



2.6 Magnetic “Safe Zone”

- Void power and ground planes directly under the magnetics. Use chassis ground in the area from the magnetics to the RJ-45 connector.
- Keep high-speed signals out of the area between the LXT971A and the magnetics.
- Do not route any digital signals between the LXT971A and the RJ-45 connectors at the edge of the board.

2.7 Differential Signal Layout

- Route differential pairs close together and away from other signals.
- Keep both traces of each differential pair as identical to each other as possible.
- Keep each differential pair on the same plane.
- Minimize vias and layer changes.
- Keep transmit and receive pairs away from each other. Run orthogonally, or separate with a ground plane layer. One recommendation to maintain this separation is to place all components for the transmit circuit on one side of the board, and all components for the receive circuit on the other side of the board.

2.8 BGA Layout Considerations

Designing with a PBGA package requires special attention to spacing of pads and routing of signals. The LXD971 Demo Board is designed with careful consideration to trace widths and signal routing. The pinout for the LXT971A ensures that the MII signals can be routed on one side

of the chip and the twisted-pair or fiber signals can be routed on the other side of the chip without crossing traces. 6 mils wide traces are used between the pads of the chip for routing and 8 mils wide traces are used outside the boundaries of the chip for routing the signals. This ensures proper spacing for grouping and routing of all the signals to their respective sections.

2.9 Boundary Scan Interface

The LXT971A supports an IEEE 1149.1 Boundary Scan Test Interface for board-level testing. This interface consists of five pins (TMS, TDI, TDO, $\overline{\text{TRST}}$, and TCK). Boundary Scan pins have internal termination and may be left floating when not in use. The BSDL file is available by contacting your local sales office or by accessing the Intel website at www.intel.com.

2.10 System Clock Requirements

The LXT971A clock circuit requires a 25 MHz ±100 ppm reference clock (REFCLK) that must be enabled at all times. Characteristics of the LXT971A clock include:

- Duty cycle distortion no greater than 35 to 65%
- TTL voltage levels (V_{OH} > 2.0V)

The reference clock input is used to generate signals and recover receive signals. It may be provided by either of two methods: connecting a crystal across the oscillator pins (XI and XO), or connecting an external clock source to pin XI. The connection of a clock source to the XI pin requires the XO pin to be left open. A crystal-based clock is recommended over a derived clock (for example, PLL-based) to minimize transmit jitter.

Regardless of clock source, careful consideration should be given to physical placement, board layout, and signal routing of the source to maintain the highest level of signal integrity. See the “Clock Layout Guidelines” on page 10 for more details.

A crystal is typically used in NIC applications. An external 25 MHz clock source, rather than a crystal, is frequently used in switch applications. Table 1 lists the crystals and crystal oscillators recommended for use with the LXT971A and LXT972A.

Table 1. Crystals/Crystal Oscillators

Manufacturer	Part Number	Type
Epson (Surface Mount)	MA-505	Crystal
Caliber (Through Hole)	AA18C1	Crystal
JDR	OSC250	Crystal Oscillator
CTS	MX045	Crystal Oscillator

2.10.1 Clock Layout Guidelines

- Keep the clock traces as short as possible.
- Route the clock traces adjacent to an unbroken ground plane.
- Use a multi-output clock driver when driving multiple inputs with a single oscillator.
- Individually terminate point-to-point interconnects to every clock load. Series termination is the most common termination technique.

3.0 Power and Ground Design

3.1 Power and Ground Planes

For high-speed communications design, the power and ground planes may be conceptually divided into four regions (the analog and digital power planes and the chassis and signal ground planes) as shown in Figure 2 on page 12.

3.1.1 Power Planes

3.1.1.1 Analog VCC Plane

The analog power region extends from the magnetics back to the LXT971A. The power plane in this area should be filtered. Only components and signals pertaining to the analog interface should be placed or routed through this region. The analog plane supplies power to the VCCA pins of the LXT971A as shown in Figure 3 on page 12.

3.1.1.2 Digital VCC Plane

The digital power region extends from the MII interface of the LXT971A through the rest of the board. Good design practices listed in the previous section should be followed throughout this area. The digital plane supplies power to VCCD and VCCIO as shown in Figure 3 on page 12. External components (oscillators and the MAC) are also supplied from the digital plane.

3.1.2 Ground Planes

3.1.2.1 Signal Ground

The signal ground region should be one continuous, unbroken plane extending from the magnetics through the rest of the board.

Signal ground planes often have high-frequency noise caused by returning signal currents. While these high-frequency fluctuations are too small to cause issues in the digital circuits, they are large enough to exceed FCC limits and are often coupled onto signals running outside the digital block. Using chassis ground minimizes high-frequency noise in the logic ground plane.

3.1.2.2 Chassis Ground

A chassis ground plane can be added to the layer stack. Place this plane directly next to a signal ground plane to create a very tight capacitive coupling between the two planes. The chassis plane should then be multi-point connected to the external chassis.

Chassis ground can also be combined with the signal ground layer. For isolation, place a “moat” around the signal ground plane to separate signal ground from chassis ground.

The chassis ground region extends from the front edge of the board (RJ-45 connectors) to the magnetics, and around the entire perimeter of the board. No signals should pass through this region except for external interfaces and LED signals.

3.1.3 Avoiding Loop Antenna

When laying out ground planes, special care must be taken to avoid creating loop antenna effect.

