

82577 Schematic Checklist (Version 2.1)

Project Name				
Fab Revision				
Date				
Designer				
Intel Contact				
Reviewer				
SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
	Completed by:	Design Engineer Name:		
General	Obtain the most recent documentation and specification updates.	Documents are subject to frequent change.		
	Use this checklist in conjunction with the Customer Reference Board (CRB) reference schematics.			
	Observe instructions for special pins needing pull-up or pull-down resistors.			
	Refer to the <i>Intel® 5 Series Family Platform Design Guide (PDG)</i> , the <i>Intel® 5 Series Express Chipset EDS</i> , and Specification Update for integrated GbE LAN (MAC) details.			

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
Support Pins	Connect TEST_EN (pin 30) through a 1 K Ω pull-down resistor.			
	Connect LAN_DISABLE_N (pin 3) to a 10 K Ω pull-up resistor to 3.3 Vdc and a 10 K Ω no-stuff pull-down resistor to ground. Connect to Intel® 5 Series Express Chipset LAN_PHY_PWR_CTRL (pin K9) through a 0 Ω resistor. For the SFF Intel® 5 Series Express Chipset ballout, connect to Intel® 5 Series Express Chipset LAN_PHY_PWR_CTRL (pin F6) through a 0 Ω resistor.			
	Connect RBIAS (pin 12) to a 3.01 K Ω 1% pull-down resistor.			
	Connect RSVD_VCC3P3 (pin 1) and RSVD_VCC3P3 (pin 2) to a 3.01 K Ω 5% pull-up resistor to 3.3 Vdc.	A value of 3.01 K Ω is suggested. A range of 3.01 K Ω to 10 K Ω can also be used.		
	Connect the Epad (pin 49) to ground plane.	Refer to the <i>Intel® 5 Series Family PDG</i> for more information.		

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
PCIe Interface	Connect PCIe interface pins to corresponding pins on the Intel® 5 Series Express Chipset.	Recommend that the default connection be lane six of the Intel® 5 Series Express Chipset. However, the PCIe interface pins can be connected to any available PCIe port.		
	Connect PETn (pin 39) and PETp (pin 38) to Intel® 5 Series Express Chipset PERn6 (pin BA34) and PERp6 (pin AW34), respectively. For the SFF Intel® 5 Series Express Chipset ballout, connect PETn (pin 39) and PETp (pin 38) to Intel® 5 Series Express Chipset PERn6 (pin AU33) and PERp6 (pin AR33), respectively.			
	Connect PERn (pin 42) and PERp (pin 41) to Intel® 5 Series Express Chipset PETn6 (pin BC34) and PETp6 (pin BD34), respectively. For the SFF Intel® 5 Series Express Chipset ballout, connect PERn (pin 42) and PERp (pin 41) to Intel® 5 Series Express Chipset PETn6 (pin BA31) and PETp6 (pin AW31), respectively.			
	Place AC coupling capacitors (0.1 µF) near the PCIe transmitter.	Size 0402, X7R is recommended.		

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
PCIe Interface	<p>Connect PE_CLKn (pin 45) and PE_CLKp (pin 44) to Intel® 5 Series Express Chipset CLKOUT_PCIE5N (pin AJ50) and CLKOUT_PCIE5P (pin AJ52), respectively. For the SFF Intel® 5 Series Express Chipset ballout, connect PE_CLKn (pin 45) and PE_CLKp (pin 44) to Intel® 5 Series Express Chipset CLKOUT_PCIE5N (pin AL49) and CLKOUT_PCIE5P (pin AL51), respectively.</p>	<p>The PCIe clock buffer can be connected to any clock port. It does not have to be port 5. This connection should match the CLK_REQ_N port number.</p>		
	<p>Connect CLK_REQ_N (pin 48) through a 10 KΩ pull-up resistor to 3.3 Vdc and then through a series 0 Ω resistor to Intel® 5 Series Express Chipset PCIECLKRQ5# (pin H6). For the SFF Intel® 5 Series Express Chipset ballout, connect CLK_REQ_N (pin 48) through a 10 KΩ pull-up resistor to 3.3 Vdc and then add an empty series 0 Ω resistor to Intel® 5 Series Express Chipset PCIECLKRQ5# (pin G3).</p>	<p>CLK_REQ_N can be connected to one of the eight Intel® 5 Series Express Chipset inputs (PCIECLKRQ[7:0]#). This connection should match the PCIe clock output port number.</p> <p>If connecting this PHY output to the PCIECLKRQ1# or PCIECLKRQ2# pins (powered by the core well), the pull-up resistor needs to be connected to the +V3.3S rail. If connecting to any of the other PCIECLKRQ# pins (powered by Sus well), the pull-up resistor needs to be connected to the +V3.3A rail.</p>		

