Intel® 815EP Chipset Platform

Design Guide

August 2002

Document Reference Number: 290692-002

Contents

Figures

Tables

Revision History

Rev.	Description	Date
-001	• Initial Release	November 2000
-002	• Revised USB General Guidelines in Section 8.11, USB • Replaced Figure 53. USB Data Signals in Section 8.11, USB • Revised Section 8.9, AC97 • Added AC '97 Audio Codec Detect Circuit and Configuration Options, Section 8.9 • Added SPKR Pin Considerations, Section 8.9.3 • Added Disabling the Native USB Interface of the Intel® ICH2, Section 8.11.1 • Revised Section 8.17, LAN Layout Guidelines • Revised LAN Connect Interface Checklist, Section 11.4.3 • Revised AC '97 Checklist, Section 11.4.14 • Added Section 8.18.3 FWH Decoupling • Revised Primary IDE Connector Requirements, Section 8.8.3 • Revised Secondary IDE Connector Requirements, Section 8.8.4 • Revised Intel® 82562ET / 82562EH Dual Footprint Guidelines, Section 8.17.4.6 • Added 3.3V/V5REF Sequencing Requirement, Section 10.4.3 • Added Section 8.16.8, Power-well Isolation Control • Revised Section 8.16.1, RTC Crystal, Note 1 • Revised Section 2, General Design Considerations • Revised Section 8.18.2, FWH Vpp Design Guidelines • Added Section 8.17.4.6, Intel® 82562ET/EM Disable Guidelines • Added RTCRST# to Section 11.4.13, RTC Checklist • Revised Packing/Power in Section 1.2.2.1 • Replaced Figure 79. Power Delivery Map in Section 10, Power Delivery • Added SUSCLK to Section 11.4.13, RTC Checklist • Added Section 8.5, Power Supply PS_ON Considerations • Replaced Figure 56. External Circuitry for the Intel® ICH2 RTC, in Section 8.16.3	August 2002

This page is intentionally left blank.

1. Introduction

This design guide organizes Intel's design recommendations for Intel® 815EP chipset-based systems. In addition to providing motherboard design recommendations such as layout and routing guidelines, this document also addresses system design issues such as thermal requirements for 815EP chipset-based systems.

This document contains design recommendations, debug recommendations, and a system checklist. These design guidelines have been developed to ensure maximum flexibility for board designers while reducing the risk of board-related issues.

There are no Intel schematics specifically for the 815EP platform. The 815E Customer Reference Board Schematics, Rev 1.0, may be used as a guide for 815EP design. Motherboard designers who elect to use 815E schematics should understand the following:

- All 815E single function device pins associated with internal graphics (digital video out/TV out signals, display interface signals and clock signals specific to the display cache) will be renamed in the 815EP datasheet. A table of these pins in this Design Guide. See Table 1

The debug recommendations in this Design Guide should be consulted when debugging an 815EP chipset-based system. However, these debug recommendations should be understood before completing board design, to ensure that the debug port, in addition to other debug features, will be implemented correctly.

1.1. Reference Documents

- *Intel® 815 Chipset Family: 82815EP Memory Controller Hub (MCH) Datasheet* (document number: TBD), available December 2000, as an Intel confidential white-paper document.
- *Intel® 82802AB/82802AC Firmware Hub (FWH) Datasheet* (document number: 290658)
- *Intel® 82801BA I/O Controller Hub (ICH2) Datasheet* (document number: 290687)
- *Pentium® II Processor AGTL+ Guidelines* (document number: 243330)
- *Pentium® II Processor Power Distribution Guidelines* (document number: 243332)
- *Pentium® II Processor Developer's Manual* (document number: 243341)
- *Pentium® III Processor Specification Update* (latest revision from website)
- *AP 907 Pentium® III Processor Power Distribution Guidelines* (document number: 245085)
- *AP-585 Pentium® II Processor AGTL+ Guidelines* (document number: 243330)
- *AP-587 Pentium® II Processor Power Distribution Guidelines* (document number: 243332)
- *CK815E Clock Synthesizer Design Guidelines*
- *PCI Local Bus Specification*, Revision 2.2
- *Universal Serial Bus Specification*, Revision 1.0

- *VRM 8.4 DC-DC Converter Design Guidelines*

1.2. System Overview

The 815EP chipset platform contains a Memory Controller Hub (MCH) component and Intel® I/O Controller Hub 2 (ICH2) component for desktop platforms.

The MCH provides the processor interface (optimized for the Intel® Pentium®III processors and Intel® Celeron® processors), DRAM interface, a hub interface, and an AGP interface. This product provides flexibility and scalability in external graphic choices and memory subsystem performance. Competitive external graphics may be scaled via the AGP card interface and PC100 SDRAM system memory may be scaled to PC133 system memory.

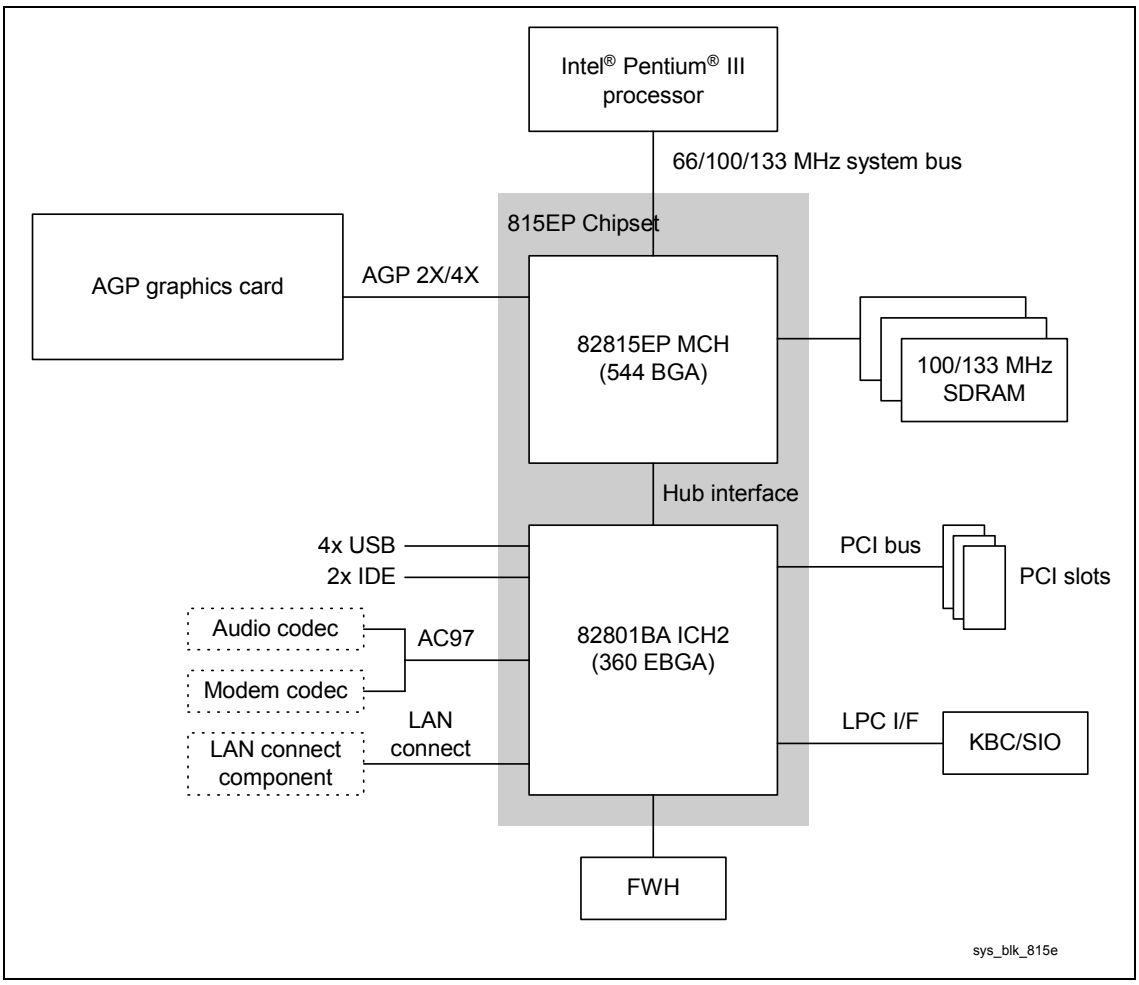
The Accelerated Hub Architecture interface (i.e., the chipset component interconnect) is designed into the chipset to provide an efficient, high-bandwidth communication channel between the 815EP chipset's graphics and memory controller hub and the I/O hub controller. The chipset architecture also enables a security and manageability infrastructure through the Firmware Hub component.

An ACPI-compliant 815EP chipset platform can support the *Full-on (S0), Stop Grant (S1), Suspend to RAM (S3), Suspend to Disk (S4),* and *Soft-off (S5)* power management states. The chipset also supports *wake-on-LAN*** for remote administration and troubleshooting. The chipset architecture removes the requirement for the ISA expansion bus that was traditionally integrated into the I/O subsystem of PCIsets/AGPsets. This removes many of the conflicts experienced when installing hardware and drivers into legacy ISA systems. The elimination of ISA provides true *plug-and-play* for the platform. Traditionally, the ISA interface was used for audio and modem devices. The addition of AC'97 allows the OEM to use *software-configurable* AC'97 audio and modem coder/decoders (codecs), instead of the traditional ISA devices.

1.2.1. System Features

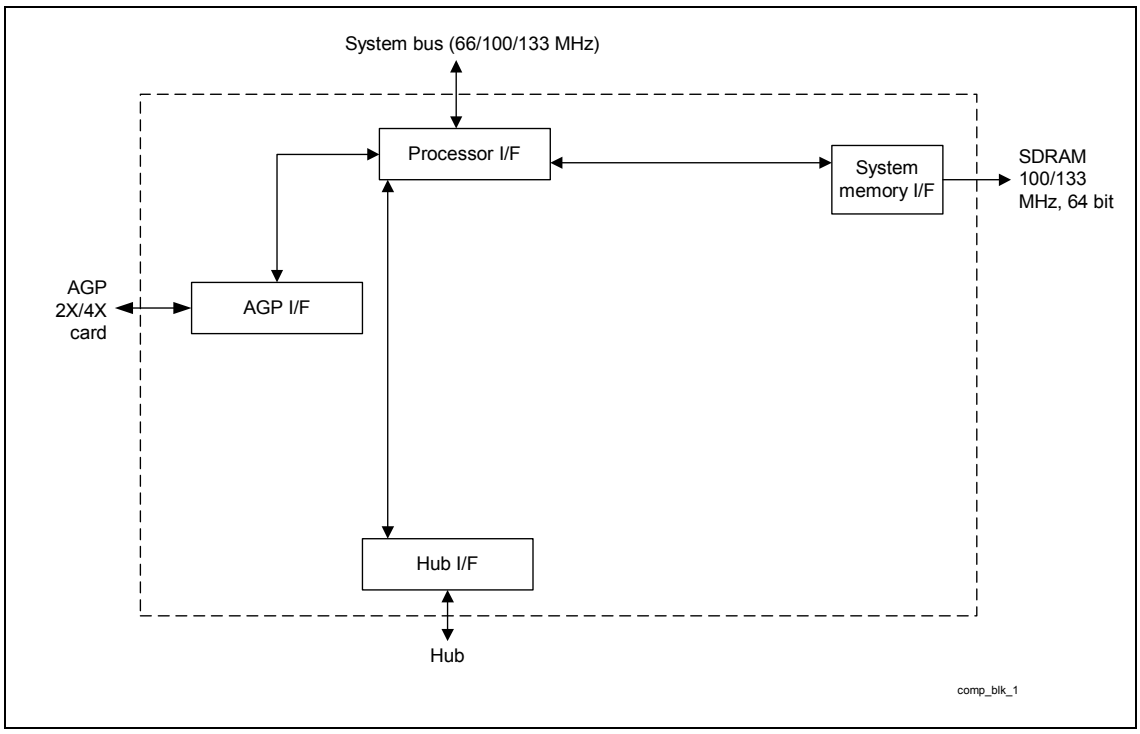
The 815EP chipset contains two components: the 82815EP Memory Controller Hub (MCH) and the 82801BA I/O Controller Hub 2 (ICH2). The MCH integrates a 66/100/133-MHz, P6 family system bus controller, AGP (2X/4X) discrete graphics card, 100/133-MHz SDRAM controller, and a high-speed accelerated hub architecture interface for communication with the ICH2. The ICH2 integrates an UltraATA/100 controller, 2 USB host controllers with a total of 4 ports, LPC interface controller, FWH interface controller, PCI interface controller, AC-link, integrated LAN controller, and a hub interface for communication with the MCH.