- Run all ground planes as solid square or rectangular regions.
- Avoid creating loops with ground planes around other planes. The only exception to this rule is chassis ground as shown in Figure 2.
- Ensure the chassis ground loop (running the perimeter of the board) is voided at some point.
- Ensure the gap of the voided area in chassis ground is large enough to prevent a ground loop.

Figure 2. Power and Ground Placement

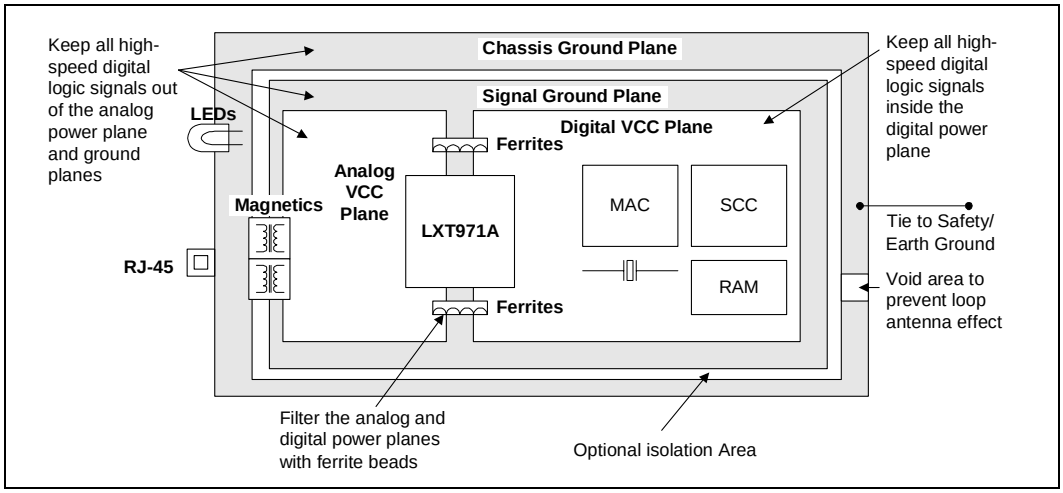
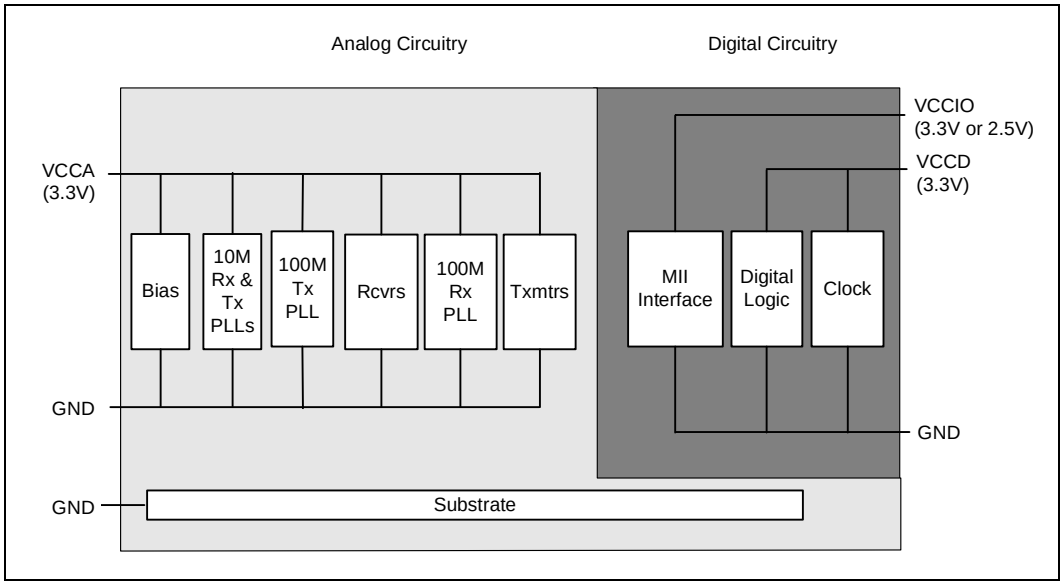


Figure 3. Internal Routing of Analog and Digital Power Signals



3.2 Design Considerations

Power supply ripple and digital switching noise can be created by:

- Poorly-regulated or over-burdened power supplies
- Data busses running at high clock rates
- DC-to-DC converters

Noise created by these sources can be coupled into the transmitter and receiver and out onto the network. Coupling may also occur through the LXT971A analog power and ground pins or other termination circuits (magnetic center taps). See the Network Interface section on page 16. This condition contributes to EMI and data corruption.

Use the criteria in Table 2 for evaluating acceptable noise levels in the analog region of the power and ground planes.

Table 2. Criteria for Analog Noise Levels

Noise Level	Acceptability
Under 50 mV	Acceptable
50 mV to 80 mV	Marginally Acceptable
Above 80 mV	Unacceptable

3.3 Design Implementation

Following good general design and layout guidelines prevents most common signal and noise issues. The following recommendations apply to the design and layout of the power and ground planes:

- Divide the VCC plane into two sections as shown in Figure 2 on page 12 (analog and digital). The break between the two planes should run under the device.
- When dividing the VCC plane, it is not necessary to add extra layers to the board. Simply create moats or cut-out regions in existing layers.
- Join the digital and analog sections at one or more points by ferrite beads. Ensure the maximum current rating of each bead is at least 150% of the nominal current that is expected to flow through it. Each LXT971A and its transformer draws a maximum of 65 mA from the analog supply so beads rated at 100 mA should be used. See Figure 4 on page 14 for current load listings.
- Place a bulk capacitor (10 µF) on each side of each ferrite bead to stop switching noise from traveling through the ferrite.
- For designs with multiple LXT971As, it is acceptable to supply all from one analog VCC plane. This plane can be joined to the digital VCC plane at multiple points, with a ferrite bead at each one. It is also acceptable to create an individual analog VCC *mini-plane* for each device.
- To improve EMI performance, use a ferrite bead between the analog voltage plane and the magnetic transmit center tap as shown in Figure 4 on page 14.