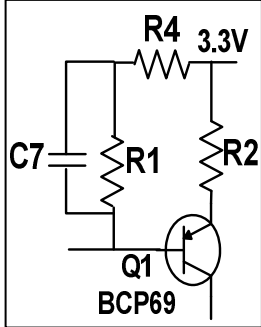
SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
Pcie Interface		<p>When the PHY is powered down, connecting to the +V3.3A rail results in a small amount of current leakage (~300 μA) through the PHY CLK_REQ_N buffer. This results in a small residual voltage (~0.6 Vdc) on the PHY +V3.3 power rail. The power impact is less than 1 mW and does not affect PHY functionality or reliability. Designers can avoid this leakage by connecting the PHY CLK_REQ_N output to the PCIECLKREQ#1 or PCIECLKREQ#2 pins or by adding a blocking FET.</p> <p>Note: Additional guidance is available in the Intel® 5 Series Family PDG.</p>		
	<p>Connect PE_RST_N (pin 36) through a series 0 Ω resistor to Intel® 5 Series Express Chipset PLTRST# (pin D5). For the SFF Intel® 5 Series Express Chipset ballout, connect PE_RST_N (pin 36) through a series 0 Ω resistor to Intel® 5 Series Express Chipset PLTRST# (pin D4).</p>			

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
SMBus	Connect pull-up resistors to SMB_CLK (pin 28) and SMB_DATA (pin 31).	Use 2.2 K Ω 5% pull-up resistors. Ensure pull-ups are connected to the power rail that is present while in Sx states and while transitioning from G3 to S5.		
		Note that the PHY SMBus address is 0xC8 and default MAC SMBus address is 0xE0.		
	Connect SMB_CLK and SMB_DATA to Intel® 5 Series Express Chipset SMLOCLK (pin C6) and SMLODATA (pin G8), respectively. For the Intel® 5 Series Express Chipset SFF ballout, connect SMB_CLK and SMB_DATA to Intel® 5 Series Express Chipset SMLOCLK (pin B8) and SMLODATA (pin J9), respectively.	The Intel® 5 Series Express Chipset has a dedicated SMBus for the PHY (SMBus channel 0). No other device (such as an external BMC) can be connected to SMLOCLK or SMLODATA.		
Clock Source (Crystal Option)	Use a 25 MHz 30 ppm accuracy @ 25 °C crystal. Avoid components that introduce jitter.	Parallel resonant crystals are required. The calibration load should be 18 pF. Specify Equivalent Series Resistance (ESR) to be 50 Ω or less. Avoid PLL clock buffers.		

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
Clock Source (Crystal Option)	Connect two load capacitors to the crystal; one on XTAL_OUT (pin 9) and one on XTAL_IN (pin 10). Use 33 pF capacitors as a starting point, but be prepared to change the value based on testing.	Capacitance affects accuracy of the frequency and must be matched to crystal specifications, including estimated trace capacitance in calculation. Use capacitors with low ESR (types COG or NPO, for example). Refer to the Datasheet and the Intel Ethernet Controllers Timing Device Selection Guide for more information.		
	Connect a series 10 pF capacitor to XTAL_OUT (pin 9).	Refer to the <i>Intel® 82577 Specification Update</i> for more details.		
	Connect XTAL_IN (pin 10) and XTAL_OUT (pin 9) to the appropriate crystal pins.			