Figure 1. System Block Diagram



1.2.2. Component Features

Figure 2. Component Block Diagram



1.2.2.1. Intel® 82815EP MCH

- Processor/System Bus Support
  - Optimized for the Intel® Pentium® III processor at 133-MHz system bus frequency
  - Support for Intel® Celeron® processors (FC-PGA) 533 MHz and >566 MHz (66 MHz system bus)
  - Supports 32-bit AGTL+ bus addressing (no support for 36-bit address extension)
  - Supports uniprocessor systems
  - AGTL+ bus driver technology (gated AGTL+ receivers for reduced power)
- Integrated DRAM controller
  - 32 MB to 256 MB using 64-Mb technology, 512 MB using 128-Mb technology
  - Supports up to 3 double-sided DIMMS (6 rows)
  - 100-MHz, 133-MHz SDRAM interface
  - 64-bit data interface
  - Standard SDRAM (synchronous) DRAM support (x-1-1-1 access)
  - Supports only 3.3-V DIMM DRAM configurations
  - No registered DIMM support
  - Support for symmetrical and asymmetrical DRAM addressing
  - Support for x8, x16 DRAM devices width
  - Refresh mechanism: CAS-before-RAS only
  - Support for DIMM serial PD (presence detect) scheme via SMBus interface
  - STR power management support via self-refresh mode using CKE

- Accelerated Graphics Port (AGP) Interface
  - Supports AGP 2.0, including 4X AGP data transfers, but not the 2X/4X Fast Write protocol
  - AGP universal connector support via dual-mode buffers to allow AGP 2.0 3.3-V or 1.5-V signaling
  - 32-deep AGP request queue
  - AGP address translation mechanism with integrated fully associative 20-entry TLB
  - High-priority access support
  - Delayed transaction support for AGP reads that can not be serviced immediately
  - AGP semantic traffic to the DRAM is not snooped on the system bus and is therefore not coherent with the processor caches.
- Packaging/Power
  - 544 BGA with local memory port
  - 1.85 V core and mixed 3.3 V, 1.5 V, and AGTL+ IO. Note that the 82801BA ICH2 has a 1.8 V requirement and the 82815EP MCH has a 1.85 V requirement. Instead of separate voltage regulators to meet these requirements, a single voltage regulator can be set to 1.79 5V to 1.910V. See Figure 72, the *Power Delivery Map*.

1.2.2.2. Intel® 815E to 815EP Signal Name Changes

82815E pins associated with display interface signals, digital video out/TV-out signals, and some clock, power , and ground signals have name changes. The following table shows the old 82815E signal name, the ball number, and the new 82815EP signal name.

Table 1. Intel® 82815E to 82815EP Pin Name Changes

815E Signal Name	Ball#	815EP Signal Name
LTVDATA0	AD16	NC
LTVDATA1	AF17	NC
LTVDATA2	AE17	NC
LTVDATA3	AD17	NC
LTVDATA4	AF18	NC
LTVDATA5	AD18	NC
LTVDATA6	AF20	NC
LTVDATA7	AD20	NC
LTVDATA8	AC20	NC
LTVDATA9	AF21	NC
LTVDATA10	AE21	NC
LTVDATA11	AD21	NC
LTVBLANK#	AB19	NC
TVCLKIN/INT#	AC18	PU1.8
LTVCLKOUT0	AE19	NC
LTVCLKOUT1	AF19	NC
LTVVSYNC	AC16	NC
LTVHSYNC	AB17	NC
LTVDA	AA20	PU

815E Signal Name	Ball#	815EP Signal Name
LTVCK	AB21	PU
DDCK	AB18	PU
DDDA	AA18	PU
DCLKREF	AE24	PD
IWASTE	Y20	PD
IREF	AD23	PD
VSYNC	AF22	NC
HSYNC	AF23	NC
RED	AD22	NC
GREEN	AE22	NC
BLUE	AE23	NC
LOCLK	R22	NC
LRCLK	P22	PD
VSSDA	Y19	VSS
VSSDACA	AE25	VSS
VSSDACA	AF24	VSS

NOTES:

NC = No Connect. These pins should float

PU = Pull Up. These GPIO(x) related pins should be pulled up to the appropriate voltage through a weak pull-up resistor. (8.2K to 10K Ohm resistor.) The appropriate voltage depends upon in which voltage well the GPIO(x) resides. Typically, this is the 3.3V core voltage well.

PD = Pull Down. These pins should be pulled down to ground through a weak pull-down resistor. (8.2K to 10K Ohm resistor.)

VSS = Connect to ground.

PU1.8 = Pull Up to 1.8V through a weak pull-up resistor. (8.2K to 10K Ohm resistor.)

1.2.2.3. Intel® 82801BA I/O Controller Hub 2 (ICH2)

The I/O Controller Hub 2 allows the I/O subsystem to access the rest of the system, as follows:

- Upstream accelerated hub architecture interface for access to the MCH
- PCI 2.2 interface (6 PCI Req/Grant pairs)
- Bus master IDE controller: supports Ultra ATA/100
- USB controller
- SMBus controller
- FWH interface
- LPC interface
- AC'97 2.1 interface
- Integrated system management controller
- Alert-on-LAN*
- Integrated LAN controller
- Packaging / power
  - 360 EBGA
  - 1.8-V core and 3.3-V standby

1.2.2.4. Firmware Hub (FWH)

The hardware features of this device include:

- An integrated hardware Random Number Generator (RNG)
- Register-based locking
- Hardware-based locking
- 5 GPIs
- Packaging/Power
  - 40L TSOP and 32L PLCC
  - 3.3-V core and 3.3 V / 12 V for fast programming

1.2.3. Platform Initiatives

1.2.3.1. Intel® PC 133

Intel PC133 initiative provides the memory bandwidth necessary to obtain high performance from the Pentium III processor and AGP graphics controllers. The 815EP chipset's SDRAM interface supports 100-MHz and 133-MHz operation. The latter delivers 1.066 GB/s of theoretical memory bandwidth compared with the 800-MB/s theoretical memory bandwidth of 100-MHz SDRAM systems.

1.2.3.2. Accelerated Hub Architecture Interface

As I/O speeds increase, the demand placed on the PCI bus by the I/O bridge becomes significant. With the addition of AC'97 and Ultra ATA/100, coupled with the existing USB, I/O requirements could impact PCI bus performance. The 815EP chipset's *accelerated hub architecture* ensures that the I/O subsystem, both PCI and the integrated I/O features (IDE, AC'97, USB, LAN), receives adequate bandwidth. By placing the I/O bridge on the accelerated hub architecture interface instead of PCI, I/O functions integrated into the ICH2 and the PCI peripherals are ensured the bandwidth necessary for peak performance.

1.2.3.3. Internet Streaming SIMD Extensions

The PentiumIII processor provides 70 new SIMD (single instruction, multiple data) instructions. The new extensions are floating-point SIMD Extensions. Intel® MMX™ technology provides integer SIMD instructions. The Internet Streaming SIMD extensions complement the MMX technology SIMD instructions and provide a performance boost to floating-point-intensive 3D applications.

1.2.3.4. AGP 2.0

The AGP 2.0 interface allows graphics controllers to access main memory at over 1 GB/s, twice the bandwidth of previous AGP platforms. AGP 2.0 provides the infrastructure necessary for *photorealistic 3D*. In conjunction with the Internet Streaming SIMD Extensions, AGP 2.0 delivers the next level of 3D graphics performance.

1.2.3.5. Integrated LAN Controller

The 815EP chipset platform incorporates an ICH2 integrated LAN Controller. Its bus master capabilities enable the component to process high-level commands and perform multiple operations; this lowers processor utilization by off-loading communication tasks from the processor.

The ICH2 functions with several options of LAN connect components to target the desired market segment. The Intel® 82562EH provides a HomePNA 1-Mbit/sec connection. The Intel® 82562ET provides a basic Ethernet 10/100 connection. The Intel® 82562EM provides an Ethernet 10/100 connection with the added flexibility of Alert on LAN.

1.2.3.6. Ultra ATA/100 Support

The 815EP chipset platform incorporates the ICH2 IDE controller with two sets of interface signals (primary and secondary) that can be independently enabled, tri-stated or driven low. The platform supports Ultra ATA/100 for transfers up to 100MB/sec, in addition to Ultra ATA/66, and Ultra ATA/33 modes.

1.2.3.7. Expanded USB Support

The 815EP chipset platform contains two USB Host Controllers. Each Host Controller includes a root hub with two separate USB ports each, for a total of 4 USB ports. The addition of a second USB Host Controller expands the functionality of the platform.

1.2.3.8. Manageability and Other Enhancements

The 815EP chipset platform integrates several functions designed to manage the system and lower the total cost of ownership (TCO) of the system. The platform supports all features in the 815EP chipset, in addition to the following features. These system management functions are designed to report errors, diagnose the system, and recover from system lockups, without the aid of an external microcontroller.

SMBus

The ICH2 integrates a SMBus controller. The SMBus provides an interface for managing peripherals such as serial presence detection (SPD) and thermal sensors. The slave interface allows an external microcontroller to access system resources.

Interrupt Controller

The interrupt capabilities of the 815EP chipset platform expand support for up to 8 PCI interrupt pins and PCI 2.2 message-based interrupts. In addition, the ICH2 supports system bus interrupt delivery.

Firmware Hub (FWH)

The 815EP chipset platform supports firmware hub BIOS memory sizes up to 8 MB for increased system flexibility.

1.2.3.9. AC'97 6-Channel Support

The *Audio Codec '97* (AC'97) Specification defines a digital interface that can be used to attach an *audio codec* (AC), a *modem codec* (MC), an *audio/modem codec* (AMC), or both an AC and a MC. The AC'97 Specification defines the interface between the system logic and the audio or modem codec known as the *AC-link*.

The 815EP chipset's AC'97 (with the appropriate codecs) not only replaces ISA audio and modem functionality, but also improves overall platform integration by incorporating the AC-link. Using 815EP chipset's integrated AC-link reduces cost and eases migration from ISA.

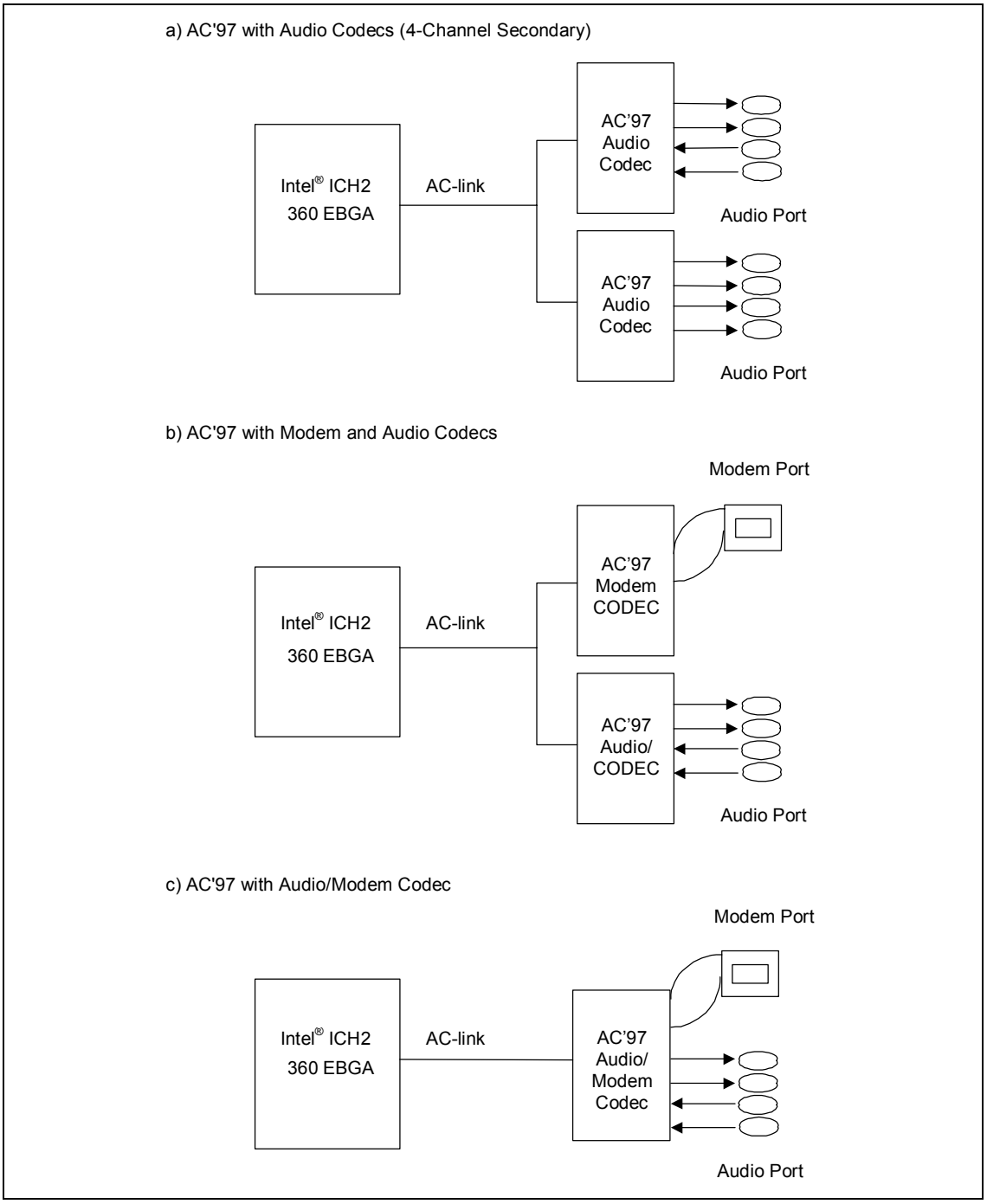
By using an audio codec, the AC-link allows for cost-effective, high-quality, integrated audio on the 815EP chipset platform. In addition, an AC'97 soft modem can be implemented with the use of a modem codec. Several system options exist when implementing AC'97. The 815EP chipset's integrated digital link allows several external codecs to be connected to the ICH2. The system designer can provide audio with an audio codec, a modem with a modem codec, or an integrated audio/modem codec (Figure 3c). The digital link is expanded to support two audio codecs (Figure 3a) or a combination of an audio and modem codec (Figure 3b).