- Place a high-frequency bypass cap (.01 µf) near each VCC pin as shown in Figure 5.
- Place a 10 µF bulk capacitor between VCCIO and GND close to the device.
- Use a continuous, unbroken ground plane.

Figure 4. Power Supply Current

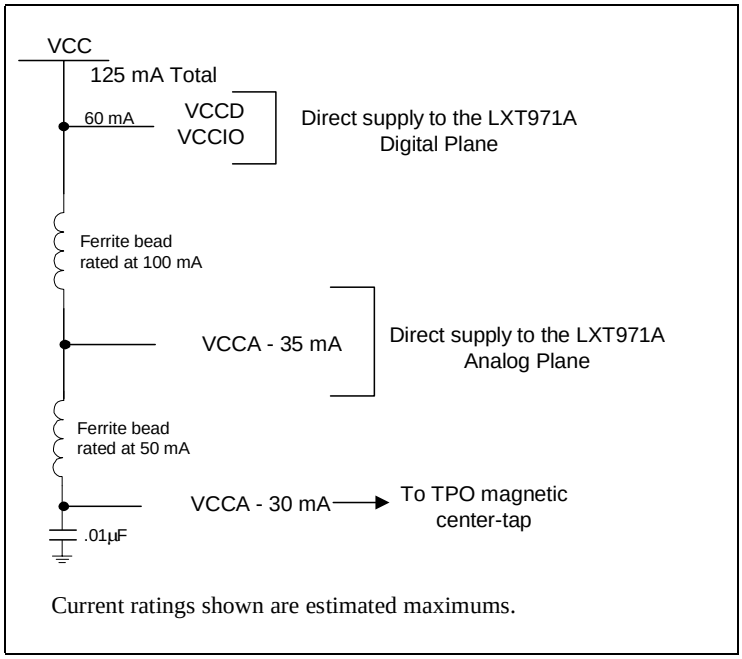
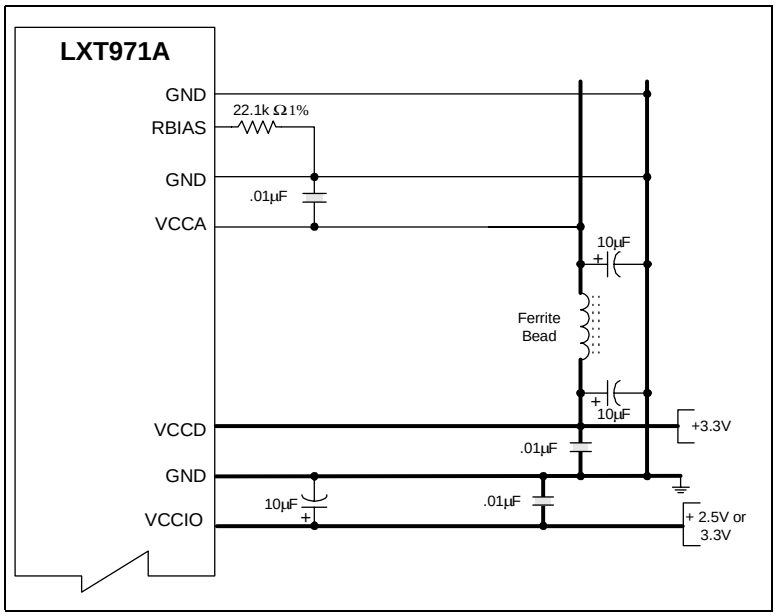