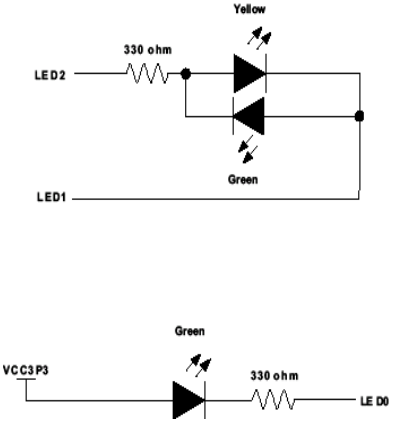
SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
Clock Source (Oscillator Option)	Connect the output of the clock oscillator to pin XTAL_IN (pin 10) through appropriate signal conditioning (capacitance-coupled voltage divider). Do not connect XTAL_OUT (pin 9).	Use a 25 MHz 50 ppm oscillator. The oscillator needs to maintain 50 ppm under all applicable temperature and voltage conditions. Avoid PLL clock buffers. Place voltage divider near the XTAL_IN of the PHY (less than 325 mils). Refer to the <i>Intel® 5 Series Family PDG</i> for details about the signal conditioning circuit.		
	Use a local decoupling capacitor on the oscillator power supply. Include a bulk 1 μ f capacitor as well as a high frequency 0.1 μ f decoupling capacitor.	If isolated with a ferrite bead, include a bulk decoupling capacitor next to the oscillator. Refer to the <i>Intel® 5 Series Family PDG</i> for more details.		
	The signal from the oscillator must be AC coupled into the 82577.	The 82577 has internal circuitry to set the input common mode voltage.		

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
Clock Source (Oscillator Option)	The amplitude of the clock signal going into the 82577 should be within the maximum input clock amplitude specification for the 82577.	The clock oscillator has an internal voltage regulator to isolate it from the external noise of other circuits to minimize jitter. For example, if a 3.3 Vdc oscillator is used, it's output signal should be attenuated to a maximum value with a resistive divider circuit. Refer to the <i>Intel® 5 Series Family PDG</i> for more details.		
	If the oscillator driver is further away than two inches then add a 33 Ω series resistor directly at the output.			
3.3 and 1.0 Vdc Power (Without BCP69 Circuitry)	Provide a 3.3 Vdc supply. Use the power rail that is present while in Sx states and while transitioning from G3 to S5.	This is necessary to support wake up from power down states.		
	Connect VDD1P0 to the Intel® 5 Series Express Chipset 1.05 Vdc Switching Voltage Regulator (SVR).	Use a 0 Ω 0805 resistor to allow isolation of the Intel® 5 Series Express Chipset 1.05 Vdc supply instead of the BCP69 circuitry.		
	CTRL1P0 (pin 7) can be left as no connect if using the Intel® 5 Series Express Chipset Switched Voltage Regulator (SVR) sharing configuration.			

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
3.3 and 1.0 Vdc Power (With BCP69 Circuitry)	Provide a 3.3 Vdc supply. Use the power rail that is present while in Sx states and while transitioning from G3 to S5.	This is necessary to support wake up from power down states.		
	<p>Leave empty sites for R1 (4.99 KΩ), R4 (0 Ω), R2 (0 Ω) and C7 (0.01 μF), as shown. Refer to the CRB reference schematics for more BCP69 connection details.</p> <p>Note: Specific CRBs might have different designations for R1, R4, and C7. The designations shown are for reference only.</p> <p>Place decoupling and bulk capacitors close to the 82577, using short, wide traces and large vias. If power is distributed on traces, bulk capacitors should be used at both ends. If power is distributed on cards, bulk capacitors should be used at the connector.</p>			
Magnetics	Qualify magnetic modules carefully for return loss, insertion loss, open circuit inductance, common mode rejection, and crosstalk isolation.	Magnetics module is critical to passing IEEE PHY conformance tests and EMI test. Refer to the 82577 Datasheet for magnetics requirements.		

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
Magnetics	<p>VCT (Pin 6) can be left as a no connect (on the PHY side). On the magnetics center tap, use 0.1 μF bypass capacitors. Connect to ground through a 1 μF bulk decoupling capacitor placed near the magnetics center tap input to the transformer.</p>	<p>Ceramic capacitors with low ESR should be used.</p> <p>Note: Some magjacks have local decoupling (0.1 μF) integrated into the part.</p>		
	<p>Bob Smith termination:</p> <p>If the RJ-45 connector does not have integrated Bob Smith termination, use 4 x 75 Ω resistors for cable-side center taps and unused pins.</p> <p>Use a high-voltage isolation capacitor attached to the termination plane. A suggested value is 1500 pF/3 KV.</p>	<p>Terminate pair-to-pair common mode impedance of the CAT5 cable. If the discrete magnetics already has Bob Smith termination, then there is no need to add.</p> <p>Note: Some magjacks have Bob Smith termination integrated into the part.</p> <p>Name the common connection between the 75 Ω resistors and high-voltage isolation capacitor so it can be used as a reference plane.</p>		
MDI Termination	<p>The 82577 does not require any MDI termination.</p>			