Modem implementation for different countries must be taken into consideration, as telephone systems may vary. By implementing a split design, the audio codec can be on board and the modem codec can be placed on a riser. Intel is developing a Communications and Networking Riser connector.

The digital link in the ICH2 is AC'97 Rev. 2.1 compliant, supporting two codecs with independent PCI functions for audio and modem. Microphone input and left and right audio channels are supported for a high-quality, two-speaker audio solution. Wake-on-ring-from-suspend also is supported with the appropriate modem codec.

The 815EP chipset platform expands audio capability with support for up to six channels of PCM audio output (i.e., full AC3 decode). Six-channel audio consists of Front Left, Front Right, Back Left, Back Right, Center and Woofer, for a complete surround sound effect. ICH2 has expanded support for two audio codecs on the AC-link.

Figure 3. AC'97 Audio and Modem Connections



1.2.3.10. Low-Pin-Count (LPC) Interface

In the 815EP chipset platform, the Super I/O (SIO) component has migrated to the Low-Pin-Count (LPC) interface. Migration to the LPC interface allows for lower-cost Super I/O designs. The LPC Super I/O component requires the same feature set as traditional Super I/O components. It should include a keyboard and mouse controller, floppy disk controller, and serial and parallel ports. In addition to the Super I/O features, an integrated game port is recommended because the AC'97 interface does not provide support for a game port. In systems with ISA audio, the game port typically existed on the audio card. The fifteen-pin game port connector provides for two joysticks and a two-wire MPU-401 MIDI interface. Consult your preferred Super I/O vendor for a comprehensive list of the devices offered and the features supported.

In addition, depending on system requirements, specific system I/O requirements may be integrated into the LPC Super I/O. For example, a USB hub may be integrated to connect to the ICH2 USB output and extend it to multiple USB connectors. Other SIO integration targets include a device bay controller or an ISA-IRQ-to-serial-IRQ converter to support a PCI-to-ISA bridge. Contact your Super I/O vendor to ensure the availability of desired LPC Super I/O features.

1.2.3.11. Security – The Intel® Random Number Generator (RNG)

The 815EP chipset based system contains the first of Intel's platform security features, the Intel® Random Number Generator (RNG). The RNG is a component of the 82802 Firmware Hub (FWH), and it supplies applications and security middleware products with true non-deterministic random numbers, through the Intel® Security Driver.

Better random numbers lead to better security. Most cryptographic functions, especially functions that provide authentication or encryption services, require random numbers for such purposes as key generation. One attack on those cryptographic functions is to predict the random numbers being used to generate those keys. Current methods that use system and user input to seed a pseudo-random number generator have proved vulnerable to such attacks. The RNG uses thermal noise across a resistor to generate true non-deterministic, unpredictable random numbers.

Applications often access cryptographic functions through security middleware products such as Microsoft's CAPI*, RSA's BSAFE*, and the OpenGroup's CDSA*. Intel is working to ensure that middleware products and applications are enabled to take advantage of this capability. By implementing the BIOS requirements and testing and loading the Intel Security Driver, it is possible to ensure that the RNG is enabled for a platform design.

The ICH2 BIOS Specification contains complete details regarding BIOS requirements for enabling the RNG. In summary, the system BIOS must contain a System Device Node for the FWH device for plug-and-play operating systems (OS) to use the Random Number Generator through the Security Driver. The devnode is required for the OS to find the FWH at enumeration time, and the specific devnode number associates the FWH with the Security Driver.

- The BIOS must report a single device node for the FWH.
- Intel-specific EISA ID (devnode number must be INT0800)
- Device type: System peripherals / other
- Device attrib: Non-configurable and cannot be disabled
- ANSI ID string: “Intel FWH”
- Memory range descriptor: Describing feature space
- For PnP OSes, BIOS ranges are allocated through E820h and ACPI structures, as in current BIOSes.
- For non-PnP OSes, FWH ranges should be reserved through the Int 15h E820h function.

A complete 815EP chipset-based system must have the Security Driver loaded for applications to take advantage of the Random Number Generator. The Security Driver implements an interface that middleware and some applications call to access the RNG. The Security Driver can be obtained from the PCG Chipset Driver download website at http://developer.intel.com/design/chipsets/drivers/SWDev/.

This page is intentionally left blank.

2. General Design Considerations

This section documents motherboard layout and routing guidelines for 815EP chipset-based systems. This section does not discuss the functional aspects of any bus or the layout guidelines for an add-in device.

If the guidelines listed in this document are not followed, it is very important that thorough signal integrity and timing simulations be completed for each design. Even when the guidelines are followed, critical signals should be simulated to ensure the proper signal integrity and flight time. Any deviation from these guidelines should be simulated.

The trace impedance typically noted (i.e., 60 Ω ± 15%) is the "nominal" trace impedance for a 5-mil-wide trace. That is, it is the impedance of the trace when not subjected to the fields created by changing current in neighboring traces. When calculating flight times, it is important to consider the minimum and maximum impedance of a trace, based on the switching of neighboring traces. The use of wider spaces between the traces can minimize this trace-to-trace coupling. In addition, these wider spaces reduce the settling time.

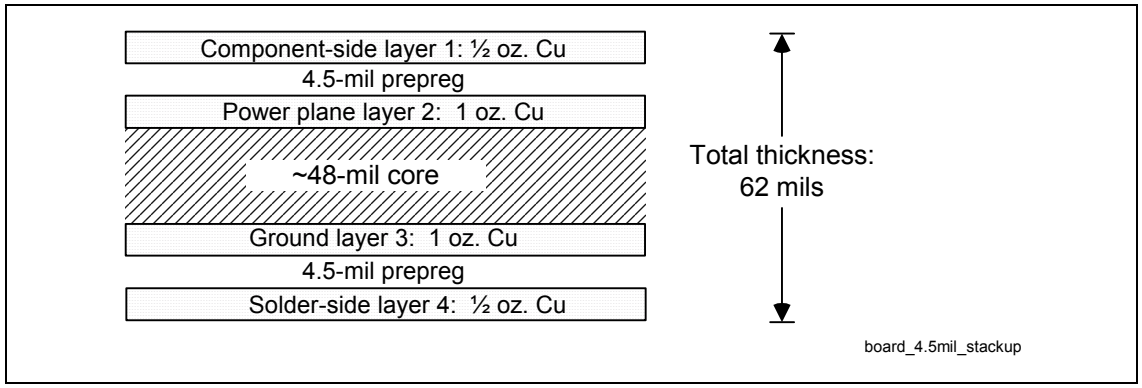
Coupling between two traces is a function of the coupled length, the distance separating the traces, the signal edge rate, and the degree of mutual capacitance and inductance. To minimize the effects of trace-to-trace coupling, the routing guidelines documented in this section should be followed.

Additionally, these routing guidelines are created using a PCB stack-up similar to that illustrated in the following figure. If this stack-up is not used, extremely thorough simulations of every interface must be completed. Using a thicker dielectric (prepreg) will make routing very difficult or impossible.

2.1. Nominal Board Stack-Up

The 815EP chipset platform requires a board stack-up yielding a target impedance of 60 Ω ± 15% with a 5-mil nominal trace width. The following figure presents an example stack-up that achieves this. It is a 4-layer printed circuit board (PCB) construction using 53%-resin FR4 material.

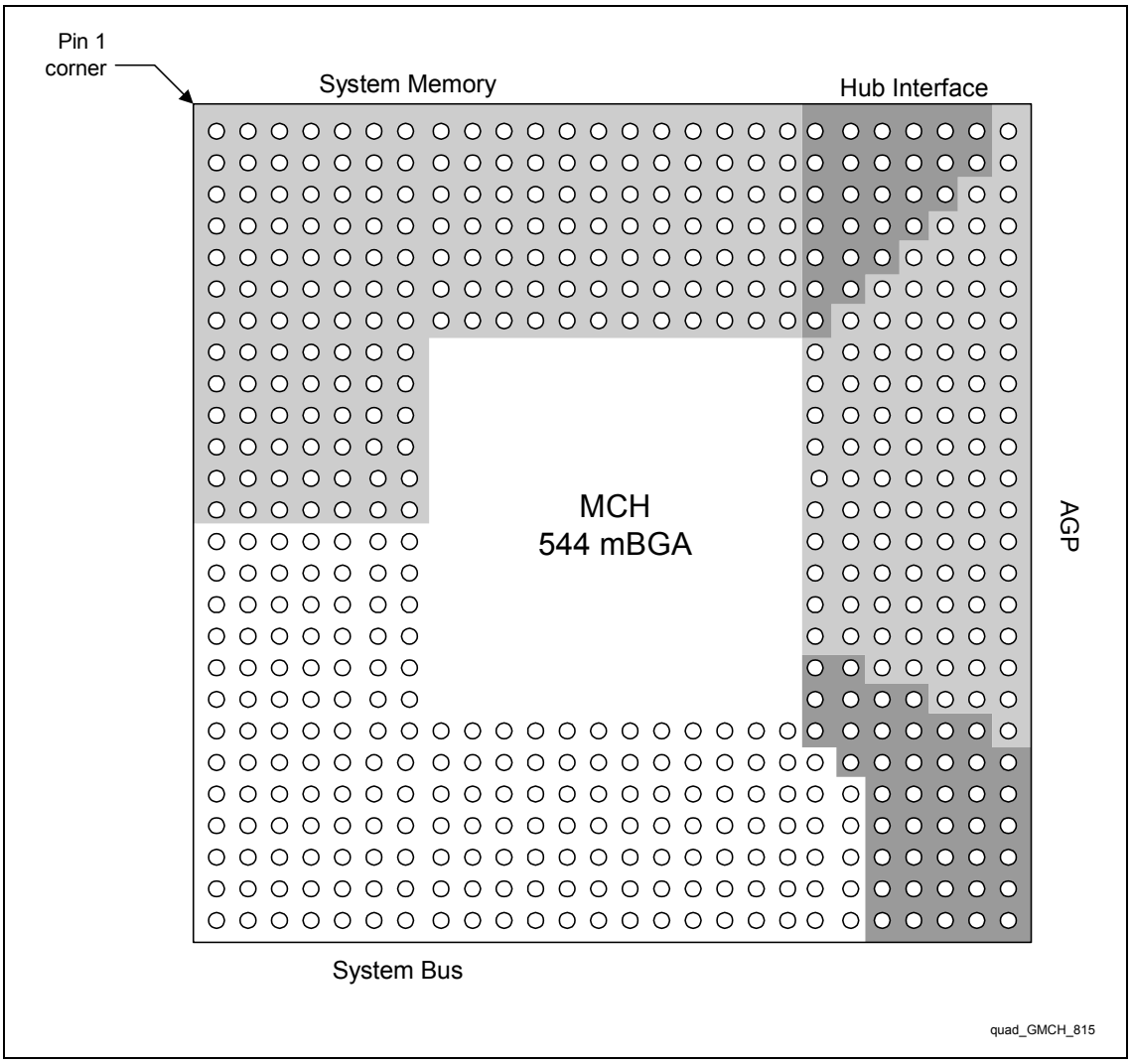
Figure 4. Board Construction Example for 60-Ω Nominal Stack-Up



This page is intentionally left blank.