Figure 5. Power and Ground Decoupling

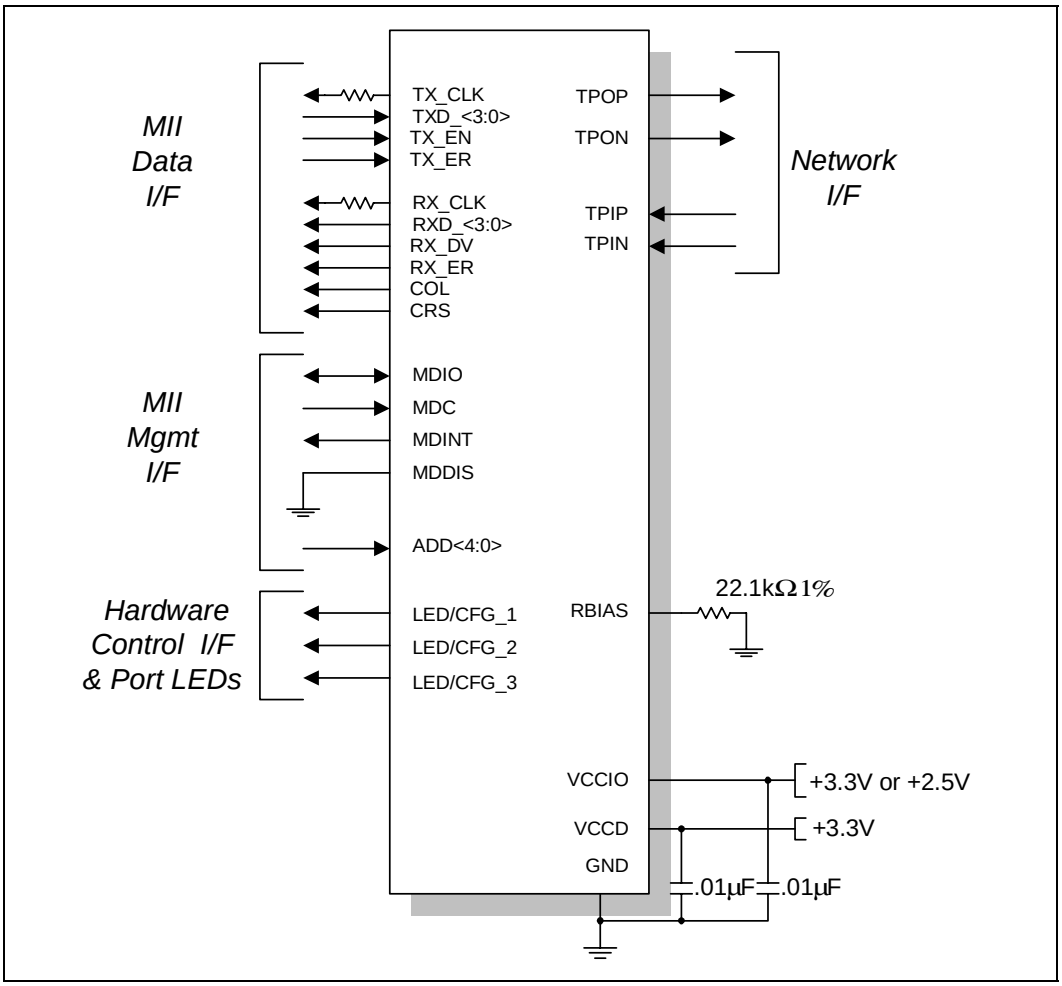


4.0 MII Interface

The LXT971A MII uses nine signals to pass received data to the MAC (RXD<3:0>, RX_CLK, RX_DV, RX_ER, COL, and CRS). There are seven signals used to transmit data from the MAC (TXD<3:0>, TX_CLK, TX_EN, and TX_ER). The MII operates at 25 MHz for 100 Mbps links and 2.5 MHz for 10 Mbps links.

The LXT971A MII has high output impedance (250 - 350Ω) and normally only requires termination on the data and status output signals in designs with long traces (>3 inches). Series termination resistors are strongly advised on the RX_CLK and TX_CLK signals to minimize reflections. Place the resistor as close to the device as possible. Use a software trace termination package to select an optimal resistance value for the specific trace. If this is not possible, use a 50Ω resistor value. Figure 6 shows the MII interface for the LXT971A.

Figure 6. LXT971A MII Interface



5.0 Network Interfaces

5.1 Twisted-Pair Interface

The twisted-pair interface consists of magnetics, connectors, and termination networks for the receiver and transmitter. The LXT971A requires magnetics with a 1:1 turns ratio for both the receive and the transmit transformers. A circuit known as a “Bob Smith” termination (see Figure 10 on page 20) may be used to ground unused signal pairs.

5.1.1 Receive Interface Circuit

The receive interface circuit consists of magnetics including a main winding, common-mode choke, and external termination resistance matching the line impedance.

5.1.1.1 Common-Mode Choke

Receive magnetics generally include a common-mode choke. Some vendors place this filter on the line (primary) side of the main winding; others place it on the device (secondary) side. Either approach is acceptable.

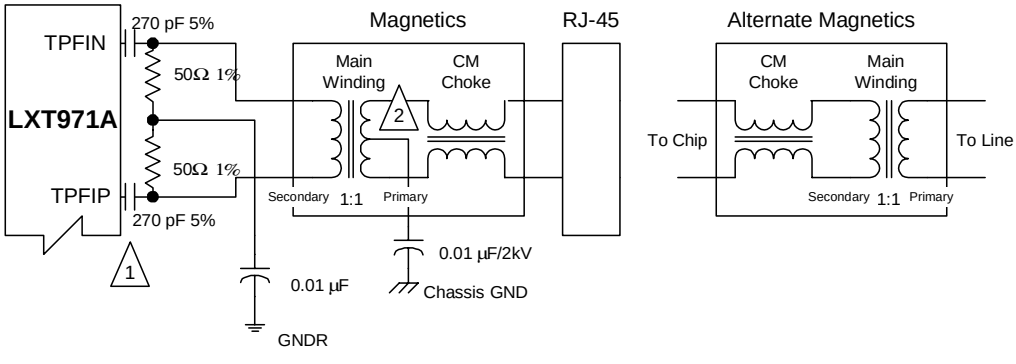
If using a magnetic with the common-mode choke on the device side, do not attach a bypass cap from the device-side center tap to ground. Noise from the ground can couple through the cap into the center tap, bypassing the common-mode choke, and cause EMI problems.

5.1.1.2 Termination Circuitry

Figure 7 shows the recommended receive termination. A 100Ω load is placed across the receive TPFIP/TPFIN input pair. This is accomplished using two 50Ω resistors with a common-mode bypass capacitor (0.01 µF) to ground. This provides additional common-mode shielding (when the reference ground is quiet) and a potential discharge path for ESD events on the receiver.