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
Chassis Ground (10/100/1000Base-T interface)	If possible, provide a separate chassis ground to connect the shroud of the RJ-45 connector and to terminate the line side of the magnetic module.	This design improves EMI behavior. Also, if using integrated magnetics with USB, do not isolate ground for RJ45.		
Chassis Ground (10/100/1000Base-T interface)	Place pads for approximately 4-6 "stitching" capacitors to bridge the gap from chassis ground to signal ground.	Typical values range from 0.1 μ F to 4.7 μ F. The correct value should be determined experimentally.		
Decoupling	Use decoupling capacitors generously. The minimum acceptable decoupling by the power supply is identified in the remarks field (not including decoupling at magnetic):	Place near 3.3 Vdc LAN pin: one 10 μ F and one 0.1 μ F capacitor.		
		Place near 1.0 Vdc LAN pin: one 10 μ F and one 0.1 μ F capacitor.		
		Place near VDD3P3_OUT and VDD3P3 pins: one 1 μ F capacitor.		

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
LED Circuits	<p>Basic recommendation is a single green LED for Activity and a dual (bi-color) LED for Speed. Many other configurations are possible. LEDs are configurable through the NVM.</p>	<p>Two LED configuration is compatible with integrated magnetic modules. For the Link/Activity (green) LED, connect the cathode to the LED0 pin and the anode to VCC. For the bi-color speed LED pair, have the LED2 signal drive one end. The other end should be connected to LED1. When LED2 is low, the 100 Mb/s (green) LED is lit. When LED1 is low the 1000 Mb/s (yellow) LED is lit.</p> 		

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
LED Circuits	Connect LEDs to 3.3 Vdc as indicated in reference schematics.	Use the power rail for designs supporting wake-up (present in Sx states and G3 to S5 transition). Consider adding one or two filtering capacitors per LED for extremely noisy situations. Suggested starting value is 470 pF.		
	Add current limiting resistors to LED paths.	Typical current limiting resistors are 250 Ω to 330 Ω when using a 3.3 Vdc supply. Current limiting resistors are typically included with integrated magnetic modules.		

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
Miscellaneous	Intel® 5 Series Express Chipset output SLP_LAN#, pin F6 (pin E5 SFF ballout), can be used to gate power rails that do not need to be on when host WoL and manageability hardware are disabled.	<p>Specific configurations that leave power on/off depending on WoL and manageability hardware settings are design dependent.</p> <p>Refer to the <i>Intel® 5 Series Express Chipset EDS</i> for more details.</p> <p>SLP_LAN# replaces the glue logic in previous designs that was an OR FUNCTION between WOL_EN and SLP_M#.</p> <p>For Intel® 5 Series Family, this glue logic is not required. Design a circuit in such a way that when SLP_LAN# is low, cut power to the PHY. When SLP_LAN# is high, PHY power is on.</p>		
	Intel® 5 Series Express Chipset LAN_RST#, pin A10 (pin C11 SFF ballout), timing requirement needs to be met.	Refer to the <i>Intel® 5 Series Express Chipset EDS</i> for more details on LAN_RST# timing requirements.		
	Leave an empty site for a TVS device between the RJ-45 and the magnetics assembly.	TVS diodes can be added to the MDI for additional ESD protection.		

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
Miscellaneous	For designs that support docking stations, verify the connections for the center tap and MDI.	If a LAN switch is used, verify the connections to and from the LAN switch.		
	Use a LAN switch to reduce stub length on the MDI interface between the local RJ-45 and the docking station.	Stubs on the MDI interface in systems with a local RJ-45 and a docking connector causes IEEE conformance test failures. The LAN switch disconnects the unused interface, which improves IEEE performance.		
Mfg Test	The 82577 allows a JTAG Test Access Port.	The 82577 does not support BSDL. The 82577 supports XOR for manufacturing line tests.		
	Add empty 10 K Ω pull-up resistors for JTAG_TCK (pin 35) and JTAG_TMS (pin 33).	The JTAG pins can be pulled up to 3.3 Vdc supply.		
	Route JTAG to test points.			