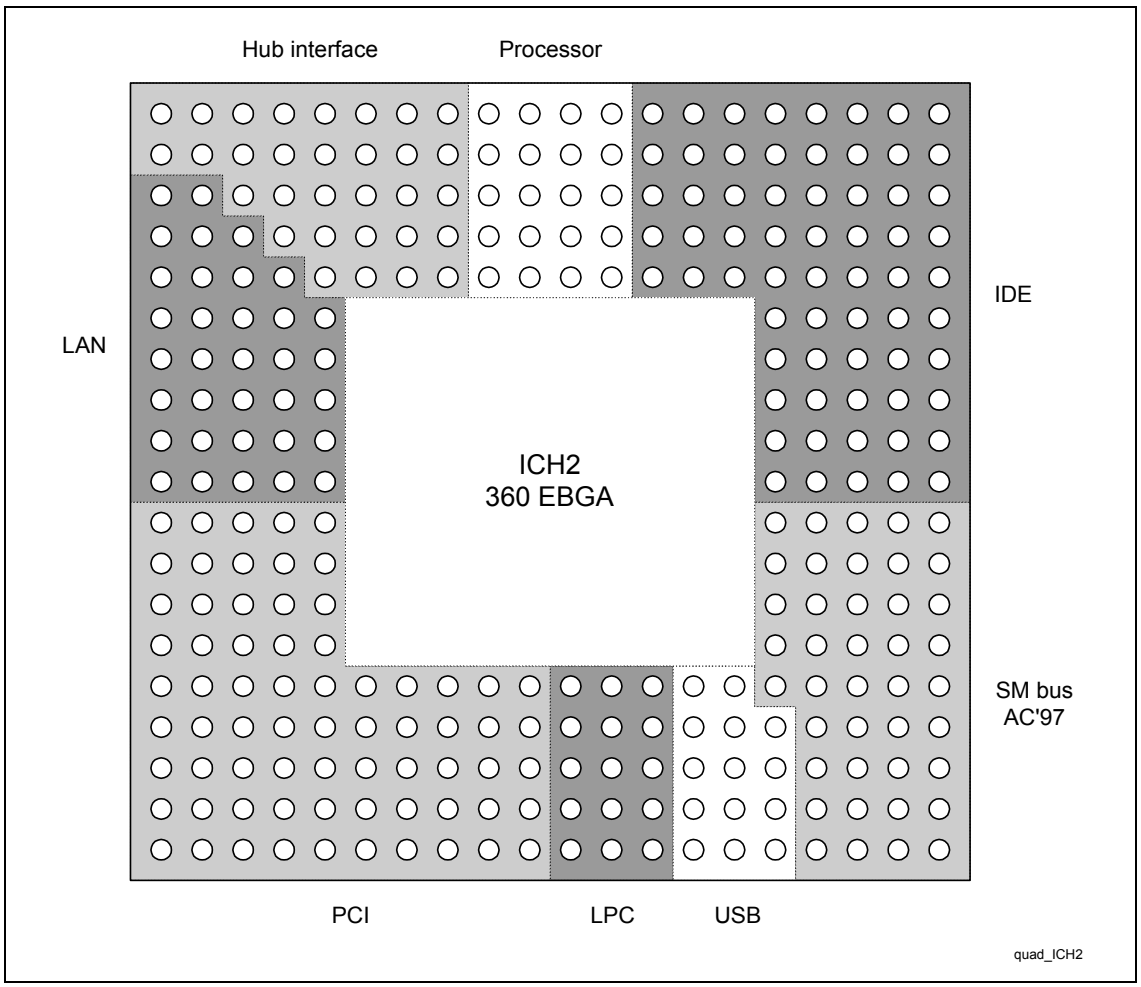
3. Component Quadrant Layouts

Figure 5. Intel® 82815EP MCH 544-mBGA Quadrant Layout (Top View)



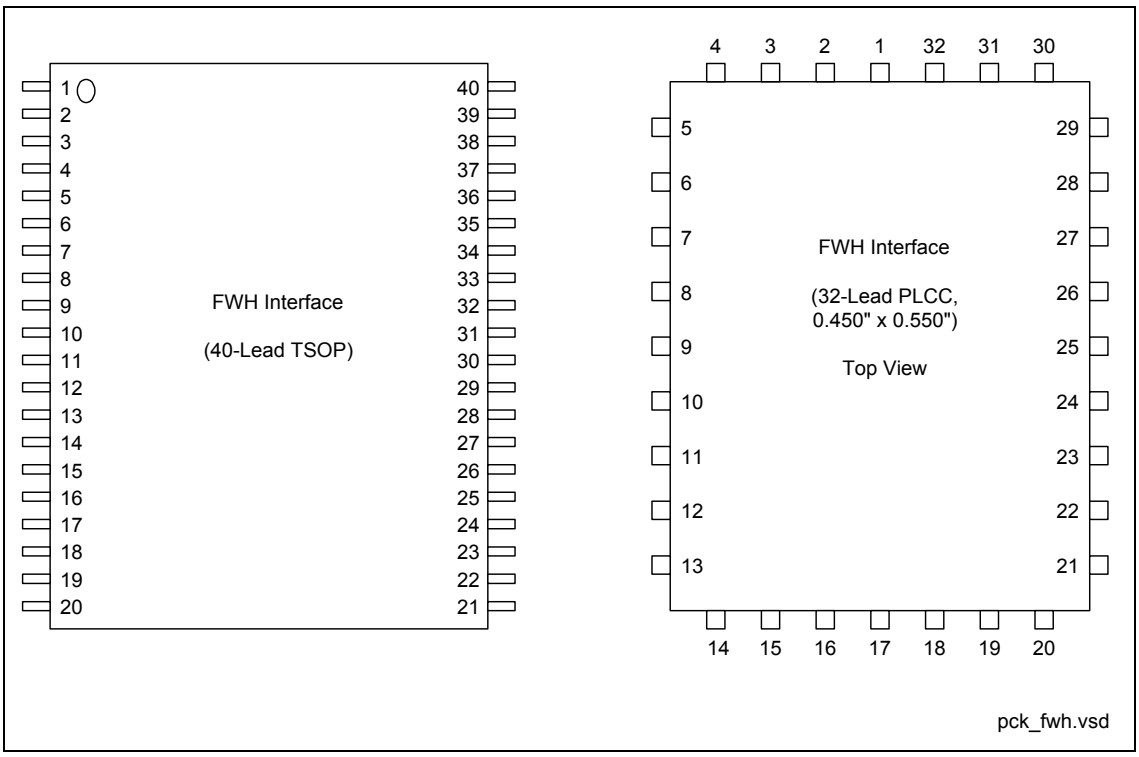
The previous figure illustrates the relative signal quadrant locations on the MCH ballout. It does not represent the actual ballout. Refer to the *Intel® 815 Chipset Family: 82815EP Graphics and Memory Controller Hub (MCH) Datasheet* for the actual ballout.

Figure 6. Intel® ICH2 Quadrant Layout (Top View)



The diagram in the previous figure illustrates the relative signal quadrant locations on the ICH2 ballout. It does not represent the actual ballout. Refer to the *Intel® 82801BA I/O Controller Hub 2 (ICH2) Datasheet* for the actual ballout.

Figure 7. Firmware Hub (FWH) Packages



This page is intentionally left blank.

4. System Bus Design Guidelines

4.1. Introduction

The newest generation of the Pentium III processor delivers higher performance by integrating the Level 2 cache into the processor and running it at the processor's core speed. The Pentium III processor runs at a higher core and system bus speeds than previous-generation IA-32 processors, while maintaining hardware and software compatibility with earlier Pentium III processors. New Flip Chip-Pin Grid Array (FC-PGA) package technology enables compatibility with the PGA370 socket.

This section presents the design considerations for flexible platforms capable of using the 815EP chipset with the full range of Pentium III processors using the PGA370 socket.

4.1.1. Terminology

For this document, the following terminology applies:

- *Flexible PGA370* refers to new-generation 815EP chipsets using the new, "flexible" PGA370 socket. In general, these designs support 100/133-MHz system bus operation, VRM 8.4 DC-DC converter guidelines, and the Pentium III processor (FC-PGA) in single-microprocessor based designs.

Note: The system bus speed supported by the design is based on the capabilities of the used processor, chipset, and clock driver.

4.2. System Bus Routing Guidelines

The following layout guide supports designs using Pentium III processors with the 815EP chipset. The solution covers system bus speeds of 100/133 MHz for Pentium III processors. The solution proposed in this section requires the motherboard design to terminate the system bus AGTL+ signals with 56 Ω ± 5% Rtt resistors. Pentium III processors in FC-PGA must also be configured to 110-Ω internal Rtt resistors.

4.2.1. Initial Timing Analysis

The following table lists the AGTL+ component timings of the processors and 815EP chipset's MCH defined at the pins. **These timings are for reference only. Obtain each processor's specifications from its respective processor electrical, mechanical, and thermal specification and the appropriate 815EP chipset component specification.**

Table 2. Intel® Pentium® III Processor AGTL+ Parameters for Example Calculations

IC Parameters	Intel® Pentium® III Processor (FC-PGA) at 133-MHz System Bus	82815EP MCH	Notes
Clock to Output maximum (T_{CO_MAX})	3.25 ns (for 66/100/133-MHz system bus speeds)	4.1	2
Clock to Output minimum (T_{CO_MIN})	0.40 ns (for 66/100/133-MHz system bus)	1.05	2
Setup time (T_{SU_MIN})	1.20 ns for BREQ Lines 0.95 for all other AGTL+ Lines @ 133 MHz 1.20 ns for all other AGTL+ Lines @ 66/100 MHz	2.65	2,3
Hold time (T_{HOLD})	1.0 ns (for 66/100/133-MHz system bus speeds)	0.10	

NOTES:
1. All times in nanoseconds.
2. **Numbers in table are for reference only**. These timing parameters are subject to change. Check the appropriate component documentation for the valid timing parameter values.
3. T_{SU_MIN} = 2.65 ns assumes that the MCH sees a minimum edge rate equal to 0.3 V/ns.

The following table contains an example AGTL+ initial maximum flight time, and Table 4 contains an example minimum flight time calculation for a 133-MHz, uniprocessor system using the Pentium III processor (FC-PGA) and the 815EP chipset's system bus. Note that assumed values were used for the clock skew and clock jitter. **The clock skew and clock jitter values depend on the clock components and the distribution method chosen for a particular design and must be budgeted into the initial timing equations, as appropriate for each design.**

The following table and Table 4 were derived assuming the following:

- CLK_{SKEW} = 0.20 ns (Note: This assumes that the clock driver pin-to-pin skew is reduced to 50 ps by tying the two host clock outputs together (i.e., "ganging") at the clock driver output pins, and that the PCB clock routing skew is 150 ps. The system timing budget must assume 0.175 ns of clock driver skew if outputs are not tied together as well as the use of a clock driver that meets the CK-815 Clock Synthesizer/Driver Specification.)
- CLK_{JITTER} = 0.250 ns

See the respective processor's electrical, mechanical, and thermal specification, the appropriate 815EP chipset documentation, and the *CK-815E Clock Synthesizer/Driver Specification* for details on clock skew and jitter specifications. Exact details regarding the host clock routing topology are provided with the platform design guideline.

Table 3. Example T_{FLT_MAX} Calculations for 133-MHz Bus

Driver	Receiver	Clk Period[2]	T_{CO_MAX}	T_{SU_MIN}	Clk_{SKEW}	Clk_{JITTER}	M_{ADJ}	Recommended T_{FLT_MAX}
Processor	MCH	7.50	3.25	2.65	0.20	0.25	0.40	1.1
MCH	Processor	7.50	4.1	1.20	0.20	0.25	0.40	1.35

NOTES:
1. All times in nanoseconds
2. BCLK period = 7.50 ns @ 133.33 MHz

Table 4. Example T_{FLT_MIN} Calculations (Frequency Independent)

Driver	Receiver	T_{HOLD}	Clk_{SKEW}	T_{CO_MIN}	Recommended T_{FLT_MIN}
Processor	MCH	0.10	0.20	0.40	0.10
MCH	Processor	1.00	0.20	1.05	0.15

NOTES:
1. All times in nanoseconds

The flight times in Table 3 include margin to account for the following phenomena that Intel observed when multiple bits are switching simultaneously. These multi-bit effects can adversely affect the flight time and signal quality and sometimes are not accounted for during simulation. Accordingly, the maximum flight times depend on the baseboard design and additional adjustment factors or margins are recommended.