The 270 pF coupling capacitors work with the receiver circuitry of the LXT971A improving the signal-to-noise ratio for the receiver at long line lengths. Place the 270 pF series coupling capacitors as close to the LXT971A as possible.

Figure 7. Receive Interface Circuitry



1. Place these capacitors as close to the LXT971A as possible.
2. Magnetics without a receive pair center tap do not require a 2 kV termination.

5.1.2 Transmit Interface Circuit

The recommended termination circuitry with the magnetics on the transmit interface for both Switch and NIC RJ-45 configurations are shown in Figure 8A and Figure 9A. This circuit includes:

- Magnetic center tap (device-side) tied to VCCA via a ferrite bead and bypassed to GND using .01 µF capacitor
- One ferrite bead per device, rated at 50 mA to supply center tap current

Designs requiring reduced power can be supplied with an alternative 2.5V power source on the magnetics center tap (device side) instead of VCCA as shown in Figure 8B and Figure 9B. This saves up to 25 mW of system power.

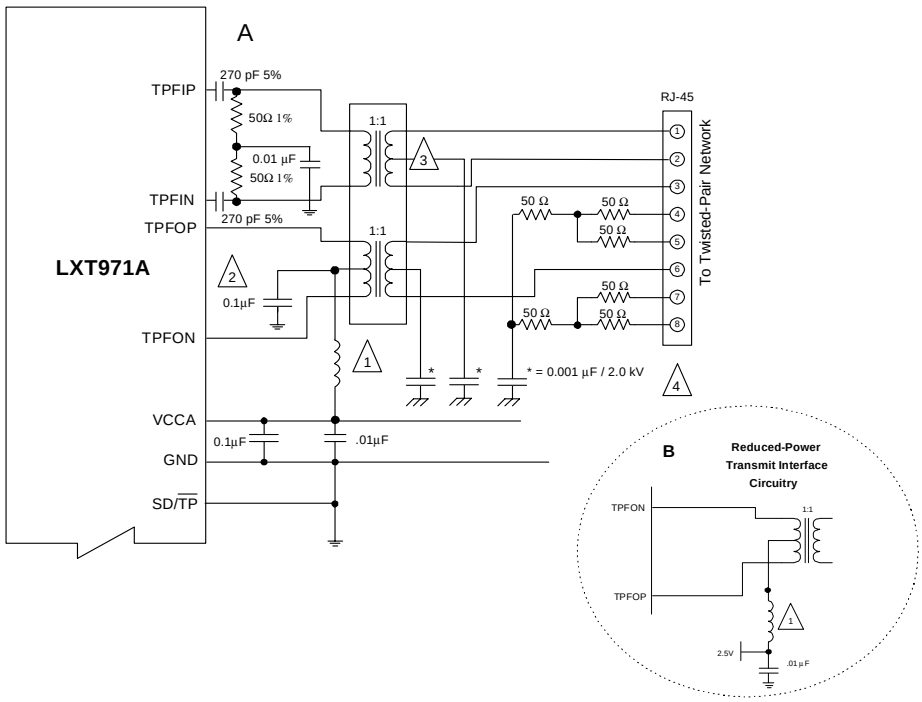
The output stage of the transmitter shown in Figure 8A and Figure 9A is designed to match the 100Ω characteristic impedance of an unshielded CAT5 twisted-pair wire. The external resistor that is typically required for impedance matching is integrated in the transmitter of the LXT971A. The internal termination provides a constant current reference in both 10BASE-T and 100BASE-TX applications and meets all IEEE transmitter requirements such as return loss, while reducing external component requirements. It has no impact in fiber designs

5.1.2.1 Common-Mode Choke

The transmit magnetics always include a common-mode choke. Some vendors place this choke on the line-side (secondary) of the main winding while others place it on the device-side (primary). A few vendors include two transmit chokes — one on each side of the main winding.

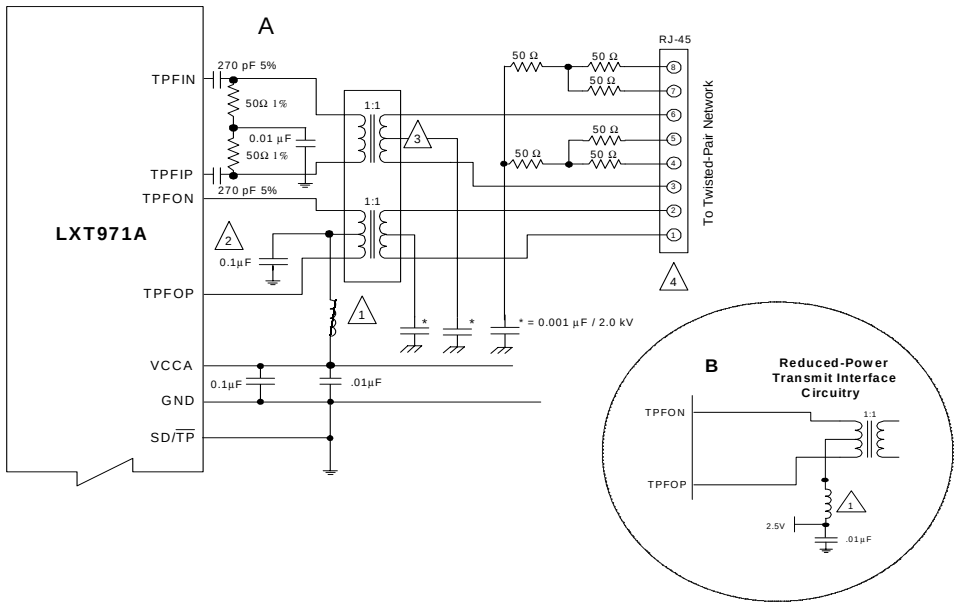
The line-side center tap can be bypassed to chassis ground, but this should be carefully evaluated in the system application. Bypassing both center taps of the transmit winding may produce undesirable results by creating a low-impedance AC coupling between the chassis ground and circuit ground. Consider potential noise sources and ground plane characteristics when evaluating bypass options.

