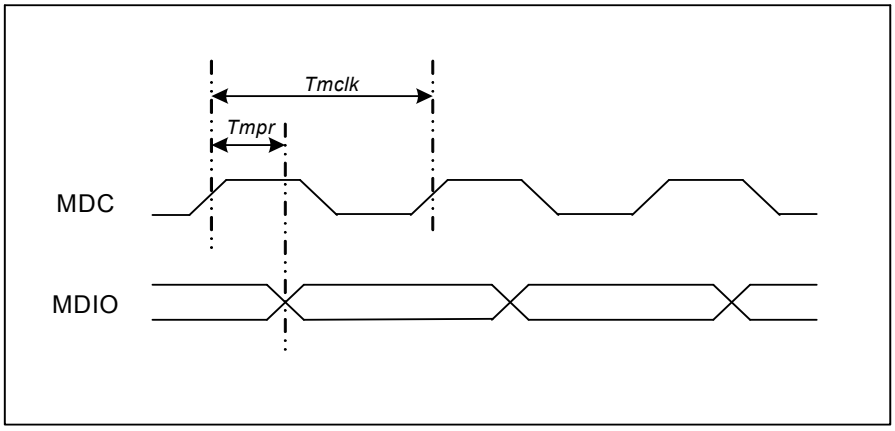


Figure 11-10 MDIO Output AC Timing Diagram

11.4.2.8 Reset Signals

For power-on indication, the 82599 can either use an internal power-on circuit, which monitors the 1.2V power supply, or external reset using the LAN_PWR_GOOD pin. The POR_BYPASS pin defines the reset source (when high, the device uses the LAN_PWR_GOOD pad as power-on indication).

The timing between the power-up sequence and the different reset signals when using the internal power indication is described in Section 11.3.1.1.

The BYPASS mode is described in Section 11.4.2.8.1.

A schematic of the power-on logic can be found in Figure 11-11:

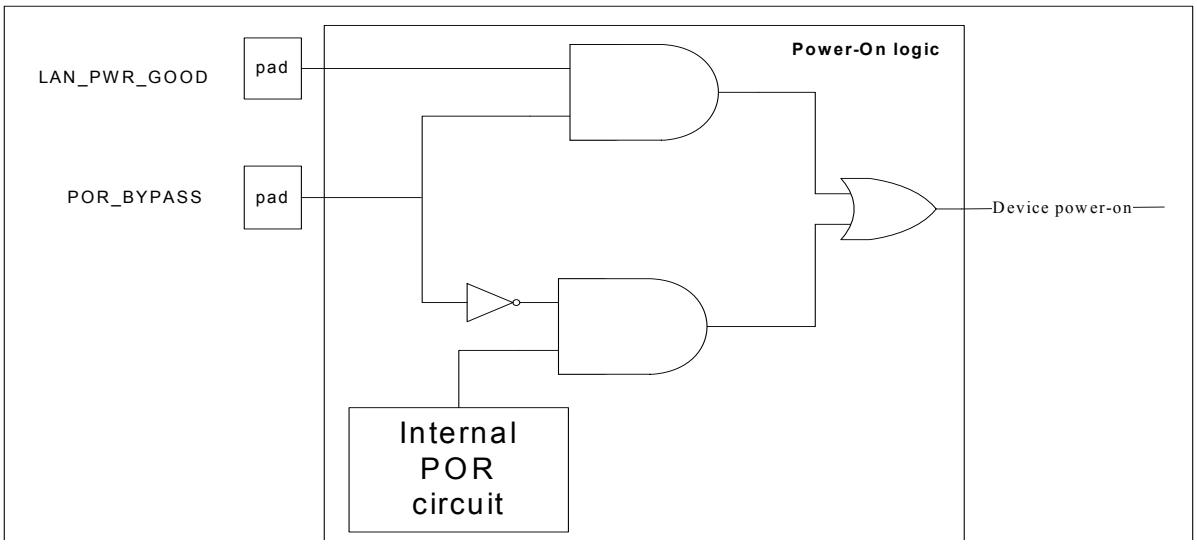


Figure 11-11 Power-On Reset Logic

11.4.2.8.1 Power-On Reset BYPASS

When asserting the POR_BYPASS pad, the 82599 uses the LAN_PWR_GOOD pin as power-on indication. Otherwise, the 82599 uses an internal power on detection circuit in order to generate the internal power on reset signal.

Table 11-24 lists the timing for the external power-on signal.

Table 11-24 External Reset Specification

Symbol	Title	Description	Min	Max	Units
Tlpgw	LAN_PWR_GOOD Minimum Width	Minimum width for LAN_PWR_GOOD.	5	N/A	µs
Tlpg	LAN_PWR_GOOD High Hold	Hold time following power-up (power supplies in acceptable operating range).	40	80	ms

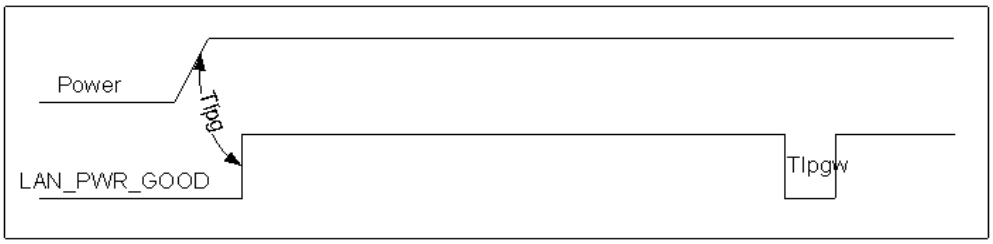


Figure 11-12 External Reset Timing Diagram

11.4.3 PCIe Interface AC/DC Specification

The 82599 PCIe interface supports the electrical specifications defined in:

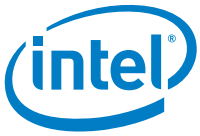
- PCI Express* 2.0 Card Electromechanical Specification.
- PCI Express* 2.0 Base Specification, Chapter 4.

Note: Reference clock specifications are detailed in both the base and CEM specification. Please consult both specifications to understand the full set of requirements. Sections 4.3.7 (Base specification) and 2.1.3 (CEM specification) in particular.

11.4.4 Network (MAUI) Interface AC/DC Specification

11.4.4.1 KR Interface AC/DC Specification

The 82599 MAUI interface supports the 10GBASE-KR electrical specification defined in IEEE802.3ap clause 72.



11.4.4.2 SFP+ Interface AC/DC Specification

The 82599 MAUI interface supports the SFI electrical specification defined in the SFP+ MSA (SFF Committee SFF-8431).

11.4.4.3 KX4 Interface AC/DC Specification

The 82599 MAUI interface supports the 10GBASE-KX4 electrical specification defined in IEEE802.3ap clause 71.

11.4.4.4 BX4 Interface AC/DC Specification

The 82599 MAUI interface supports the 10GBASE-BX4 electrical specification defined in PICMG 3.1.

11.4.4.5 CX4 Interface AC/DC Specification

The 82599 MAUI interface supports the 10GBASE-CX4 electrical specification defined in IEEE802.3ak clause 54.

11.4.4.6 XAUI Interface AC/DC Specification

The 82599 MAUI interface supports the 10G XAUI electrical specification defined in IEEE802.3ae clause 47.

11.4.4.7 KX Interface AC/DC Specification

The 82599 MAUI interface supports the 1000BASE-KX electrical specification defined in IEEE802.3ap clause 70.