- SSO push-out or pull-in
- Rising or falling edge rate degradation at the receiver caused by inductance in the current return path, requiring extrapolation that causes additional delay
- Crosstalk on the PCB and inside the package can cause variation in the signals.

There are additional effects that **may not necessarily** be covered by the multi-bit adjustment factor and should be budgeted as appropriate to the baseboard design. Examples include:

- The effective board propagation constant (S_{EFF}), which is a function of:
  - Dielectric constant (ε_r) of the PCB material
  - Type of trace connecting the components (stripline or microstrip)
  - Length of the trace and the load of the components on the trace. Note that the board propagation constant multiplied by the trace length is a **component** of the flight time, **but not necessarily equal to** the flight time.

4.3. General Topology and Layout Guidelines

The following topology and layout guidelines are preliminary and subject to change. The guidelines are derived from empirical testing with the 810E chipset as well as correlative simulations with preliminary 815EP chipset package models. Refer to the Celeron processor datasheet and the Pentium III processor for the PGA370 socket datasheet for detailed information on processor signal groups and pin definitions.

In the Single-Ended Termination (SET) topology for the 370-pin socket (PGA370), the termination should be placed close to the processor on the motherboard. There is no termination present at the chipset end of the network. Due to the lack of termination, SET will exhibit much more ringback than the dual-terminated topology. Extra care will be required in SET simulations to make sure that the ringback specs are met under the worst-case signal quality conditions. 815EP chipset designs require all AGTL+ signals to be terminated with a 56-Ω termination on the motherboard. To ensure processor signal integrity requirements, **it is highly recommended that all system bus signal segments be referenced to the ground plane for the entire route.**

Figure 8. Topology for 370-Pin Socket Designs with Single-Ended Termination (SET)

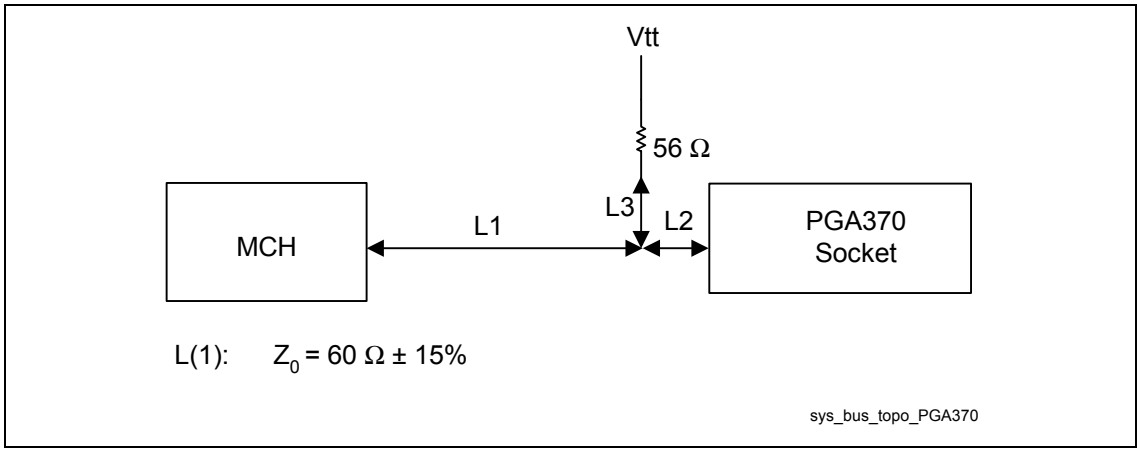


Table 5. Segment Descriptions and Lengths for Figure 8

Segment	Description	Min. Length (inches)	Max. Length (inches)
L1+L2	MCH to Rtt stub	1.90	4.50
L2	PGA370 pin to Rtt stub	0.0	0.20
L3	Rtt stub length	0.50	2.50

NOTES:
1. All AGTL+ bus signals should be referenced to the ground plane for the entire route.

- AGTL+ signals should be routed with trace lengths within the range specified for L1+L2 from the processor pin to the chipset.
- Use an intragroup AGTL+ spacing : line width : dielectric thickness ratio of at least 2:1:1 for microstrip geometry. If $\varepsilon_r = 4.5$, this should limit coupling to 3.4%. For example, intragroup AGTL+ routing could use 10-mil spacing, 5-mil traces, and a 5-mil prepreg between the signal layer and the plane it references (assuming a 4-layer motherboard design).
- The recommended trace width is 5 mils, but not greater than 6 mils.

The following table contains the trace width:space ratios assumed for this topology. Three types of crosstalk are considered in this guideline: Intragroup AGTL+, Intergroup AGTL+, and AGTL+ to non-AGTL+. Intragroup AGTL+ crosstalk involves interference between AGTL+ signals within the same group. Intergroup AGTL+ crosstalk involves interference from AGTL+ signals in a particular group to AGTL+ signals in a different group. An example of AGTL+ to non-AGTL+ crosstalk is when CMOS and AGTL+ signals interfere with each other.

Table 6. Trace Width:Space Guidelines

Crosstalk Type	Trace Width:Space Ratios[1, 2]
Intragroup AGTL+ signals (same group AGTL+)	5:10 or 6:12
Intergroup AGTL+ signals (different group AGTL+)	5:15 or 6:18
AGTL+ to System Memory Signals	5:30 or 6:36
AGTL+ to non-AGTL+	5:20 or 6:24

NOTES:
1. Edge to edge spacing.
2. Units are in mils.

4.3.1.1. Motherboard Layout Rules for AGTL+ Signals

Ground Reference

It is strongly recommended that AGTL+ signals be routed on the signal layer next to the ground layer (referenced to ground). It is important to provide an effective signal return path with low inductance. The best signal routing is directly adjacent to a solid GND plane with no splits or cuts. Eliminate parallel traces between layers not separated by a power or ground plane.

Reference Plane Splits

Splits in reference planes disrupt signal return paths and increase overshoot/undershoot due to significantly increased inductance.

Processor Connector Breakout

It is strongly recommended that AGTL+ signals do not traverse multiple signal layers. Intel recommends breaking out all signals from the connector on the same layer. If routing is tight, break out from the connector on the opposite routing layer over a ground reference and cross over to main signal layer near the processor connector.

Note: Following the previously mentioned layout rules is critical for AGTL+ signal integrity, particularly for the 0.18-micron process technology.

Minimizing Crosstalk

The following general rules minimize the impact of crosstalk in a high-speed AGTL+ bus design:

- Maximize the space between traces. Wherever possible, maintain a minimum of 10 mils (assuming a 5-mil trace) between trace edges. It may be necessary to use tighter spacing when routing between component pins. When traces must be close and parallel to each other, minimize the distance that they are close together and maximize the distance between the sections when the spacing restrictions are relaxed.
- Avoid parallelism between signals on adjacent layers, if there is no AC reference plane between them. As a rule of thumb, route adjacent layers orthogonally.
- Since AGTL+ is a low-signal-swing technology, it is important to isolate AGTL+ signals from other signals by at least 25 mils. This will avoid coupling from signals that have larger voltage swings, such as 5-V PCI.
- AGTL+ signals must be well isolated from system memory signals. AGTL+ signal trace edges must be at least 30 mils from system memory trace edges within 100 mils of the ball of the 82815EP MCH.
- Select a board stack-up that minimizes the coupling between adjacent signals. Minimize the nominal characteristic impedance within the AGTL+ specification. This can be done by minimizing the height of the trace from its reference plane, which minimizes crosstalk.
- Route AGTL+ address, data, and control signals in separate groups to minimize crosstalk between groups. Keep at least 25 mils between each group of signals.
- Minimize the dielectric used in the system. This makes the traces closer to their reference plane and thus reduces the crosstalk magnitude.
- Minimize the dielectric process variation used in the PCB fabrication.
- Minimize the cross-sectional area of the traces. This can be done by means of narrower traces and/or by using thinner copper, but the trade-off for this smaller cross-sectional area is higher trace resistivity, which can reduce the falling-edge noise margin because of the I*R loss along the trace.

4.3.1.2. Motherboard Layout Rules for Non-AGTL+ (CMOS) Signals

Table 7. Routing Guidelines for Non-AGTL+ Signals

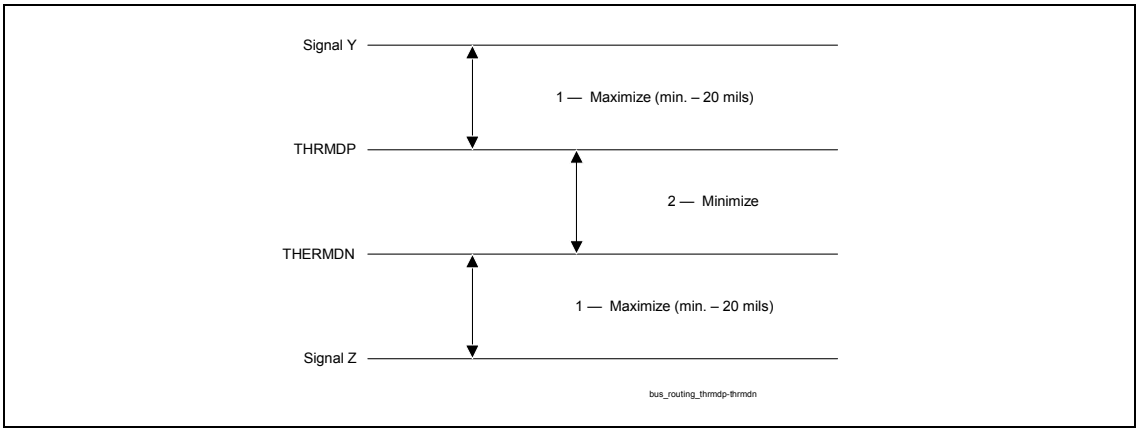
Signal	Trace Width	Spacing to Other Traces	Trace Length
A20M#	5 mils	10 mils	1” to 9”
FERR#	5 mils	10 mils	1” to 9”
FLUSH#	5 mils	10 mils	1” to 9”
IERR#	5 mils	10 mils	1” to 9”
IGNNE#	5 mils	10 mils	1” to 9”
INIT#	5 mils	10 mils	1” to 9”
LINT[0] (INTR)	5 mils	10 mils	1” to 9”
LINT[1] (NMI)	5 mils	10 mils	1” to 9”
PICD[1:0]	5 mils	10 mils	1” to 9”
PREQ#	5 mils	10 mils	1” to 9”
PWRGOOD	5 mils	10 mils	1” to 9”
SLP#	5 mils	10 mils	1” to 9”
SMI#	5 mils	10 mils	1” to 9”
STPCLK	5 mils	10 mils	1” to 9”
THERMTRIP#	5 mils	10 mils	1” to 9”

NOTES:
1. Route these signals on any layer or combination of layers.

4.3.1.3. THRMDP and THRMDN

These traces (THRMDP and THRMDN) route the processor’s thermal diode connections. The thermal diode operates at very low currents and may be susceptible to crosstalk. The traces should be routed close together to reduce loop area and inductance.

Figure 9. Routing for THRMDP and THRMDN



NOTES:
1. Route these traces parallel and equalize lengths within ±0.5”.
2. Route THRMDP and THRMDN on the same layer

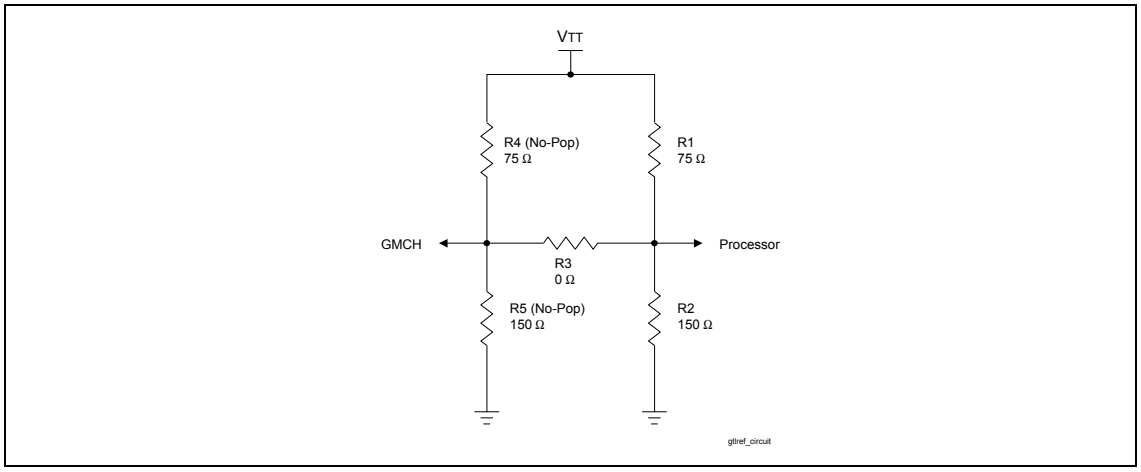
4.3.1.4. Additional Routing and Placement Considerations

- Distribute VTT with a wide trace. A 0.050" minimum trace is recommended to minimize DC losses. Route the VTT trace to all components on the host bus. Be sure to include decoupling capacitors.
- The VTT voltage should be 1.5 V ± 3% for static conditions, and 1.5 V ± 9% for worst-case transient conditions.
- Place resistor divider pairs for V_{REF} generation at the MCH component. V_{REF} also is delivered to the processor.