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
<p>INFORMATION IN THIS DOCUMENT IS PROVIDED IN CONNECTION WITH INTEL® PRODUCTS. NO LICENSE, EXPRESS OR IMPLIED, BY ESTOPPEL OR OTHERWISE, TO ANY INTELLECTUAL PROPERTY RIGHTS IS GRANTED BY THIS DOCUMENT. EXCEPT AS PROVIDED IN INTEL'S TERMS AND CONDITIONS OF SALE FOR SUCH PRODUCTS, INTEL ASSUMES NO LIABILITY WHATSOEVER, AND INTEL DISCLAIMS ANY EXPRESS OR IMPLIED WARRANTY, RELATING TO SALE AND/OR USE OF INTEL PRODUCTS INCLUDING LIABILITY OR WARRANTIES RELATING TO FITNESS FOR A PARTICULAR PURPOSE, MERCHANTABILITY, OR INFRINGEMENT OF ANY PATENT, COPYRIGHT OR OTHER INTELLECTUAL PROPERTY RIGHT. Intel products are not intended for use in medical, life saving, life sustaining applications.</p> <p>Intel may make changes to specifications and product descriptions at any time, without notice.</p> <p>This document contains information on products in the design phase of development. The information here is subject to change without notice. Do not finalize a design with this information.</p> <p>The 82577 GbE Controller may contain design defects or errors known as errata which may cause the product to deviate from published specifications. Current characterized errata are available on request.</p> <p>*Other names and brands may be claimed as the property of others.</p> <p>Copyright © Intel Corporation 2004, 2005, 2006, 2007, 2008, 2009, 2010. INTEL CONFIDENTIAL</p>				

82577 Layout Checklist (Version 2.1)

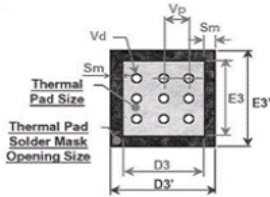
Project Name			
Fab Revision			
Date			
Designer			
Intel Contact			

<u>SECTION</u>	<u>CHECK ITEMS</u>	<u>REMARKS</u>	<u>DONE</u>	<u>COMMENTS</u>
	Completed by:	Design Engineer Name:		
General	Obtain the most recent documentation and specification updates.	Documents are subject to frequent change.		
	Route the transmit and receive differential traces before routing the digital traces.	Layout of differential traces is critical.		
	IMPORTANT: Recommend that all signals are routed in reference to a solid plane. If signals cross planes, stitching caps are required between the planes within 40 mils of the signal.			
	If the differential signals transition from a ground referenced layer to a power referenced layer, place a decoupling capacitor on the power and ground within 40 mils of the signal vias.			
	Obtain the most recent stack-up information from your Printed Circuit Board (PCB) vendor.			

<u>SECTION</u>	<u>CHECK ITEMS</u>	<u>REMARKS</u>	<u>DONE</u>	<u>COMMENTS</u>
General	Refer to the <i>Intel® 5 Series Family Platform Design Guide (PDG)</i> for detailed routing requirements.			
Placement of the 82577	Place the 82577 at least one inch from the edge of the board.	With closer spacing, fields can follow the surface of the magnetics module or wrap past edge of the board. As a result, EMI might increase.		
	Place the 82577 at least 0.3 inches from the LAN switch or one inch from the magnetics (if a LAN switch is not used).	Placing the 82577 closer than one inch to the magnetics or 0.3 inches to the LAN switch likely results in failed EMI and common mode noise.		
	Place the LAN switch at least one inch from the magnetics.			
	Place the RBIAS compensation resistor less than one inch from the silicon.			
Clock Source (Crystal Option)	Place the crystal less than 0.75 inches from the 82577.	This reduces EMI.		
	Place the crystal load capacitors within 0.09 inches of the crystal.			
	Keep clock lines at least five times the dielectric away from other digital traces (especially reset signals), I/O ports, board edge, transformers and differential pairs.	This helps reduce EMI. Do not route the crystal traces as differential pairs		
Clock Source (Oscillator Option)	Make sure that the oscillator has it's own local power supply decoupling capacitors. Make sure the decoupling capacitors are within 0.25 inches of the oscillator.	Helps oscillator stability.		