11.4.4.8 BX Interface AC/DC Specification

The 82599 MAUI interface supports the 1000BASE-BX electrical specification defined in PICMG® 3.1 specification.

11.4.5 SerDes Crystal/Reference Clock Specification

The 82599 SerDes clock can be supplied either by connecting an external differential oscillator of 25 MHz or an external 25 MHz crystal. SerDes clock frequency is set by the OSC_FREQ_SE pin and crystal vs. oscillator are selected by the OSC_SEL pin.

The figures below show connection options to the REFCLKIN_p/_n pins by using a crystal or by using an external oscillator (PECL or CML). These schemes are not part of The 82599 specifications but rather examples that meet the required specification.

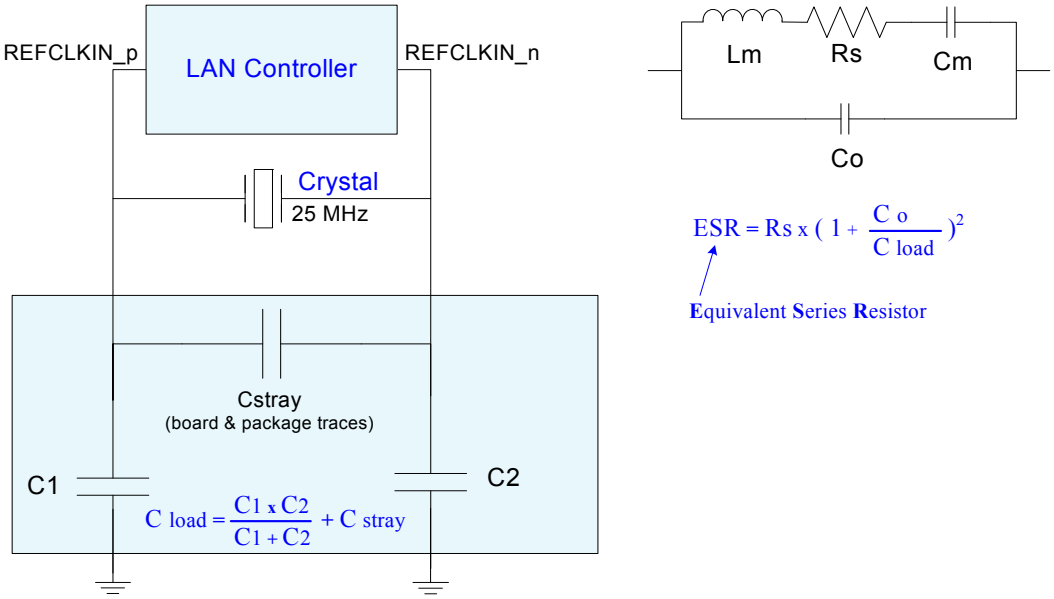
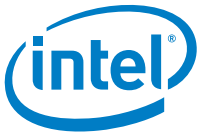


Figure 11-13 Crystal Connectivity



11.4.5.1 SerDes Crystal Specification

Table 11-25 SerDes Crystal Specifications

Parameter Name	Symbol	Recommended Value	Conditions
Frequency	f_o	25.000 [MHz]	@25 [°C]
Vibration Mode		Fundamental	
Cut		AT	
Operating /Calibration Mode		Parallel	
Frequency Tolerance @25 °C	Df/f_o @25 °C	±30 [ppm]	@25 [°C]
Temperature Tolerance	Df/f_o	±30 [ppm]	
Operating Temperature	T_{opr}	-20 to +70 [°C]	
Non Operating Temperature Range	T_{opr}	-40 to +90 [°C]	
Equivalent Series Resistance (ESR)	ESR	50 [Ω] maximum	@25 [MHz]
Load Capacitance	C_{load}	20 [pF]	
Shunt Capacitance	C_o	7 [pF] maximum	
Pullability From Nominal Load Capacitance	Df/C_{load}	15 [ppm/pF] maximum	
Max Drive Level	D_L	750 [µW]	
Insulation Resistance	IR	500 [MW] minimum	@ 100V DC
Aging	Df/f_o	±5 [ppm/year]	
External Capacitors	C_1, C_2	20 [pF]	
Board Resistance	R_s	0.1 [W]	

11.4.5.2 SerDes Reference Clock Specification

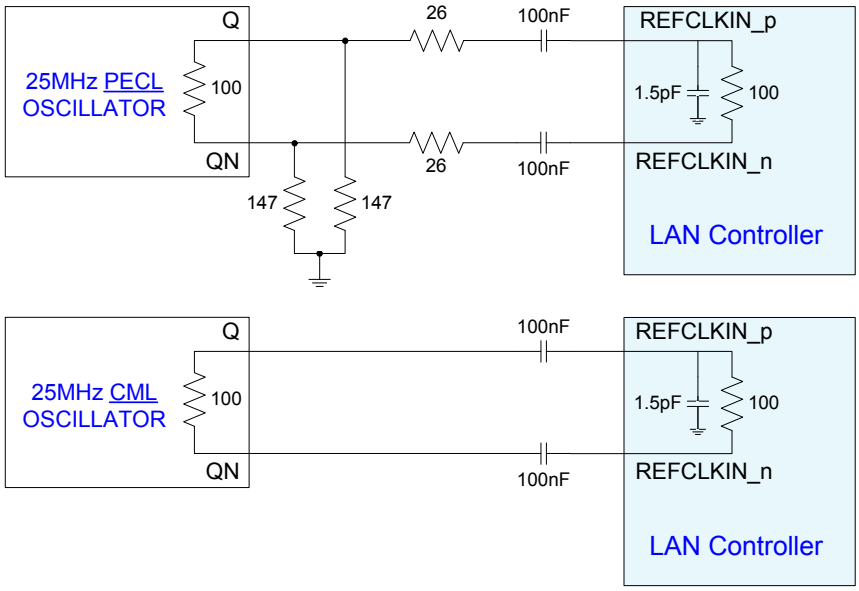


Figure 11-14 Example Diagram for External PECL or CML Oscillator Connectivity

Table 11-26 Input Reference Clock Electrical Characteristics

Sym	Parameter	Min	Typ	Max	Unit	Comments
f	Frequency		25		MHz	
Δf	Frequency Variation	-100		+100	ppm	
DC	Duty Cycle	40		60	%	
Tr	Rise Time (20% - 80%)	300		1000	ps	
Tf	Fall Time (20% - 80%)	300		1000	ps	
AMP	Differential Peak-to-Peak Amplitude	0.6		1.25	V	
R	Differential Termination Resistance		100		Ω	
C	AC Coupling		100		nF	
Cin	Input Capacitance		1.5		pF	

Table 11-26 Input Reference Clock Electrical Characteristics (Continued)

Sym	Parameter	Min	Typ	Max	Unit	Comments
DJ P2P	Deterministic Peak-to-Peak Jitter			10	ps	
p-noise	Phase Noise (high-speed serial - KR and SFI)		-145		dBc/Hz	See Figure 11-16 for phase noise graph
p-noise	Phase Noise (non-high speed serial)		-136		dBc/Hz	See Figure 11-17 for phase noise graph

Note: Intel recommends designing to the high-speed serial p-noise specification allowing for maximum flexibility with link configuration options.