4.3.2. GTLREF Topology and Layout for Debug

It is strongly recommended that resistor sites be added to the layout to split the GTLREF sources to the processor and the chipset. This will allow the designer to independently modify the reference voltage to each component for debug purposes. The recommended GTLREF circuit topology is shown the figure below.

Figure 10. GTLREF Circuit Topology



- Normal shared GTLREF (one source, routed to both the MCH and CPU)
  - Populate R1, R2, and R3 with values shown
  - Do NOT Populate R4 and R5
- Independent GTLREF for Platform Debug (independent sources for each the MCH and processor)
  - Populate R1, R2, R4, and R5 with values shown
  - Do NOT Populate R3

GTLREF Layout and Routing Guidelines

- Place all resistor sites for GTLREF generation close to the MCH.
- Route GTLREF with as wide a trace as possible.
- Use 1-0.1µF decoupling capacitor for every 2 GTLREF pins at the CPU (4 capacitors total). Place as close as possible (within 500mils) to the Socket 370 GTLREF pins.
- Use 1-0.1µF decoupling capacitor for each of the 2 GTLREF pins at the MCH (2 capacitors total). Place as close as possible to the MCH GTLREF balls.

4.4. Electrical Differences for Flexible PGA370 Designs

There are several electrical changes between the *legacy PGA370* and *flexible PGA370* design, as follows:

- Changes to the PGA370 socket pin definitions. Pentium III processors (FC-PGA) utilize a superset of the Celeron processor (PPGA) pin definition.
- Addition of VTT (AGTL+ termination voltage) delivery to the PGA370 socket.
- BSEL[1:0] implementation differences. BSEL1 has been added to select either a 100-MHz or 133-MHz system bus frequency setting from the clock synthesizer.
- Additional PLL reference voltage, 1.25 V, on new CLKREF pin.
- More stringent undershoot/overshoot requirements for CMOS and AGTL+ signals.
- Addition of on-die Rtt (AGTL+ termination resistors) for the FC-PGA processor. The requirement for on-motherboard Rtt implementation remains if supporting the Celeron PPGA processor. If only supporting FC-PGA processors, the reset signal (RESET#) still requires termination to VTT on the motherboard.

4.5. PGA370 Socket Definition Details

The following tables compare *legacy PGA370* pin names and functions with new *flexible PGA370* pin names and functions. Designers must pay close attention to the notes section of this table for compatibility concerns regarding these pin changes.

Table 8. Platform Pin Definition Comparison for Single-Microprocessor Designs

Pin #	Legacy PGA370 Pin Name	Flexible PGA370 Pin Name	Function	Type	Notes
A29	Reserved	DEP7#	Data bus ECC data	AGTL+, I/O	2
A31	Reserved	DEP3#	Data bus ECC data	AGTL+, I/O	2
A33	Reserved	DEP2#	Data bus ECC data	AGTL+, I/O	2
AA33	Reserved	VTT	AGTL+ termination voltage	Power/other	4
AA35	Reserved	VTT	AGTL+ termination voltage	Power/other	4
AC1	Reserved	A33#	Additional AGTL+ address	AGTL+, I/O	2
AC37	Reserved	RSP#	Response parity	AGTL+, I	2
AF4	Reserved	A35#	Additional AGTL+ address	AGTL+, I/O	2
AH20	Reserved	VTT	AGTL+ termination voltage	Power	
AH4	Reserved	RESET#	Processor reset (used by the Intel® Pentium® III processor FC-PGA)	AGTL+, I	3
AJ31	GND	BSEL1	System bus frequency select	CMOS, I/O	1
AK16	Reserved	VTT	AGTL+ termination voltage	Power	
AK24	Reserved	AERR#	Address parity error	AGTL+, I/O	2
AL11	Reserved	AP0#	Address parity	AGTL+, I/O	2
AL13	Reserved	VTT	AGTL+ termination voltage	Power	
AL21	Reserved	VTT	AGTL+ termination voltage	Power	
AM2	GND	Reserved	Reserved	Reserved	1
AN11	Reserved	VTT	AGTL+ termination voltage	Power	
AN13	Reserved	AP1#	Address parity	AGTL+, I/O	2
AN15	Reserved	VTT	AGTL+ termination voltage	Power	

Pin #	Legacy PGA370 Pin Name	Flexible PGA370 Pin Name	Function	Type	Notes
AN21	Reserved	VTT	AGTL+ termination voltage	Power/other	4
AN23	Reserved	RP#	Request parity	AGTL+, I/O	
B36	Reserved	BINIT#	Bus initialization	AGTL+, I/O	2
C29	Reserved	DEP5#	Data bus ECC data	AGTL+, I/O	2
C31	Reserved	DEP1#	Data bus ECC data	AGTL+, I/O	2
C33	Reserved	DEP0#	Data bus ECC data	AGTL+, I/O	2
E23	Reserved	VTT	AGTL+ termination voltage	Power/other	4
E29	Reserved	DEP6#	Data bus ECC data	AGTL+, I/O	2
E31	Reserved	DEP4#	Data bus ECC data	AGTL+, I/O	2
G35	Reserved	VTT	AGTL+ termination voltage	Power/other	
G37	Reserved	See Note 5			
S33	Reserved	VTT	AGTL+ termination voltage	Power/other	4
S37	Reserved	VTT	AGTL+ termination voltage	Power/other	4
U35	Reserved	VTT	AGTL+ termination voltage	Power/other	4
U37	Reserved	VTT	AGTL+ termination voltage	Power/other	4
V4	Reserved	BERR#	Bus error	AGTL+, I/O	2
W3	Reserved	A34#	Additional AGTL+ address	AGTL+, I/O	2
X4	RESET#	RESET2#	Processor reset (used by the Intel® Celeron® processor (PPGA))	AGTL+, I	3
X6	Reserved	A32#	Additional AGTL+ address	AGTL+, I/O	2
Y33	GND	CLKREF	1.25-V PLL reference	Power	1

NOTES:

1. These signals are defined as ground (Vss) in legacy designs utilizing the PGA370 socket. For new flexible PGA370 designs, use the new signal definitions. These new signal definitions are backwards compatible with the Celeron processor (PPGA).
2. While these signals are not used with 815EP chipset designs, they are available for chipsets that do support these functions. Only the Pentium III processor (FC-PGA) offers these capabilities in the PGA370 platform.
3. The AGTL+ reset signal, RESET#, is delivered to pin X4 on legacy PGA370 designs. In flexible PGA370 designs, it is delivered to X4 and AH4 pins.
4. RESET2# is not required for platforms that do not support the Celeron processor. Pin X4 should then be connected to ground.
5. These pins must be connected to the 1.5-V VTT plane.
6. This pin must be connected to VTT for platforms using the Pentium III processor based on the cA2 stepping. Refer to the Pentium III processor specification update for stepping details.

Table 9. Processor Pin Definition Comparison

Pin #	Intel® Celeron® Processor (PPGA) Pin Name	Celeron Processor FC-PGA Pin Name	Intel® Pentium® III Processor FC-PGA Pin Name	Function
A29	Reserved	Reserved	DEP7#	Data bus ECC data
A31	Reserved	Reserved	DEP3#	Data bus ECC data
A33	Reserved	Reserved	DEP2#	Data bus ECC data
AA33	Reserved	Reserved	VTT	AGTL+ termination voltage
AA35	Reserved	Reserved	VTT	AGTL+ termination voltage
AC1	Reserved	Reserved	A33#	Additional AGTL+ address
AC37	Reserved	Reserved	RSP#	Response parity
AF4	Reserved	Reserved	A35#	Additional AGTL+ address
AH20	Reserved	Reserved	VTT	AGTL+ termination voltage
AH4	Reserved	Reserved	RESET#	Processor reset (used by the Intel® Pentium® III processor in FC-PGA)
AJ31	GND	BSEL1	BSEL1	System bus frequency select
AK16	Reserved	Reserved	VTT	AGTL+ termination voltage
AK24	Reserved	Reserved	AERR#	Address parity error
AL11	Reserved	Reserved	AP0#	Address parity
AL13	Reserved	Reserved	VTT	AGTL+ termination voltage
AL21	Reserved	Reserved	VTT	AGTL+ termination voltage
AM2	GND	Reserved	Reserved	Reserved
AN11	Reserved	Reserved	VTT	AGTL+ termination voltage
AN13	Reserved	Reserved	AP1#	Address parity
AN15	Reserved	Reserved	VTT	AGTL+ termination voltage
AN21	Reserved	Reserved	VTT	AGTL+ termination voltage
AN23	Reserved	Reserved	RP#	Request parity
B36	Reserved	Reserved	BINIT#	Bus initialization
C29	Reserved	Reserved	DEP5#	Data bus ECC data
C31	Reserved	Reserved	DEP1#	Data bus ECC data
C33	Reserved	Reserved	DEP0#	Data bus ECC data
E23	Reserved	Reserved	VTT	AGTL+ termination voltage
E29	Reserved	Reserved	DEP6#	Data bus ECC data
E31	Reserved	Reserved	DEP4#	Data bus ECC data
G35	Reserved	Reserved	VTT	AGTL+ termination voltage
S33	Reserved	Reserved	VTT	AGTL+ termination voltage
S37	Reserved	Reserved	VTT	AGTL+ termination voltage
U35	Reserved	Reserved	VTT	AGTL+ termination voltage

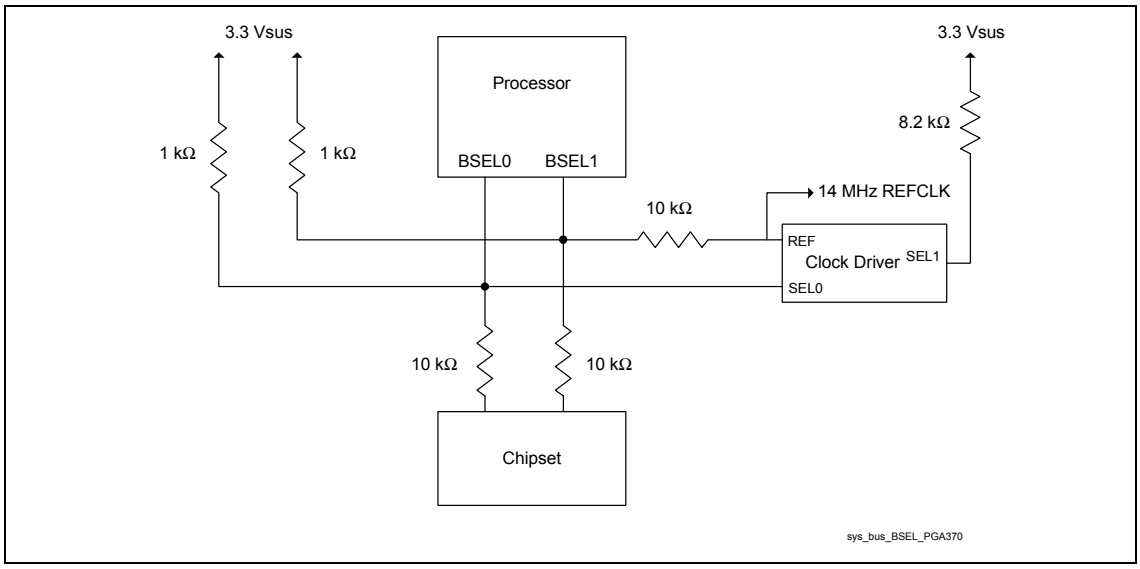
Pin #	Intel® Celeron® Processor (PPGA) Pin Name	Celeron Processor FC-PGA Pin Name	Intel® Pentium® III Processor FC-PGA Pin Name	Function
U37	Reserved	Reserved	VTT	AGTL+ termination voltage
V4	Reserved	Reserved	BERR#	Bus error
W3	Reserved	Reserved	A34#	Additional AGTL+ address
X4	RESET#	RESET#	RESET2#	Processor reset (used by the Intel® Celeron® processors)
X6	Reserved	Reserved	A32#	Additional AGTL+ address
Y33	GND	Reserved	CLKREF	1.25-V PLL reference

4.6. BSEL[1:0] Implementation Differences

A Pentium III processor in an FC-PGA utilizes the BSEL1 pin to select either the 100-MHz or 133-MHz system bus frequency setting from the clock synthesizer. While the BSEL0 signal is still connected to the PGA370 socket, a Pentium III processor in an FC-PGA does not utilize it. Only Celeron processors in a PPGA utilize the BSEL0 signal. Pentium III processors in an FC-PGA are 3.3-V tolerant for these signals, as are the clock and chipset. However, the Celeron PPGA processor utilizes 2.5-V logic levels on the BSEL signals. Therefore, *flexible PGA370* designs utilize 2.5-V logic levels on the BSEL[1:0] signals to support the widest range of processors.