Figure 8. Typical Twisted-Pair Interface - Switch



1. Center tap current may be supplied from 3.3V VCA as shown. Additional power savings may be realized by supplying the center tap from a 2.5V current source. A separate ferrite bead (rated at 50 mA) should be used to supply center tap current.
2. The 100Ω transmit load termination resistor typically required is integrated in the LXT971A.
3. Magnetics without a receive pair center tap do not require a 2 kV termination.
4. RJ-45 connections shown are for a standard switch application. For a standard NIC RJ-45 setup, see Figure 9 on page 19.

Figure 9. Typical Twisted-Pair Interface - NIC



1. Center tap current may be supplied from 3.3V VCA as shown. Additional power savings may be realized by supplying the center tap from a 2.5V current source. A separate ferrite bead (rated at 50 mA) should be used to supply center tap current.
2. The 100Ω transmit load termination resistor typically required is integrated in the LXT971A.
3. Magnetics without a receive pair center tap do not require a 2 kV termination.
4. RJ-45 connections shown are for a standard NIC application. Tx/Rx crossover may be required for repeater and switch applications.

5.1.2.2 Meeting IEEE Requirements

Designers should focus on two key areas to optimize return loss performance with the LXT971A. First, minimize shunt capacitance on the board, and second, carefully select the magnetics.

Adherence to the following guidelines helps to ensure each design meets IEEE requirements for the 100BASE-TX PMD layer as called out in the ANSI X3.263 specification.

Guidelines for Reducing System Shunt Capacitance.

- Avoid multiple layer changes in TPFON/P and TPFIN/P signal routing.
- Keep the magnetics as close as possible to the LXT971A, and keep TPFOP and TPFON traces as short as possible.
- In multi-chip applications, use quad magnetics optimized for dual-high RJ-45 connectors to allow the most compact layout.
- Use the termination circuit shown in Figure 10 on page 20.
- Provide EMI shielding by placing a ground plane under TPFOP and TPFON and the magnetics. To achieve an optimum layout for EMI and return loss performance, place the shielding ground plane two to three layers away to minimize shunt capacitance between the traces and the ground plane.

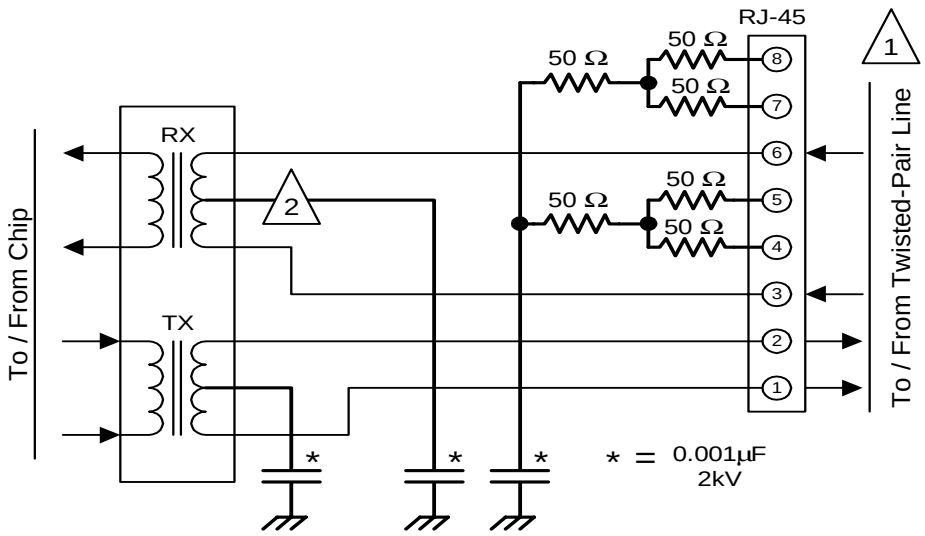
5.1.3 Bob Smith Termination

A "Bob Smith" termination is provided for the unused signal pairs of the twisted-pair interface (RJ-45 pins 4, 5, 7, and 8) and the media-side center taps. The circuit is used to enhance EMI and ESD performance of the system. Although there are many variations of this technique, one common implementation is shown in Figure 10. Note the signals are referenced to chassis ground rather than circuit ground.

A Bob Smith termination can be broken down into two circuits. One circuit provides termination for the unused signal pairs of the twisted-pair interface. The unused pairs are connected together through a 75Ω impedance matching circuit to chassis ground through a 0.001 µF, 2 kV capacitor. The capacitor provides a discharge path for noise immunity on the unused pairs.