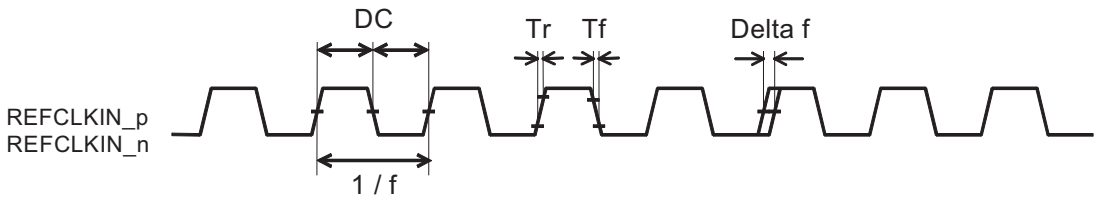


Figure 11-15 External Clock Reference Timing

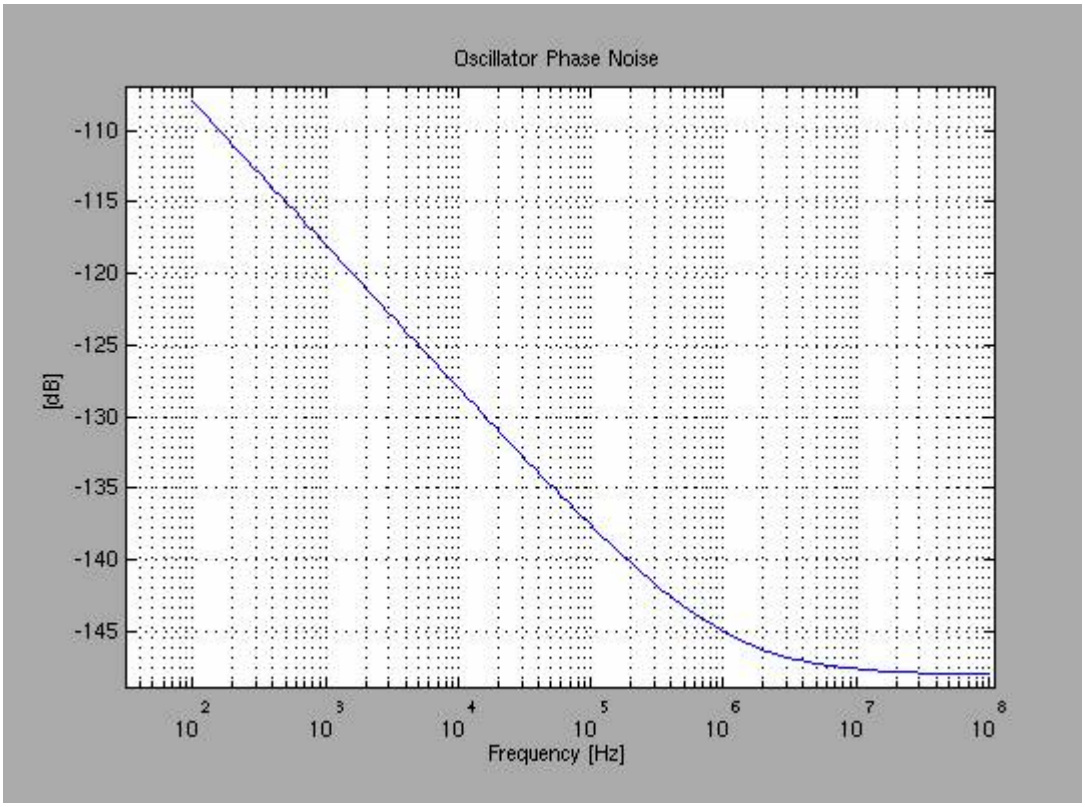


Figure 11-16 Maximum External Oscillator Phase Noise as a Function of Frequency (High-Speed Serial)

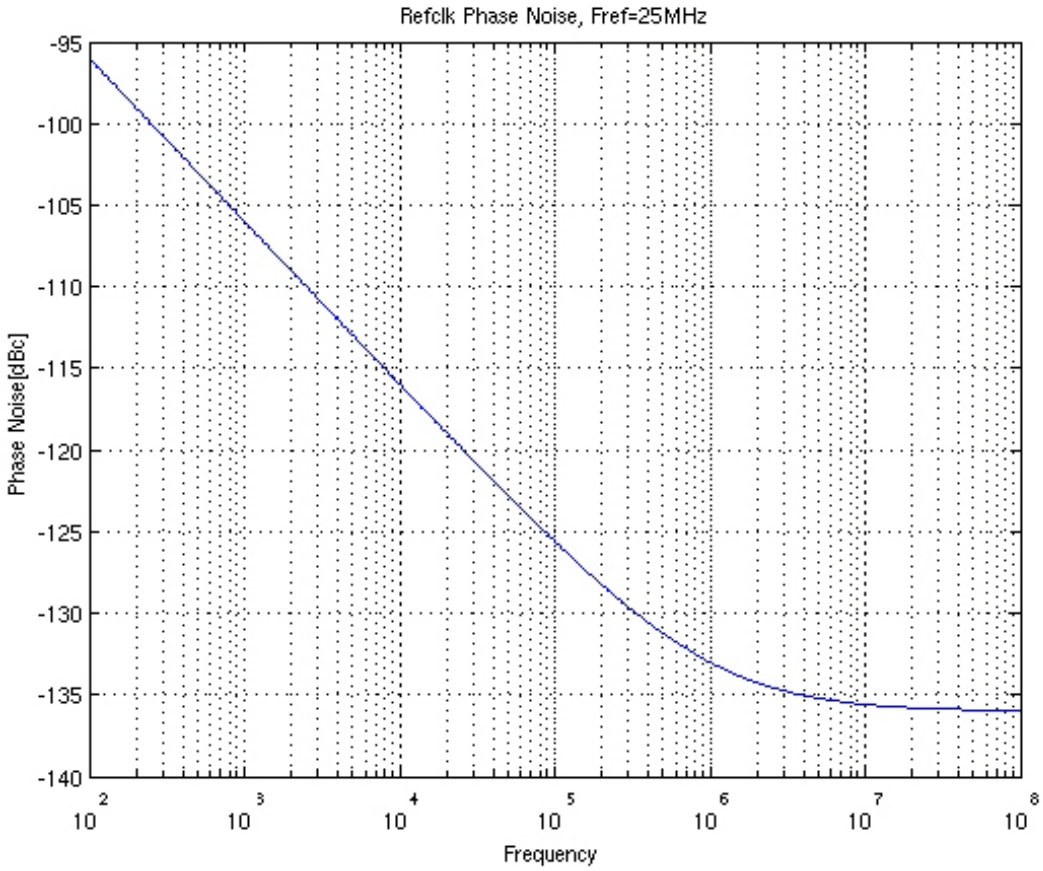


Figure 11-17 Refclk Phase Noise as a Function of Frequency (Non-High Speed Serial)

11.5 Package

11.5.1 Mechanical

The 82599 is assembled in a 25 x 25 FCBGA package with an 8-layer substrate.