CK-815 has been designed to support selections of 66 MHz, 100 MHz, and 133 MHz. The REF input pin has been redefined to be a frequency selection strap (BSEL1) during power-on and then becomes a 14-MHz reference clock output. The following figure details the new BSEL[1:0] circuit design for *flexible PGA370* designs. Note that BSEL[1:0] now are pulled up using 1-kΩ resistors. Also refer to Figure 12 for more details.

Figure 11. BSEL[1:0] Circuit Implementation for PGA370 Designs



4.7. CLKREF Circuit Implementation

The CLKREF input, utilized by the Pentium III processor (FC-PGA), requires a 1.25-V source. It can be generated from a voltage divider on the Vcc2.5 or Vcc3.3 sources utilizing 1% tolerance resistors. A 4.7-µF decoupling capacitor should be included on this input. See the following figure and the following table for example CLKREF circuits. **Do not use VTT as the source for this reference!**

Figure 12. Examples for CLKREF Divider Circuit

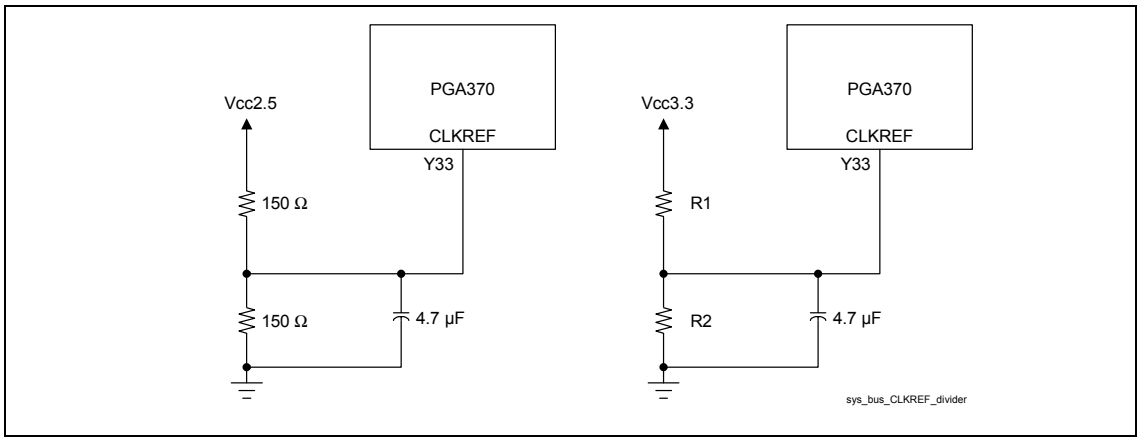


Table 10. Resistor Values for CLKREF Divider (3.3-V Source)

R1 (Ω)	R2 (Ω)	CLKREF Voltage (V)
182	110	1.243
301	182	1.243
374	221	1.226
499	301	1.242

4.8. Undershoot/Overshoot Requirements

Undershoot and overshoot specifications become more critical as the process technology for microprocessors shrink due to thinner gate oxide. Violating these undershoot and overshoot limits will degrade the life expectancy of the processor.

The Pentium III processor in FC-PGA has more restrictive overshoot and undershoot requirements for system bus signals than previous processors. These requirements stipulate that a signal at the output of the driver buffer and at the input of the receiver buffer must not exceed the maximum absolute overshoot voltage limit (2.18 V) and the minimum absolute undershoot voltage limit (-0.58 V). Exceeding these limits will damage the FC-PGA processor. There is also a time-dependent, non-linear overshoot and undershoot requirement that depends on the amplitude and duration of the overshoot/undershoot. See the appropriate Pentium III FC-PGA processor's electrical, mechanical and thermal specification for more details on the FC-PGA processor overshoot/undershoot specifications. A new undershoot/overshoot checking tool will be made available to assist in understanding whether or not simulation results or actual oscilloscope measurements meet signal integrity requirements in the datasheet.

4.9. Processor Reset Requirements

Flexible PGA370 designs must route the AGTL+ reset signal from the chipset to two pins on the processor as well as to the debug port connector. This reset signal is connected to pins AH4 (RESET#) and X4 (RESET2#) at the PGA370 socket. Finally**, the AGTL+ reset signal must always be terminated to VTT on the motherboard**.

Designs that do not support the debug port will not utilize the 240-Ω series resistor or the connection of RESET# to the debug port connector**.** RESET2# is not required for platforms that do not support the Celeron processor. Pin X4 should then be connected to ground.

The routing rules for the AGTL+ reset signal are shown in the following figure.

Figure 13. RESET#/RESET2# Routing Guidelines

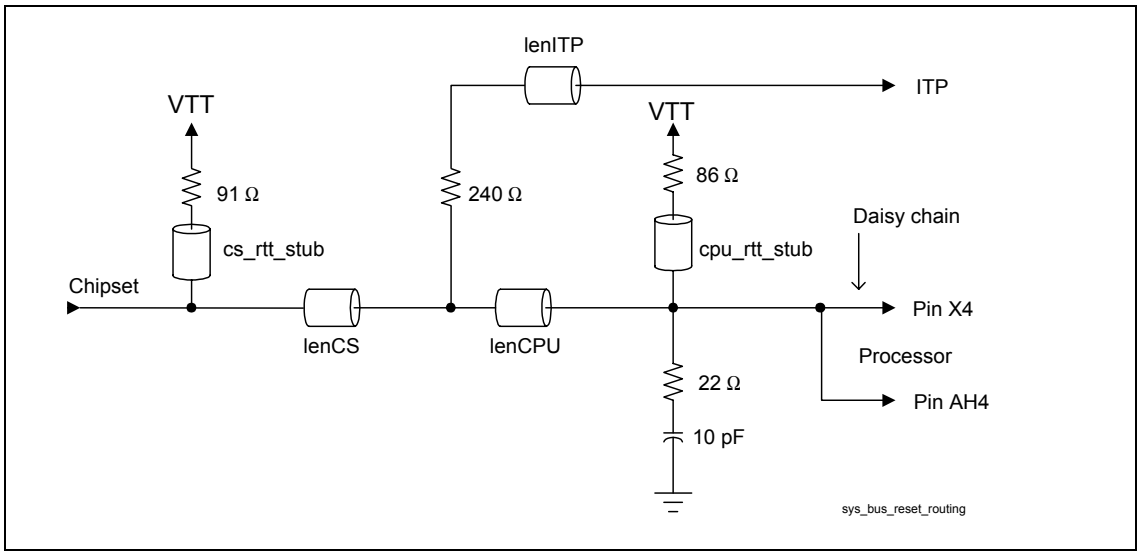


Table 11. RESET#/RESET2# Routing Guidelines (see Figure 13)

Parameter	Minimum (in)	Maximum (in)
LenCS	0.5	1.5
LenITP	1	3
LenCPU	0.5	1.5
cs_rtt_stub	0.5	1.5
cpu_rtt_stub	0.5	1.5

4.10. Determining the Processor Installed Via Hardware Mechanisms

The following table provides the logic decoding to determine which processor is installed in a PGA370 design.

Table 12. Determining the Installed Processor via Hardware Mechanisms

VID	VCORE_DET	CPUPRES#	Notes
1001	0	0	Intel® Pentium® III processor (FC-PGA) installed.
1011	0	0	Intel® Celeron® processor (FC-PGA) installed.
0001	1	0	Celeron processor (PPGA) installed.
1111	X	1	No processor installed.

4.11. Processor PLL Filter Recommendations

Intel PGA370 processors have internal phase lock loop (PLL) clock generators that are analog and require quiet power supplies to minimize jitter.

4.11.1. Topology

The general desired topology is shown in Figure 15. Not shown are the parasitic routing and local decoupling capacitors. Excluded from the external circuitry are parasitics associated with each component.

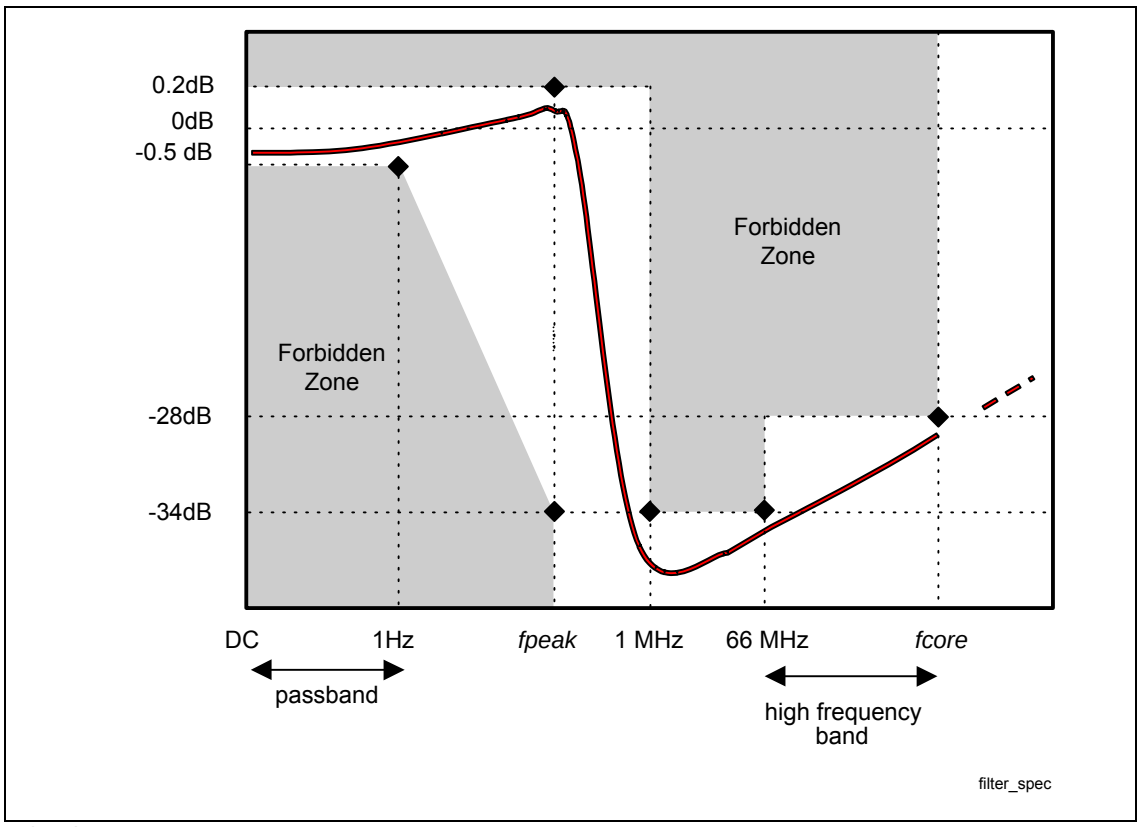
4.11.2. Filter Specification

The function of the filter is to protect the PLL from external noise through low-pass attenuation. The low-pass specification, with input at VCC_{CORE} and output measured across the capacitor, is as follows:

- < 0.2-dB gain in pass band
- < 0.5-dB attenuation in pass band (see DC drop in next set of requirements)
- > 34-dB attenuation from 1 MHz to 66 MHz
- > 28-dB attenuation from 66 MHz to core frequency

The filter specification is graphically shown in the following figure.

Figure 14. Filter Specification



NOTES:

1. Diagram not to scale.
2. No specification for frequencies beyond *fcore*.
3. *fpeak* should be less than 0.05 MHz.

Other requirements:

- Use shielded-type inductor to minimize magnetic pickup.
- Filter should support DC current > 30 mA.
- DC voltage drop from VCC to PLL1 should be < 60 mV, which in practice implies series R < 2 Ω. This also means pass-band (from DC to 1 Hz) attenuation < 0.5 dB for VCC = 1.1 V, and < 0.35 dB for VCC = 1.5 V.