The second circuit provides termination for the media-side center taps and is comprised of individual 0.001 µF, 2 kV capacitors to chassis ground. Separate capacitors are used for the receive and transmit center taps. This improves isolation by eliminating the low impedance path between receiver and transmitter that would exist if a single common cap were used. The capacitors provide a high-frequency path to ground, enhancing ESD and EMI performance.

Figure 10. Bob Smith Termination Circuit



1. RJ-45 connections shown for standard NIC. TxRx crossover may be required for switch applications.
2. Magnetics without a receive pair center tap do not require a 2 kV termination.

5.2 Magnetic Requirements

The LXT971A requires a 1:1 ratio for both the receive transformers and the transmit transformers. The transmit isolation voltage should be rated at 1.5 kV to protect the circuitry from static voltages across the connectors and cables. Refer to Table 3 for magnetics requirements.

Refer to Table 4 for a list of magnetic manufacturers and part numbers. This list constitutes a reference only and is not a recommendation. The system designer must ensure that all components, both individually and collectively, are suitable for the intended application.

Table 3. Magnetic Requirements

Parameter	Min	Nom	Max	Units	Test Condition
Rx turns ratio	–	1 : 1	–	–	–
Tx turns ratio	–	1 : 1	–	–	–
Insertion loss	0.0	0.6	1.1	dB	–
Primary inductance	350	–	–	µH	–
Transformer isolation	–	1.5	–	kV	–
Differential to common mode rejection	40	–	–	dB	.1 to 60 MHz
	35	–	–	dB	60 to 100 MHz
Return Loss	-16	–	–	dB	30 MHz
	-10	–	–	dB	80 MHz

Table 4. Magnetic Manufacturers

Port/Ratio	Manufacturer[1]	Temperature	Part Numbers Single-Port
Rx = 1:1 Tx = 1:1	BELFUSE	Commercial	S558-5999-T7
		Industrial	S558-5999-T5
	DELTA	Industrial	LF8416
	HALO	Commercial	TG110-S050N2 TG110-S050P2
		Industrial	TG110-E050N5 TG22-E150NL
	PULSE	Commercial	H1102
		Industrial	Hx1148

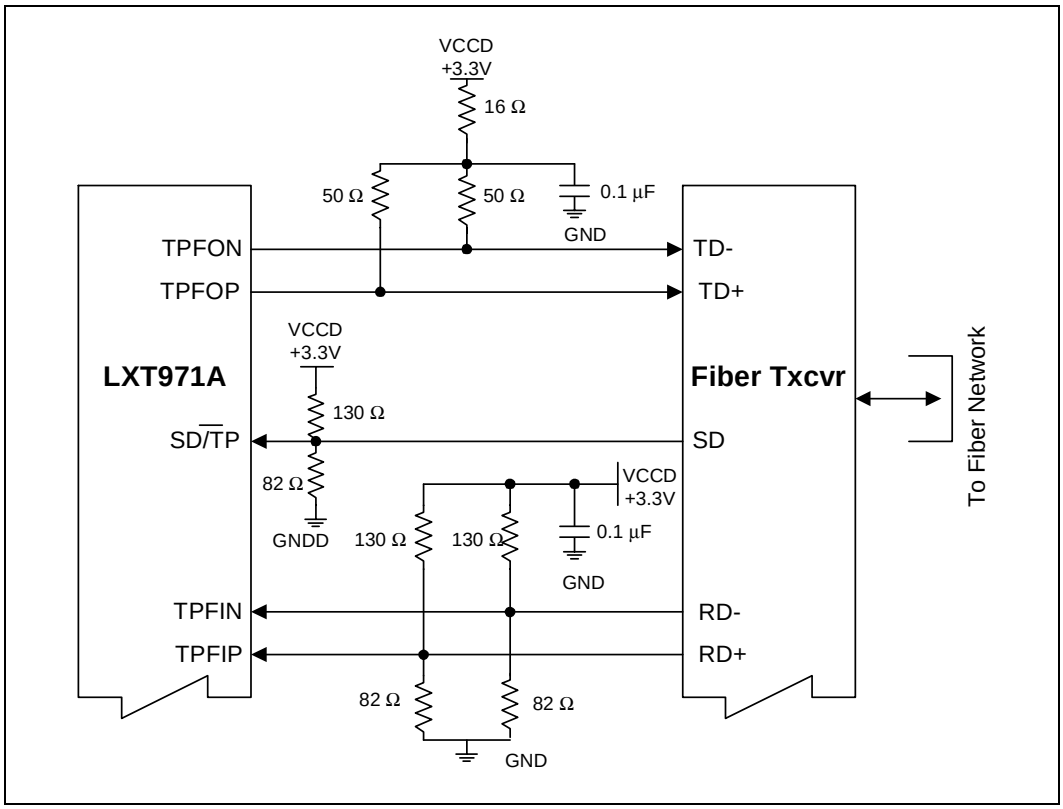
1. Device manufacturers may have additional magnetics with varying pinouts.

5.3 Fiber Interface

The fiber interface consists of two pseudo-ECL (PECL) signal pairs that attach to an external fiber optic transceiver. Both fiber data pairs (TPFOP/N and TPFIP/N) should be DC-coupled to the fiber transceiver.

Figure 11 shows both circuits. The combinations of bias resistors shown provide the ideal biasing points for an equivalent load impedance of 50Ω.