<u>SECTION</u>	<u>CHECK ITEMS</u>	<u>REMARKS</u>	<u>DONE</u>	<u>COMMENTS</u>
Clock Source (Oscillator Option)	If the oscillator is shared or is more than two inches away from the 82577, a back-termination resistor should be placed near the oscillator. Note that it is better to not share this clock source. If it is shared keep the traces for the two clock lines matched in length.	This enables tuning to ensure that reflections do not distort the clock waveform. Matching shared clock lines keeps the reflections from one spur of the clock from affecting the other.		
	Keep clock lines at least five times the dielectric thickness away from other digital traces (especially reset signals), I/O ports, board edge, transformers and differential pairs.	This helps reduce EMI.		
	Use a 50 Ω transmission line for the trace between the oscillator and the 82577.	This is a single-ended transmission line. Make sure the transmission line does not cross any plane cuts. Also use stitching capacitors as needed when transferring from one reference plane to another.		
	If the oscillator is powered through a ferrite bead, make sure that the local decoupling capacitor is connected close to oscillator package power input pin. Use a pore for the isolated power input.	A local power well provides a way to isolate system noise from the main time base of the Ethernet. This improves jitter and Bit Error Rate (BER) results.		
General Power Supply Guidance	Use planes to deliver 3.3 Vdc and 1.0 Vdc.	Narrow finger-like planes and very wide traces are allowed. If traces are used, 100 mils is the minimum.		

<u>SECTION</u>	<u>CHECK ITEMS</u>	<u>REMARKS</u>	<u>DONE</u>	<u>COMMENTS</u>
General Power Supply Guidance	<p>Place decoupling and bulk capacitors close to 82577, with some along every side, using short, wide traces and large vias.</p> <p>Place these to minimize the inductance from each power pin to the nearest decoupling capacitor.</p> <p>The 3.3 Vdc and 1.0 Vdc rails should have decoupling capacitors placed near the 82577.</p>	If power is distributed on traces, bulk capacitors should be used at both ends. If power is distributed on cards, bulk capacitors should be used at the connector.		
1.0 Vdc Power (Shared With Intel® 5 Series Express Chipset SVR)	Connect VDD1P0 to the Intel® 5 Series Express Chipset 1.05 Vdc Switching Voltage Regulator (SVR).	The copper feed from the 1.05 Vdc SVR to the 82577 must provide enough current handling capability in order to meet the 82577 1.0 Vdc input requirements. Refer to the Power Loss Calculator for further guidance.		
1.0 Vdc Power (With BCP69 Circuitry)	If using the internal regulator control circuit of the 82577 with an external PNP transistor, keep the trace length from the CTRL1P0 output ball to the transistor very short (less one inch) and use 15 mil (minimum) wide trace.	A low inductive loop should be kept from the regulator control pin, through the PNP transistor, and back to the 82577 from the transistor's collector output. The power pins should connect to the collector of the transistor through a power plane to reduce the inductive path. This reduces oscillation and ripple in the power supply.		
	The 1.0 Vdc regulating circuits require 1/2 inch x 1/2 inch thermal relief pads for each PNP.	The pads should be placed on the top layer, under the PNP. Refer to BCP69 vendor documentation for additional guidance.		

SECTION	CHECK ITEMS	REMARKS	DONE	COMMENTS
Epad	Use appropriate number of vias to connect Epad to ground.	<p>Refer to the <i>Intel® 5 Series Family PDG</i> for additional details.</p> <p>Note that the Epad connection is the only source for ground and is used for both electrical and thermal.</p>  <p>Thermal pad size = $D3 * E3 = D2 * E2$. Solder mask opening size = $D3' * E3'$ = $(D3 + 2 * Sm) * (E3 + 2 * Sm)$ = $(D2 + 0.1) * E2 + 0.1$ where $Sm = 0.05$ mm (machine capability) $D2.E2$ are exposed die attach pad size</p> <p>Thermal via diameter Vd (circled hole) $Vd = 0.3 - 0.33$ mm (general drilling machine capability) where thermal via is a Plated Through Hole (PTH) and plugged in solder mask from top-side of PCB.</p> <p>The thermal via pitch Vp $Vp = 1.0 - 1.25$ mm</p>		
PCIe-Based Interface	Place the AC coupling capacitors on the PCI-based Tx traces as close as possible to the 82577 but not further than 250 mils.	Size 0402, X7R is recommended. The AC coupling capacitors should be placed near the transmitter for the PCIe-based interface.		
	Make sure the trace impedance for the PCIe-based differential pairs is 85 W +/- 15%.	Simulation shows 85 W differential trace impedances meets Intel's PCIe-based minimum receive eye requirements when using the Customer Reference Board (CRB) design stack up.		