Table 11-27 Package Specifications

Body Size	Ball Count	Ball Pitch	Ball Matrix	Substrate
25x25 mm^2	576	1 mm	24 X 24	Eight Layers

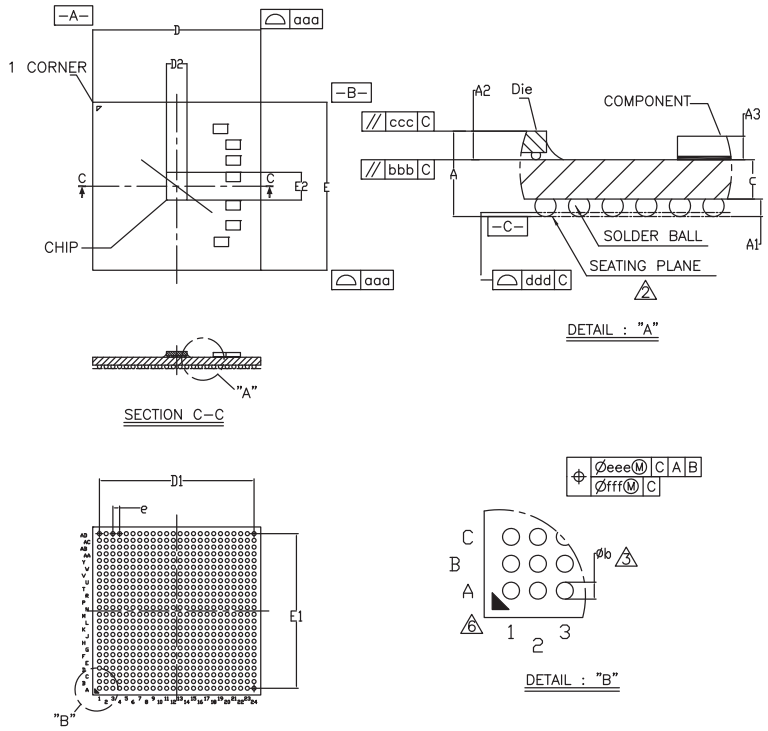
11.5.2 Thermal

For the 82599's package thermals please refer to Section 13.0.

11.5.3 Electrical

Package electrical models are part of the IBIS files.

11.5.4 Mechanical Package



Symbol	Dimension in mm			Dimension in inch		
	MIN	NOM	MAX	MIN	NOM	MAX
A	2.325	2.540	2.755	0.092	0.100	0.108
A1	0.44	0.52	0.60	0.017	0.020	0.024
A2	0.80	0.85	0.90	0.031	0.033	0.035
A3	---	---	0.70	---	---	0.028
c	1.085	1.170	1.255	0.043	0.046	0.049
D/E	24.95	25.00	25.05	0.982	0.984	0.986
D1/E1	---	23.00	---	---	0.906	---
e	---	1.00	---	---	0.039	---
b	0.55	0.60	0.65	0.022	0.024	0.024
aaa	0.20			0.008		
bbb	0.25			0.010		
ccc	0.20			0.008		
ddd	0.20			0.008		
eee	0.25			0.010		
fff	0.10			0.004		

	Die size type (mm)	Die size type (inch)
	Niantic	Niantic
D2	9.44 REF	0.372 REF
E2	7.83 REF	0.308 REF

NOTE :

1. CONTROLLING DIMENSION : MILLIMETER.
2. PRIMARY DATUM C AND SEATING PLANE ARE DEFINED BY THE SPHERICAL CROWNS OF THE SOLDER BALLS.
3. DIMENSION b IS MEASURED AT THE MAXIMUM SOLDER BALL DIAMETER, PARALLEL TO PRIMARY DATUM C.
4. THERE SHALL BE A MINIMUM CLEARANCE OF 0.25mm BETWEEN THE EDGE OF THE SOLDER BALL AND THE BODY EDGE.
5. SPECIAL CHARACTERISTICS C CLASS: A , ddd

Figure 11-18 Mechanical Package

11.6 Devices Supported

11.6.1 Flash

The 82599 supports Flash devices with a SPI interface. Section 11-28 lists the specific Flash types supported,

Table 11-28 Supported Flash Devices

Density	Atmel PN	STM PN
1 Mb	AT25F1024N-10SI-2.7 or AT25FS010	M25P10-AVMN6T
2 Mb	AT25F2048N-10SI-2.7	M25P20-AVMN6T
4 Mb	AT25F4096N-10SI-2.7	M25P40-AVMN6T
8 Mb		M25P80-AVMN6T

Table 11-28 Supported Flash Devices (Continued)

Density	Atmel PN	STM PN
16 Mb		M25P16-AVMN6T
32 Mb		M25P32-AVMN6T

Notes:

1. Since all SPI Flash memories have similar interface characteristics, there is no need to test the interface with all the proposed types. It's acceptable to test the largest Flash.
2. All supported Flashes have an address size of 24 bits.

11.6.2 EEPROM

Section 11-29 lists the specific EEPROM devices supported.

Table 11-29 Supported EEPROM Devices

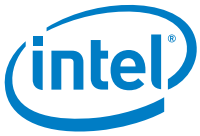
Density [Kb]	Atmel PN	STM PN	Catalyst PN
128	AT25128AN-10SI-2.7	M95128WMN6T	CAT25CS128-TE13
256	AT25256AN-10SI-2.7	M95256WMN6T	

11.6.2.1 Minimum EEPROM Sizes

- No manageability - 16 KB (128 Kb)
- SMBus/NC-SI - 32 KB (256 Kb)

11.6.2.2 Recommended EEPROM Sizes

- No manageability - 32 KB (256 Kb)
- SMBus/NC-SI - 32 KB (256 Kb)



12.0 Design Considerations and Guidelines

This section provides recommendations for selecting components, connecting interfaces, dealing with special pins, and layout guidance.

Some unused interfaces should be terminated with pull-up or pull-down resistors. These are indicated in Section 2.0 or reference schematics. There are reserved pins, identified as RSVD_3P3, RSVD_1P2 and RSVD_VSS. The 82599 might enter special test modes unless these strapping resistors are in place.

Some unused interfaces must be left open. Do not attach pull-up or pull-down resistors to any balls identified as No Connect or Reserved No Connect.

12.1 Connecting the PCIe Interface

The 82599 connects to the host system using a PCIe interface. The interface can be configured to operate in several link modes. These are detailed in Section 3.0. A link between the ports of two devices is a collection of lanes. Each lane has to be AC-coupled between its corresponding transmitter and receiver; with the AC-coupling capacitor located close to the transmitter side (within 1 inch). Each end of the link is terminated on the die into nominal 100 Ω differential DC impedance. Board termination is not required.

Refer to the *PCI Express* Base Specification, Revision 2.0* and *PCI Express* Card Electromechanical Specification, Revision 2.0.*

12.1.1 Link Width Configuration

The 82599 supports link widths of x8, x4, x2, or x1 as determined by the PCIe init configuration. The configuration is loaded using bits 9:4 in the *Max Link Width* field of the Link Capabilities register (0xAC). The 82599 default is the x8 link width.

During link configuration, the platform and the 82599 negotiate a common link width. In order for this to work, the selected maximum number of PCIe lanes must be connected to the host system.

12.1.2 Polarity Inversion and Lane Reversal

To ease routing, designers have the flexibility to use the lane reversal modes supported by the the 82599. Polarity inversion can also be used, since the polarity of each differential pair is detected during the link training sequence.

When lane reversal is used, some of the down-shift options are not available. For a description of available combinations, see Section 3.0.

12.1.3 PCIe Reference Clock

For LOM designs, the device requires a 100 MHz differential reference clock, denoted PE_CLK_p and PE_CLK_n. This signal is typically generated on the system board and routed to the PCIe port. For add-in cards, the clock is furnished at the PCIe connector.

The frequency tolerance for the PCIe reference clock is +/- 300 ppm.