Figure 11. Fiber Interface Circuit



Appendix A LXT971A Design and Layout Checklist

A.1 Power and Ground

A.1.1 Design

- ❑ Ensure that the power and ground noise levels are below 50 mV
- ❑ Ensure that the power supply is properly rated for the entire board load
- ❑ Use bulk capacitors (4.7 - 10 µF) between the power and ground planes to minimize power supply switching noise.
- ❑ Filter and shield DC-DC converters, oscillators, etc.
- ❑ Use an ample supply of .01 µF decoupling capacitors to reduce high-frequency noise on the power and ground planes.
- ❑ Join the digital and analog sections at one or more points by ferrite beads. Ensure the maximum current rating of each bead is at least 150% of the nominal current that is expected to flow through it. Each LXT971A and its transformer draws a maximum of 65 mA from the analog supply, so beads rated at 100 mA should be used.
- ❑ Place a bulk capacitor (10 µF) on each side of each ferrite bead to stop switching noise from traveling through the ferrite.
- ❑ Place a 10 µF bulk capacitor, close to the device, between VCCIO and GND.
- ❑ Place a high-frequency bypass cap (.01 µF) near each VCC pin.

A.1.2 Layout

- ❑ Avoid breaks in the ground plane, especially in areas where the ground plane is shielding high-frequency signals.
- ❑ Route high-speed signals above a continuous, unbroken ground plane.
- ❑ When possible, fill in unused areas of the signal planes with solid copper and attach them with vias to a VCC or ground plane that is not located adjacent to the signal layer. This technique is referred to as signal layer filling and can improve capacitive coupling of the power planes.
- ❑ Use a continuous, unbroken ground plane.

A.2 System Clock

A.2.1 Design

- Ensure that the system clock is a 25 MHz ± 100 ppm reference clock (REFCLK) that must be enabled at all times.
- Ensure that the duty cycle distortion is no greater that 35 to 65%.
- Ensure that the TTL voltage levels are VOH > 2.0V.
- If you have issues linking at 100 Mbps speeds, but can link at 10 Mbps, try implementing one of the crystal or crystal oscillator devices specified in Table 1 on page 10 before contacting Intel Customer Support.

A.2.2 Layout

- Keep the clock traces as short as possible.
- Route the clock traces adjacent to an unbroken ground plane.
- Use a multi-output clock driver when driving multiple inputs with a single oscillator.
- Individually terminate point-to-point interconnects to every clock load. Series termination is the most common termination technique.

A.3 MII Interface

A.3.1 Design

- The LXT971A MII have high output impedance (250 - 350Ω) and normally require termination on the data and status output signals in designs with long traces (>3 inches).
- Series termination resistors are strongly recommended on the RX_CLK and TX_CLK signals to minimize reflections. Place the resistor as close to the LXT971A as possible.
- Use a trace termination software modeling application to select an optimal resistance value for the specific trace. If this is not possible, use a 50W resistor value.

A.4 Twisted-Pair Interface

A.4.1 Design

- Ensure that the RBIAS resistor is a 22.1 kΩ 1% resistor to ground for internal reference current setup. Place the resistor close to the LXT971A.
- The LXT971A/LXT972A has integrated transmit termination and does not require external 100Ω termination to work properly.
- If you have issues linking or communicating over the twisted-pair or fiber interface, please verify that the proper termination values are populated on the board prior to escalating or contacting Intel Customer Support.

A.4.2 Layout

- Route differential pairs close together and away from other signals.
- Keep both traces of each differential pair as identical to each other as possible.
- Keep each differential pair on the same plane.
- Minimize vias and layer changes.
- Keep transmit and receive pairs away from each other. Run orthogonally, or separate with a ground plane layer. To maintain this separation, place all components for the transmit circuit on one side of the board and all components for the receive circuit on the other side of the board.
- To improve EMI performance, use a ferrite bead between the analog voltage plane and the magnetic transmit center tap.

A.5 Fiber Interface

A.5.1 Design

- Sleep mode is not functional in fiber network applications.

A.5.2 Layout

- If you have issues linking or communicating over the twisted-pair or fiber interface, please verify that the proper termination values are populated on the board prior to escalating or contacting Intel Customer Support.

A.6 Magnetics

A.6.3 Design

- Transmit isolation voltage should be rated at 1.5 kV to protect the circuitry from static voltages across the connectors and cables.
- Ensure that the magnetic transformer ratio is maintained with Rx = 1:1 and Tx = 1:1.
- If using a magnetic with the common-mode choke on the device side, do not attach a bypass cap from the device-side center tap to ground. Noise from the ground can couple through the cap into the center tap, bypassing the common-mode choke, and cause EMI problems.

A.6.4 Layout

- Void power and ground planes directly under the magnetics. Use chassis ground in the area from the magnetics to the RJ-45 connector.
- Keep high-speed signals out of the area between the LXT971A and the magnetics.
- Do not route any digital signals between the LXT971A and the RJ-45 connectors at the edge of the board.

A.7 General Design

- Validate the value of the RBIAS resistor with a DMM to ensure that the proper value has been populated on the board.
- Download the latest revision of the specification updated from the following URL (**http://developer.intel.com/design/network/products/physlayer/index.htm#Transceivers**). Identify your stepping using the topside markings of the chip, and implement the appropriate workarounds necessary for your design.
- Changes have been made to the existing documentation to include or clarify specifications or explanations over the life of the document. Ensure that you have the latest revision of the LXT971A and LXT972A Datasheets from the URL noted above if documentation is unclear or the specification is missing.