<u>SECTION</u>	<u>CHECK ITEMS</u>	<u>REMARKS</u>	<u>DONE</u>	<u>COMMENTS</u>
PCIe-Based Interface		When using the CRB design stack up, Intel recommends that board designers use a 85 W differential trace impedance for PCIe-based I/O with the expectation that the center of the impedance is always targeted at 85 W. The ±15% tolerance is provided to allow for board manufacturing process variations and not lower target impedances.		
	Match trace lengths within each PCIe-based pair on a segment-by-segment basis. Match trace lengths within a pair to five mils.	A PCIe segment is defined as any trace within the same layer. For example, transitioning from one layer to another through a via is considered as two separate PCIe segments. The differential pairs within each segment needs to be matched to 5 mils.		
	Use a minimum of 15 mil (0.381 mm) pair-to-pair spacing for board designs that use approximately 3 mil dielectric thickness in the design stack up.	Pair-to-pair spacing is five times the thickness of the material.		
	Make sure the PCIe clock differential pair is routed using 100 Ω characteristic impedance. Keep a spacing of three times the dielectric thickness between the PCIe clock and any other traces (digital, analog, and power).			
	If any of the PCIe signals cross a plane or transfers from one plane to another, add stitching capacitors to provide a return path for the common mode currents.	This reduces EMI.		
SMBus	Connect the SMBus using little or no stubs at the pull-up resistor.			

<u>SECTION</u>	<u>CHECK ITEMS</u>	<u>REMARKS</u>	<u>DONE</u>	<u>COMMENTS</u>
MDI Interface	Design traces for 100 Ω differential impedance (\pm 15%). Refer to the <i>Intel® 5 Series Family PDG</i> for spacing guidance between any signal and the MDI channels.	Primary requirement for 10/100/1000 Mb/s Ethernet. Paired 50 Ω traces do not make 100 Ω differential. An impedance calculator can be used to verify this.		
		Please use the 82577 Trace Length Calculator available from Intel. Trace width and separation of the MDI pairs is determined by the board stack up in order to achieve the correct impedance. For applications that require a longer MDI trace lengths, Intel recommends that thicker dielectric or lower Er materials be used. This permits higher differential trace impedance and wider, lower loss traces.		
		Target differential impedance is 100 Ω +/- 15%. If a stack up does not allow for 100 Ω +/- 15% differential trace impedance, consider trying a stack up that allows 95 Ω +/- 10%.		
		Stripline is NOT recommended due to thinner more resistive signal layers.		
	Keep trace length under 11 inches from the 82577 through the magnetics to the RJ-45 connector.	For applications that require a longer MDI trace length of more than 11 inches, Intel recommends that thicker dielectric or lower Er materials be used. This permits higher differential trace impedance and wider, lower loss traces. Signal attenuation can cause problems for traces longer than 11 inches.		
	Do not make 90° bends	Bevel corners with turns based on 45° angles.		
Minimize through holes (vias).	If using through holes (vias), the budget is two 10 mil finished hole size vias per trace.			

<u>SECTION</u>	<u>CHECK ITEMS</u>	<u>REMARKS</u>	<u>DONE</u>	<u>COMMENTS</u>
MDI Interface	Keep trace-to-trace length difference between each element no more than 10 mils (elements include any active or passive components).	Refer to the <i>Intel® 5 Series Family PDG</i> for additional routing and placement guidelines. An MDI segment is defined as any trace within the same layer. For example, transitioning from one layer to another through a via is considered as two separate MDI segments. The differential pairs within each segment needs to be matched to 10 mils.		
	Keep trace-to-trace length difference within 30 mils of overall length.			
	Pair-to-pair trace length does not have to be matched as differences are not critical.	The difference between the length of longest pair and the length of the shortest pair should be kept below two inches.		
	Keep differential pairs more than seven times the dielectric thickness away from each other and other traces, including NVM traces and parallel digital traces.	This minimizes crosstalk and noise injection. Tighter spacing is allowed for the first 200 mils of trace near of the components.		
	Ensure that the MDI traces (including Bob Smith termination) between the magnetics and the RJ 45 connector are at least 80 mils from all other traces.	This is to ensure the system can survive a high voltage on the MDI cable. (Hi-POT) Refer to the <i>Intel® 5 Series Family PDG</i> for routing and placement guidelines.		
	Keep traces at least 0.1 inches away from the board edge.	This reduces EMI.		
	Do not have stubs along the traces.	Stubs cause discontinuities.		