12.1.4 PCIe Analog Bias Resistor

For proper biasing of the PCIe analog interface, a 24.9 Ω 0.5% resistor needs to be connected from the PE_RBIAS to the VCC1P2 supply. The PE_RSENSE pin should be connected directly to PE_RBIAS, as close as possible to the 24.9 Ω resistor pad. To avoid noise coupled onto this reference signal, place the bias resistor close to the 82599 and keep traces as short as possible.

12.1.5 Miscellaneous PCIe Signals

The 82599 signals power management events to the system by pulling low the PE_WAKE# signal. This signal operates like the PCI PME# signal. Note that somewhere in the system, this signal has to be pulled high to the auxiliary 3.3 V supply rail.

The PE_RST# signal, which serves as the familiar reset function for the 82599, needs to be connected to the host system's corresponding signal.

12.1.6 PCIe Layout Recommendations

For information regarding the PCIe signal routing, refer to the *Intel® 82599 10 GbE Controller Checklists for further layout guidance*.

12.2 Connecting the MAUI Interfaces

The 82599 has two high speed network interfaces that can be configured in different 1 Gb/s and 10 Gb/s modes (CX4, KX, KX4, KR, XAUI, KR, and SFI+). Choose the appropriate configuration for your system configuration.

12.2.1 MAUI Channels Lane Connections

For BX, KX, KR, and SFI+ connections, only the first lane needs to be connected (TX0_L0_p, TX0_L0_n; RX0_L0_p, RX0_L0_n). For the remainder of the interfaces, all four differential pairs need to be connected for each direction.

These signals are 100 Ω terminated differential signals that are AC coupled near the receiver. Place the AC coupling capacitors less than one inch away from the receiver. For recommended capacitor values, consult the relevant IEEE 802.3 specifications and/or the relevant PICMG specifications. Capacitor size should be small to reduce parasitic inductance. Use X5R or X7R, ±10% capacitors in a 0402 or 0201 package size.

Notes:

1. SFI+ board traces generally do not require AC coupling capacitors because they are normally integrated into the SFP+ module.
2. Unused pins can be left as a no connect (NC), including the pins for the port not present on the 82599EN single port SKU.

12.2.2 MAUI Bias Resistor

For proper biasing of the MAUI analog interface, a 1 KΩ ±0.5% resistor needs to be connected between the XA_RBIAS and XA_SENSE pins. To avoid noise coupled onto this reference signal, place the bias resistor close to the 82599 and keep traces as short as possible.

12.2.3 XAUI, KX/KR, BX4, CX4, BX and SFI+ Layout Recommendations

This section provides recommendations for routing the high-speed interface. The intent is to route this interface optimally using FR4 technology. Intel has tested and characterized these recommendations.

12.2.4 Board Stack-Up Example

Printed Circuit Boards (PCBs) for these designs typically have six, eight, or more layers. Although, the the 82599 does not dictate stackup, the following examples are of typical stack-up options.

Microstrip Example:

- Layer 1 is a signal layer.
- Layer 2 is a ground layer.
- Layer 3 is used for power planes.
- Layer 4 is a signal layer. Careful routing is necessary to prevent crosstalk with layer 5.
- Layer 5 is a signal layer. Careful routing is necessary to prevent crosstalk with layer 4.
- Layer 6 is used for power planes.
- Layer 7 is a signal ground layer.
- Layer 8 is a signal layer.

Note: Layers 4 and 5 should be used mostly for low-speed signals because they are referenced to potentially noisy power planes that might also be slotted.

Stripline Example:

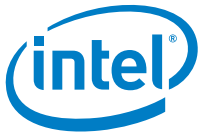
- Layer 1 is a signal layer.
- Layer 2 is a ground layer.
- Layer 3 is a signal layer.
- Layer 4 is used for power planes
- Layer 5 is used for power planes
- Layer 6 is a signal layer.
- Layer 7 is a signal ground layer.
- Layer 8 is a signal layer.

Note: To avoid the effect of the potentially noisy power planes on the high-speed signals, use offset stripline topology. The dielectric distance between the power plane and signal layer should be three times the distance between ground and signal layer.

This board stack-up configuration can be adjusted to conform to your company's design rules.

12.2.5 Trace Geometries

For 82599 SFI Designs, trace geometries are found in the Intel® 82599 10 Gigabit Ethernet Controller – SFI (SFP + Interface) Design Guide. Contact your Intel representative for details.



12.2.6 Other High-Speed Signal Routing Practices

These layout and routing recommendations are applicable for the MAUI interfaces of the the 82599.

In order to keep impedance continuity consistent around signal via anti-pad regions, Intel recommends adding the anti-pad diameter requirement of 9to 12 mils clearance to vias to ground and power. This ensures that the impedance variance is minimized. On plane layers, pairs of signal vias should share the same enlarged elliptical or oval (merged) anti-pads.

Enforce differential symmetry, even for grounds. Along with ensuring that the MAUI interface is routed symmetrically in terms of signal routing and balance, we also recommend that ground paths be routed symmetrically. This helps reduce the imbalance that can occur in the different return current paths.

In cases where there is a via and an AC coupling capacitor on the same trace, the signal trace between the via and the AC coupling capacitors on the MAUI interface, there is an intrinsic impedance mismatch because of the required capacitors. To minimize the overall impact of having vias and AC coupling capacitors, we recommend that both via and capacitor layout pad be placed within 100 mils of each other.

Note: For KR interfaces, this is not recommended unless simulations are performed and the results confirm minimal impact to impedance, insertion loss, insertion loss deviation, and crosstalk.

It is best to use a 0402 capacitor or smaller for the AC coupling components on the MAUI interface. The pad geometries for a 0402 or smaller components lend themselves to maintaining a more consistent transmission line environment. For 10 Gb/s KR, the recommended package size for the required AC coupling capacitors is the 0201 package size. Note that SFI+ board traces normally do not require AC coupling capacitors. Contact your Intel sales representative for more details.

Note: To reduce shunt capacitance from the AC capacitors' solder-pads to the reference plane beneath the solder-pads, we recommend that you void the reference plane that is directly under the capacitor. The reference plane void should have the same shape as the capacitor and it's solder pads. The size of the reference plane void should be slightly larger than the size of the capacitor and it's solder pad. If you have access to a 3-dimensional field solver, it can and should be used to determine the optimal size and shape for the reference plane void under each capacitor. To prevent noise problems, be careful not to route any traces across the capacitor-shaped voids in the reference plane.

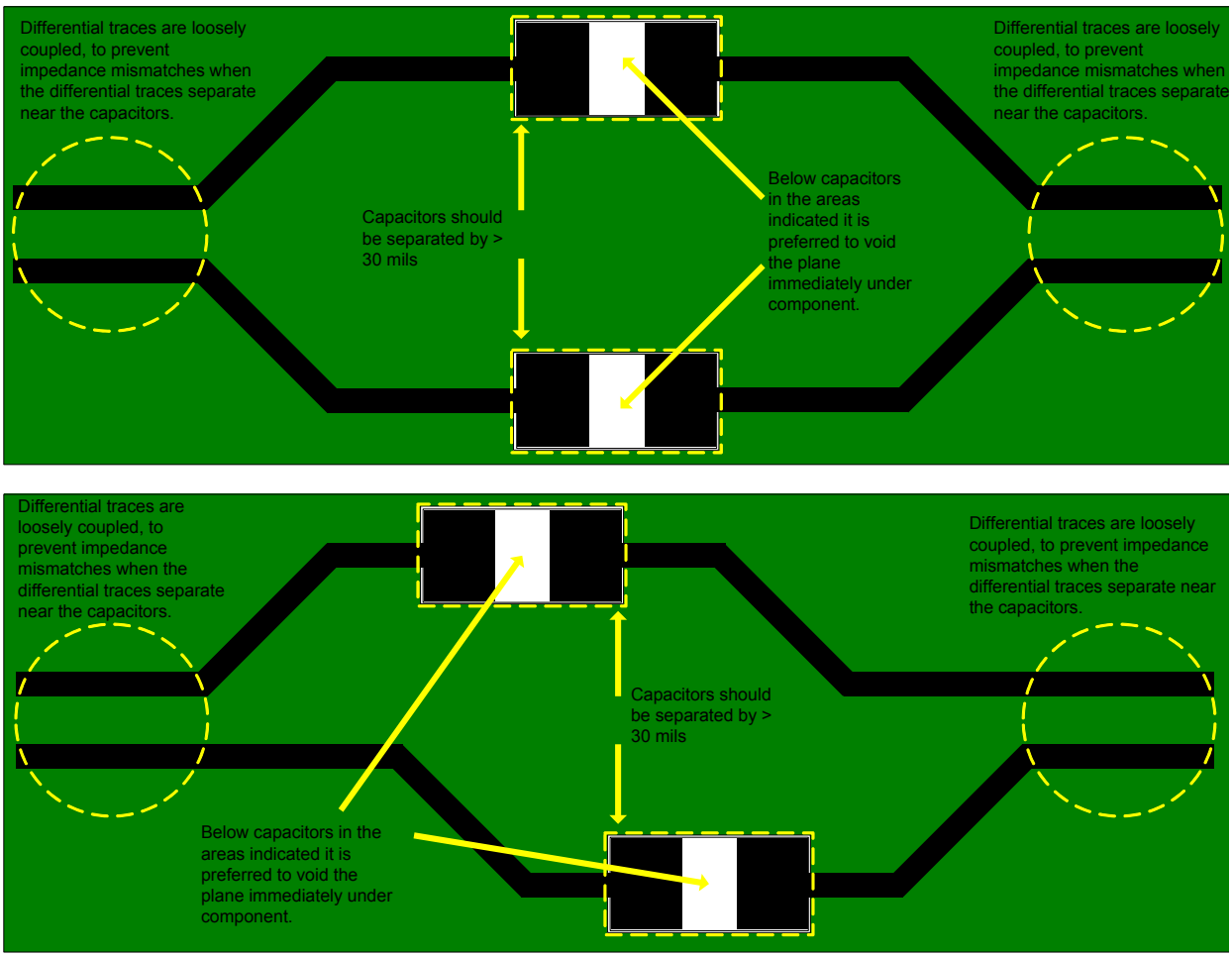


Figure 12-1 Reference Plane Voids Under AC Caps and AC Caps Separated to Reduce Stray Capacitance Between Caps

Note: The top layout of Figure 12-1 shows acceptable guidelines. The bottom layout of Figure 12-1 shows the preferred guidelines.

Use the smallest possible vias on board to optimize the impedance for the MAUI interface.

12.2.6.1 Via Usage

Use vias to optimize signal integrity. Figure 12-2 shows correct via usage. Figure 12-3 shows the type of topology that should be avoided.

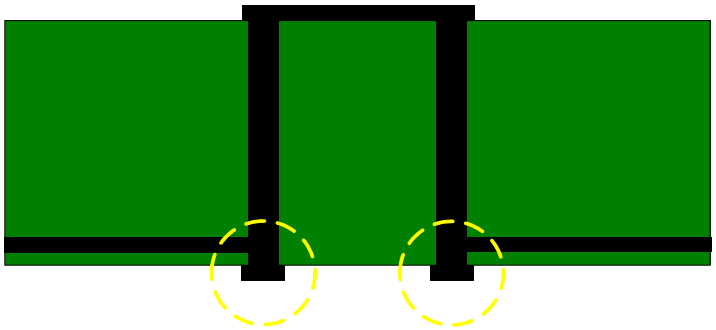


Figure 12-2 Correct Via Usage

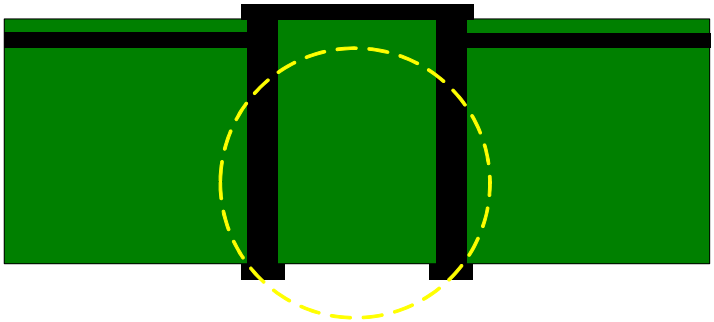


Figure 12-3 Incorrect Via Usage

Any via stubs on the KR and/or SFI+ differential signal traces must be less than 35 mils in length. Keeping KR and SFI+ signal via stubs less than or equal to 20 mils is preferable.

Note: Vias on SFI+ traces are not recommended. SFI+ Tx signals must not have any vias. Refer to the SFI+ layout section for more information.

Place ground vias adjacent to signal vias used for the MAUI interface. Do NOT embed vias between the high-speed signals, but place them adjacent to the signal vias (see Figure 12-4). This helps to create a better ground path for the return current of the AC signals, which also helps address impedance mismatches and EMC performance.

We recommend that, in the breakout region between the via and the capacitor pad, you target a Z0 for the via to capacitor trace equal to 50 Ω. This minimizes impedance imbalance.

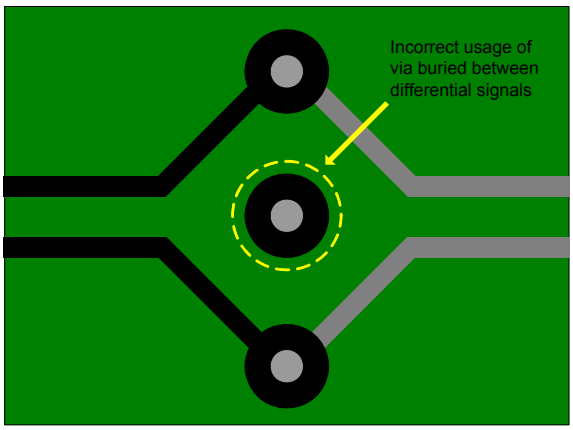


Figure 12-4 No Vias Between High-Speed Traces in the Same Differential Pair

12.2.7 Reference Planes

Do not cross plane splits with the MAUI high-speed differential signals. This causes impedance mismatches and negatively affects the return current paths for the board design and layout. Refer to Figure 12-5.

Traces should not cross power or ground plane splits if at all possible. Traces should stay seven times the dielectric height away from plane splits or voids. If traces must cross splits, capacitive coupling should be added to stitch the two planes together in order to provide a better AC return path for the high-speed signals. To be effective, the capacitors should be have low ESR and low equivalent series inductance.

Note: Even with plane split stitching capacitors, crossing plane splits is extremely high risk for 10 Gb/s KR and 10 Gb/s SFI+ designs.