<u>SECTION</u>	<u>CHECK ITEMS</u>	<u>REMARKS</u>	<u>DONE</u>	<u>COMMENTS</u>
MDI Interface	Route traces on appropriate layers.	<p>Run pairs on outer layers to improve routing. Make sure the adjacent layer is a ground layer. There must be no splits in the GND planes under the MDI channel.</p> <p>When differential signals transition from one board layer to another, place ground vias within 40 mils of the signal vias.</p> <p>Avoid broadside coupling to traces on other layers. The broadside effect significantly increases the insertion loss and reduce signal quality.</p> <p>Make sure digital signals on adjacent layers cross at 90° angles.</p> <p>Note: Refer to the general section of the layout checklist for guidance on routing signals over reference planes.</p>		
	Total trace resistance from the 82577 to the LAN magnetics center tap should not exceed 10 Ω.	The trace resistance includes the series resistance of any components that might be present such as LAN switches, inductors, or resistors.		

<u>SECTION</u>	<u>CHECK ITEMS</u>	<u>REMARKS</u>	<u>DONE</u>	<u>COMMENTS</u>
Chassis Ground	Consider using a separate chassis ground for the LAN connector.	<p>If using a discrete magnetics module, provide a separate chassis ground "island" to ground the shroud of the RJ-45 connector and to terminate the line side of the magnetics module. This design improves EMI behavior.</p> <p>Split in ground plane should be at least 50 mils wide. Split should run under center of magnetics module. Differential pairs never cross the split.</p> <p>If using an integrated magnetics module without USB, provide a separate chassis ground "island" to ground around the RJ-45 connector. Split in ground plane should be at least 50 mils wide.</p> <p>If using an integrated magnetics module with USB, do not use a separate chassis ground.</p> <p>Refer to the <i>Intel® 5 Series Family PDG</i> for more details.</p>		
Magnetics Module	Capacitors connected to center taps should be placed very close (less than 0.1 inch recommended) to the integrated magnetics module. Connect a 1 μ F capacitor near the magnetics module (1/4 inch).	<p>This improves BER.</p> <p>Note that the 0.1 μF capacitors are only needed for discrete magnetics that don't have them integrated internally.</p>		
	Place the motherboard discrete magnetics less than one inch from the motherboard RJ-45 connector. magnetics.	Keep as short as possible.		
	Place the docking discrete magnetics a minimum of one inch from the docking RJ-45 connector.	Keep as short as possible.		

<u>SECTION</u>	<u>CHECK ITEMS</u>	<u>REMARKS</u>	<u>DONE</u>	<u>COMMENTS</u>
LED Circuits	Keep LED traces away from sources of noise, for example, high speed digital traces running in parallel.	LED traces can carry noise into integrated magnetics modules, RJ-45 connectors, or out to the edge of the board, increasing EMI.		
	If using decoupling capacitors on LED lines, place them carefully.	Capacitors on LED lines should be placed near the LEDs (typically adjacent to integrated magnetics module).		

INFORMATION IN THIS DOCUMENT IS PROVIDED IN CONNECTION WITH INTEL® PRODUCTS. NO LICENSE, EXPRESS OR IMPLIED, BY ESTOPPEL OR OTHERWISE, TO ANY INTELLECTUAL PROPERTY RIGHTS IS GRANTED BY THIS DOCUMENT. EXCEPT AS PROVIDED IN INTEL'S TERMS AND CONDITIONS OF SALE FOR SUCH PRODUCTS, INTEL ASSUMES NO LIABILITY WHATSOEVER, AND INTEL DISCLAIMS ANY EXPRESS OR IMPLIED WARRANTY, RELATING TO SALE AND/OR USE OF INTEL PRODUCTS INCLUDING LIABILITY OR WARRANTIES RELATING TO FITNESS FOR A PARTICULAR PURPOSE, MERCHANTABILITY, OR INFRINGEMENT OF ANY PATENT, COPYRIGHT OR OTHER INTELLECTUAL PROPERTY RIGHT. Intel products are not intended for use in medical, life saving, life sustaining applications.

Intel may make changes to specifications and product descriptions at any time, without notice.

This document contains information on products in the design phase of development. The information here is subject to change without notice. Do not finalize a design with this information.

The 82577 GbE Controller may contain design defects or errors known as errata which may cause the product to deviate from published specifications. Current characterized errata are available on request.

*Other names and brands may be claimed as the property of others.

Copyright © Intel Corporation 2004, 2005, 2006, 2007, 2008, 2009, 2010. INTEL CONFIDENTIAL