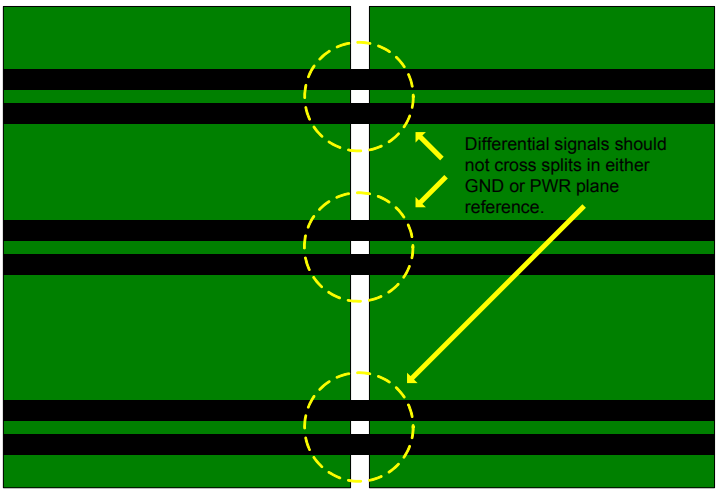


Figure 12-5 Do Not Cross Plane Splits

Keep Rx and Tx separate. This helps to minimize crosstalk effects since the TX and RX signals are NOT synchronous. This is the more natural routing method and occurs without much designer interference.

We recommend that the MAUI signals stay at least seven times the dielectric height away from any power or ground plane split (see Figure 12-6). This improves impedance balance and return current paths.

If a high-speed signal needs to reference a power plane, then ensure that the height of the secondary (power) reference plane is at least 3 x the height of the primary (ground) reference plane.

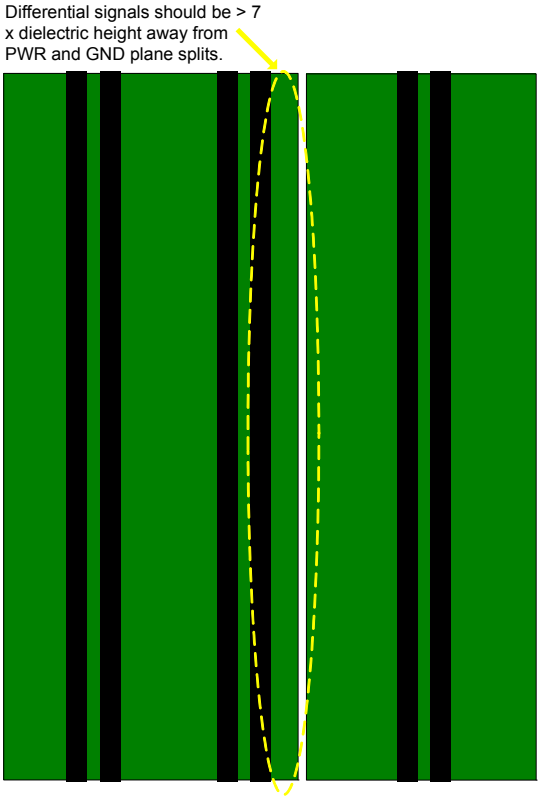
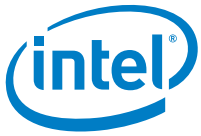


Figure 12-6 Traces Should Stay Seven Times the Dielectric Height Away From Plane Splits Or Voids

12.2.8 Dielectric Weave Compensation

Because the dielectric weave can cause different propagation velocity on each of the traces within one differential pair, Intel recommends using one or more trace routing techniques that can minimize signal skewing caused by the weave:

- Instead of routing traces parallel to either X or Y axis, traces should be routed at an angle to the weave, and the angle should be between 11 and 45 degrees. Routing both traces within each differential pair at an angle, with respect to the dielectric weave, minimizes the signal skew within each differential pair.
- The center-to-center pitch of the traces within the diff pairs can be matched to the weave pitch of the dielectric material. If you plan to uses a woven glass/epoxy dielectric material, check with the material supplier to find out the glass weave pitch prior to doing final differential trace routing.
- Traces can be routed to include a series of 45 degree bends, with bends separated by several tenths of an inch, to shift the traces in steps by a few millimeters each time. There should be an equal number left turns and right turns along the length of the traces. Trace segments between each pair of bends should be different lengths (if they are all the same length it could create an undesirable resonance in the line).



- If differential traces must be straight and orthogonal to the outline of the circuit board for most of their routed lengths, then rotate CAD artwork by 15°, with respect to the weave of the circuit board's dielectric weave.

12.2.9 Impedance Discontinuities

Impedance discontinuities cause unwanted signal reflections. Minimize vias (signal through holes) and other transmission line irregularities. A total of six through holes (a combination of vias and connector through holes) between the two chips connected by the MAUI interface is a reasonable maximum budget for each differential signal path. For example, if a backplane system has a total of three boards (blade server, mid-plane, and switch blade) in the differential signal path, then SFI+ Tx must not have any signal vias and SFI+Rx should not have more than one signal via per SFI+ signal trace. For this purpose, signal pin through-holes for board connectors are also counted as signal vias. Signal via pads on unconnected plane layers can be removed to reduce capacitance between the signal via and the surrounding metal plane. Alternatively, the anti-pad diameter can be increased to provide 9to 12 mils clearance between signal via (pads) and power or gound.

12.2.10 Reducing Circuit Inductance

Traces should be routed over a continuous reference plane with no interruptions. If there are vacant areas on a reference or power plane, the signal conductors should not cross the vacant area. Routing over a void in the reference plane causes impedance mismatches and usually increases radiated noise levels. Noisy logic grounds should NOT be located near or under high-speed signals or near sensitive analog pin regions of the LAN silicon. If a noisy ground area must be near these sensitive signals or IC pins, ensure sufficient decoupling and bulk capacitance in these areas. Noisy logic and switching power supply grounds can sometimes affect sensitive DC subsystems such as analog to digital conversion, operational amplifiers, etc.

All ground vias should be connected to every ground plane; and similarly, every power via should be connected to all equally potential power planes. This helps reduce circuit inductance. Another recommendation is to physically locate grounds to minimize the loop area between a signal path and its return path. Rise and fall times should be as slow as possible while still meeting the relevant electrical requirements. Because signals with fast rise and fall times contain many high frequency harmonics, which can radiate significantly. The most sensitive signal returns closest to the chassis ground should be connected together. This results in a smaller loop area and reduces the likelihood of crosstalk. The effect of different configurations on the amount of crosstalk can be studied using electronics modeling and simulation software.

12.2.11 Signal Isolation

To maintain best signal integrity, keep digital signals far away from the analog traces. A good rule of thumb is no digital signal should be within 7x to 10x dielectric height of the differential pairs. If digital signals on other board layers cannot be separated by a ground plane, they should be routed at a right angle (90 degrees) to the differential signal traces. If there is another LAN controller on the board, take care to keep the differential pairs from that circuit away. The same thing applies to switching regulator traces.

Rules to follow for signal isolation:

- Separate and group signals by function on separate board layers if possible. Maintain a separation that is at least seven times the thinnest adjacent dielectric height between all differential pairs (Ethernet) and other nets, but group associated differential pairs together. For example, Keep Tx signals with Tx signals and keep Rx signals with Rx signals. Note that if an Rx signal is routed between two Tx signals, the higher levels of Tx crosstalk causes the Rx signal-to-noise ratio to be worse than if the Rx signal is routed between two other Rx signals.)
- Over the length of the trace run, each differential pair should be at least seven times the thinnest adjacent dielectric height away from any parallel signal traces.
- Physically group together all components associated with one clock trace to reduce trace length and radiation.
- Isolate other I/O signals from high-speed signals to minimize crosstalk because crosstalk can increase radiated EMI and can increase susceptibility to EMI from other signals.
- Avoid routing high-speed LAN traces near other high-frequency signals associated with a video controller, cache controller, processor, or other similar devices.

12.2.12 Power and Ground Planes

Good grounding requires minimizing inductance levels in the interconnections and keeping ground returns short, signal loop areas small, and locating decoupling capacitors at or near power inputs to bypass to the signal return. This will significantly reduce EMI radiation.

These guidelines reduce circuit inductance in both backplanes and motherboards:

- Route traces over a continuous plane with no interruptions. Do not route over a split power or ground plane. If there are vacant areas on a ground or power plane, avoid routing signals over the vacant area. Routing signals over power or ground voids increases inductance and increases radiated EMI levels.
- Use distance and/or extra decoupling capacitors to separate noisy digital grounds from analog grounds to reduce coupling. Noisy digital grounds may affect sensitive DC subsystems.
- All ground vias should be connected to every ground plane; and every power via should be connected to all power planes at equal potential. This helps reduce circuit inductance.
- Physically locate grounds between a signal path and its return. This will minimize the loop area.

- Avoid fast rise/fall times as much as possible. Signals with fast rise and fall times contain many high frequency harmonics, which can radiate EMI.
- Do not route high-speed signals near switching regulator circuits.
- There should not be any test-point vias or test-point pads on KR and SFI+ traces.
- It's acceptable to put ground fill or thieving on the trace layers, but preferably not closer than 50 mils to the differential traces and the connector pins.
- If differential traces must be routed on another layer, then the signal vias should carry the signal to the opposite side of the circuit board (to be near the top of the circuit board); AND if the high-speed signals are being routed between two connectors on the same board, then before the signal traces reach the second connector, they must return to the original signal layer (before reaching the connector pin). This strategy keeps via stubs short without requiring back drilling.
- Each time differential traces make a layer transition (pass through a pair of signal vias), there must be at least one ground via located near each signal via. Two ground vias near each signal via is better. See Figure 12-7 and Figure 12-8.

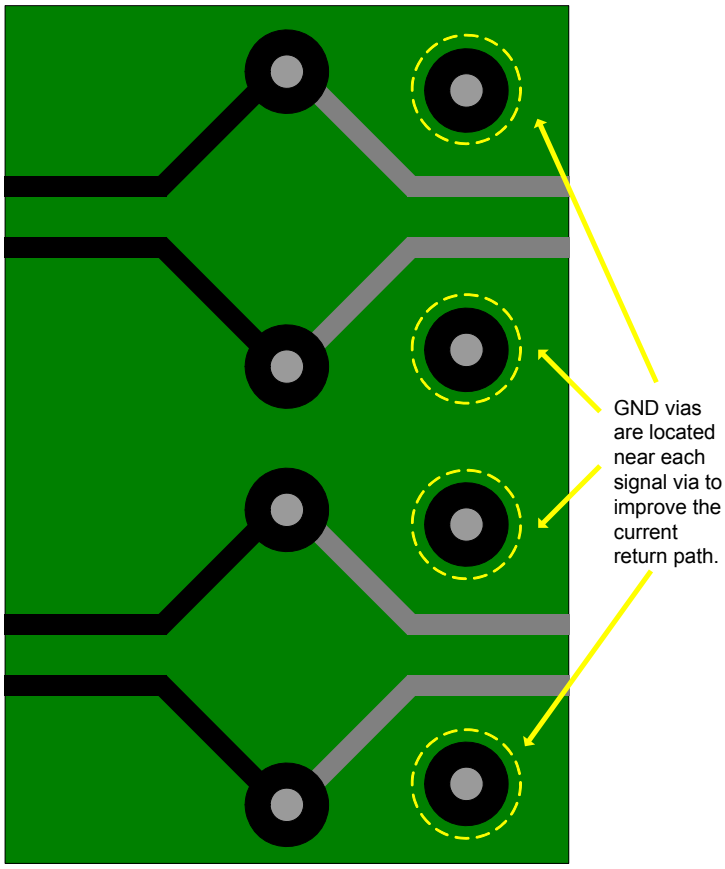


Figure 12-7 Good Ground Vias for Signal Return Paths – One Return Path Via Per Signal Via

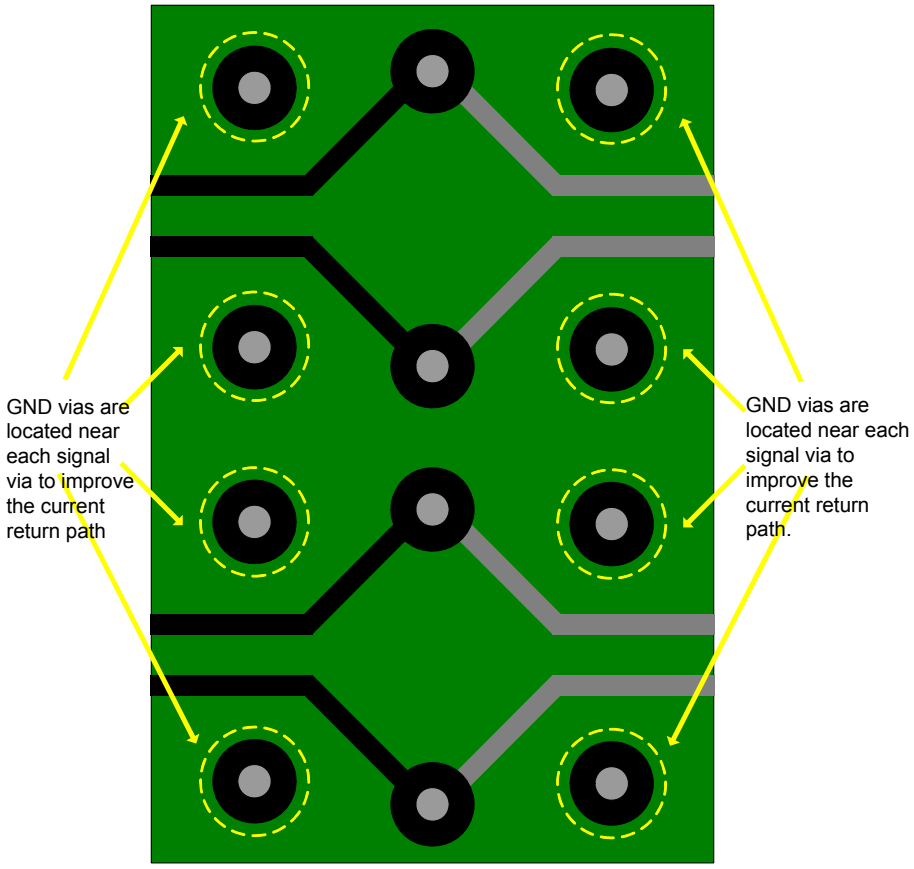


Figure 12-8 Better Ground Vias for Signal Return Paths – Two Return Path Vias Per Signal Via (Less Reflection)

If the circuit board fabrication process permits it, it is best to remove signal via pads on unconnected metal layers. See Figure 12-9 and Figure 12-10.