4.11.3. Recommendation for Intel® Platforms

The following tables contains examples of components that meet Intel's recommendations, when configured in the topology of Figure 15.

Table 13. Component Recommendations – Inductor

Part Number	Value	Tol.	SRF	Rated I	DCR (Typical)
TDK MLF2012A4R7KT	4.7 µH	10%	35 MHz	30 mA	0.56 Ω (1 Ω max.)
Murata LQG21N4R7K00T1	4.7 µH	10%	47 MHz	30 mA	0.7 Ω (± 50%)
Murata LQG21C4R7N00	4.7 µH	30%	35 MHz	30 mA	0.3 Ω max.

Table 14. Component Recommendations – Capacitor

Part Number	Value	Tolerance	ESL	ESR
Kemet T495D336M016AS	33 µF	20%	2.5 nH	0.225 Ω
AVX TPSD336M020S0200	33 µF	20%	2.5 nH	0.2 Ω

Table 15. Component Recommendations – Resistor

Value	Tolerance	Power	Note
1 Ω	10%	1/16 W	Resistor may be implemented with trace resistance, in which case a discrete R is not needed.

To satisfy damping requirements, total series resistance in the filter (from VCC_{CORE} to the top plate of the capacitor) must be at least 0.35 Ω. This resistor can be in the form of a discrete component or routing or both. For example, if the picked inductor has minimum DCR of 0.25 Ω, then a routing resistance of at least 0.10 Ω is required. Be careful not to exceed the maximum resistance rule (2 Ω). For example, if using discrete R1 (1 Ω ± 1%), the maximum DCR of the L (trace plus inductor) should be less than 2.0 - 1.1 = 0.9 Ω, which precludes the use of some inductors and sets a max. trace length.

Other routing requirements:

- The capacitor (C) should be close to the PLL1 and PLL2 pins, < 0.1 Ω per route[1].
- The PLL2 route should be parallel and next to the PLL1 route (i.e., minimize loop area).
- The inductor (L) should be close to C. Any routing resistance should be inserted between VCC_{CORE} and L.
- Any discrete resistor (R) should be inserted between VCC_{CORE} and L.

[1] These routes do not count towards the minimum damping R requirement.

Figure 15. Example PLL Filter Using a Discrete Resistor

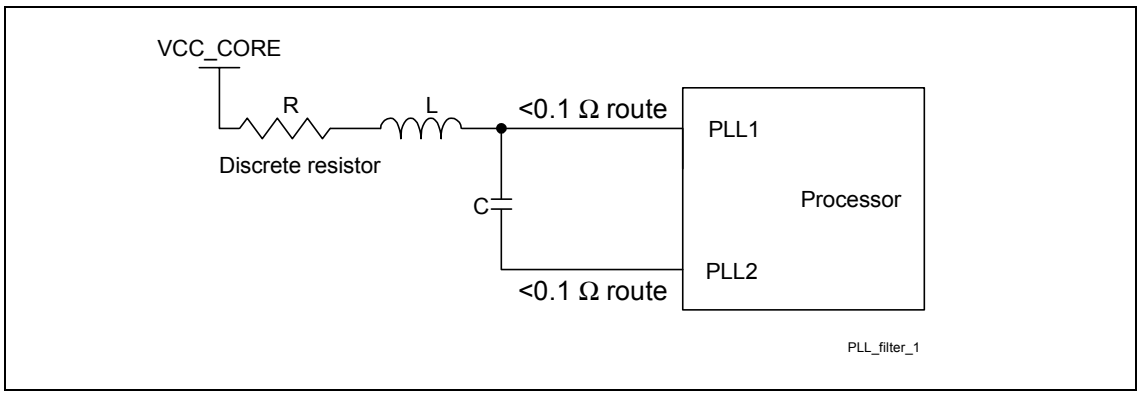
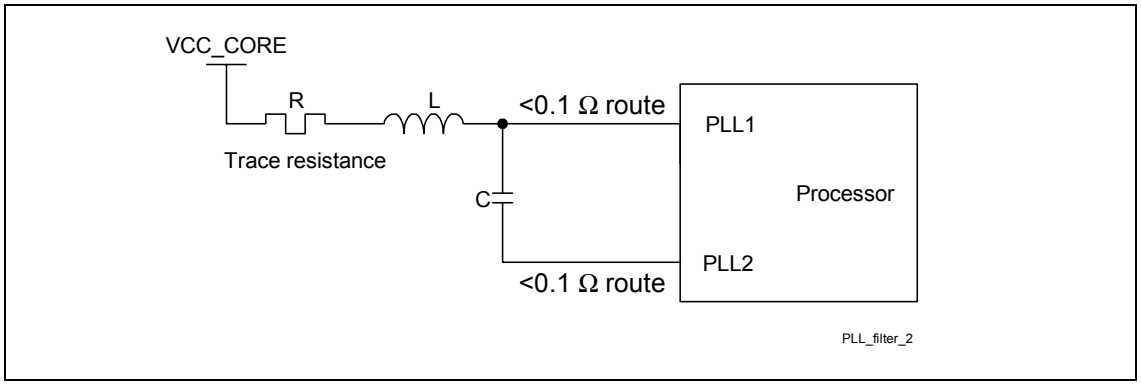


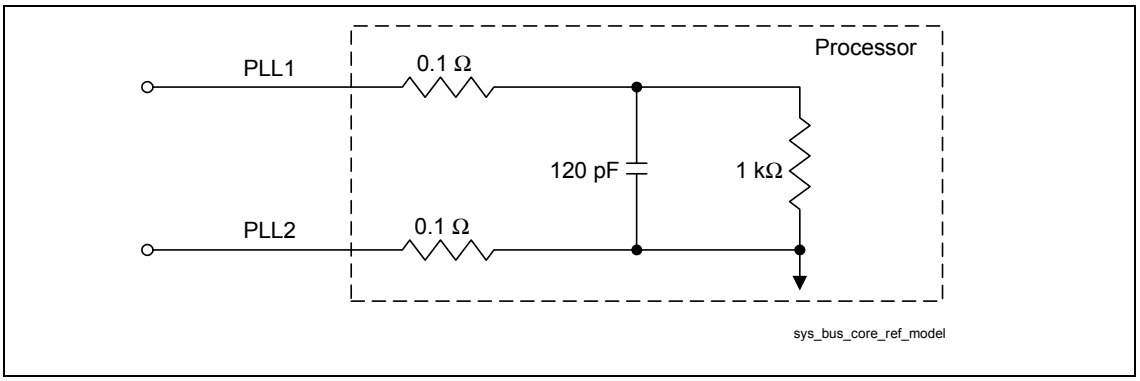
Figure 16. Example PLL Filter Using a Buried Resistor



4.11.4. Custom Solutions

As long as filter performance and requirements as specified and outlined in Section 4.11.2 are satisfied, other solutions are acceptable. Custom solutions should be simulated against a standard reference core model, which is shown in the following figure.

Figure 17. Core Reference Model



NOTES:

1. 0.1-Ω resistors represent package routing[2].
2. 120-pF capacitor represents internal decoupling capacitor.
3. 1-kΩ resistor represents small signal PLL resistance.
4. Be sure to include all component and routing parasitics.
5. Sweep across component/parasitic tolerances.
6. To observe IR drop, use DC current of 30 mA and minimum Vcc_{CORE} level.
7. For other modules (interposer, DMM, etc), adjust routing resistor if desired, but use minimum numbers.

4.12. Voltage Regulation Guidelines

A *flexible PGA370* design will need the voltage regulation module (VRM) or on-board voltage regulator (VR) to be compliant with Intel *VRM 8.4 DC-DC Converter Design Guidelines*, Rev. 1.5 or higher. This is needed to support the power supply requirements of the Pentium III processor (FC-PGA) at speeds greater than 650 MHz. Important points to note regarding VRM 8.4 specifications are as follows:

- The VR/VRM must supply the proper Vcc_{CORE} voltage to the processor, as indicated by the VID outputs.
- Transient and static tolerances are tighter in the *VRM 8.4 DC-DC Converter Design Guidelines* than in the *VRM 8.2 DC-DC Converter Design Guidelines*. This will require additional analysis of the motherboard power delivery solution.
- Maximum current for Vcc_{CORE} has increased to 18.4A for flexible motherboard designs.
- Additional motherboard decoupling for the processor power supplies is needed to meet VRM 8.4 DC-DC Converter Design Guidelines, Ver. 1.5.

4.13. Decoupling Guidelines for Flexible PGA370 Designs

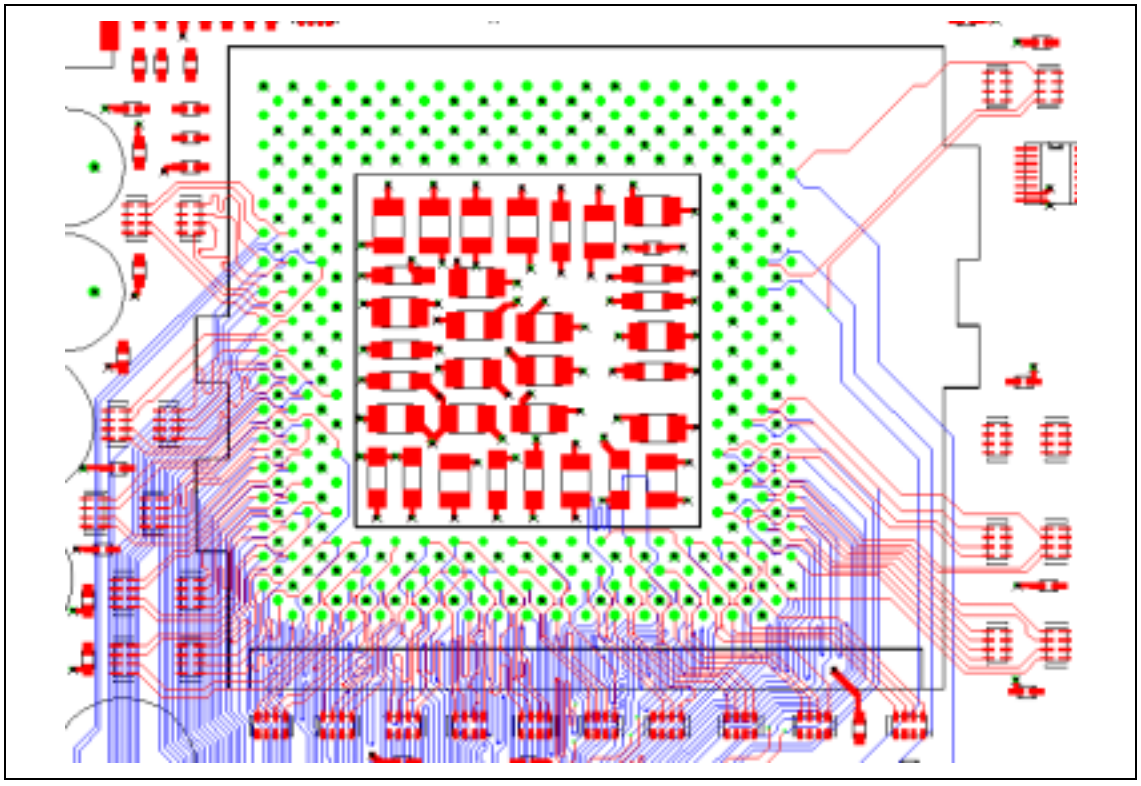
These preliminary decoupling guidelines for *flexible PGA370* designs are estimated to meet the specifications of *VRM 8.4 DC-DC Converter Design Guidelines*, Ver. 1.5 (Vcc = 1.6 V, Icc = 0.8–18.4 A).

4.13.1. Vcc_{CORE} Decoupling Design

- **Ten** or more 4.7-µF capacitors in 1206 packages.

All capacitors should be placed within the PGA370 socket cavity and mounted on the primary side of the motherboard. The capacitors are arranged to minimize the overall inductance between the Vcc_{CORE}/Vss power pins, as shown in the following figure.

Figure 18. Capacitor Placement on the Motherboard



4.13.2. V_{TT} Decoupling Design

For Itt = 3.0 A (max.)

- ***Nineteen*** 0.1-µF capacitors in 0603 packages placed within 200 mils of AGTL+ termination R-packs, with one capacitor for every two R-packs. These capacitors are shown on the exterior of the previous figure.

4.13.3. V_{REF} Decoupling Design

- ***Four*** 0.1-µF capacitors in 0603 package placed near V_{REF} pins (within 500 mils).

4.14. Thermal/EMI Considerations

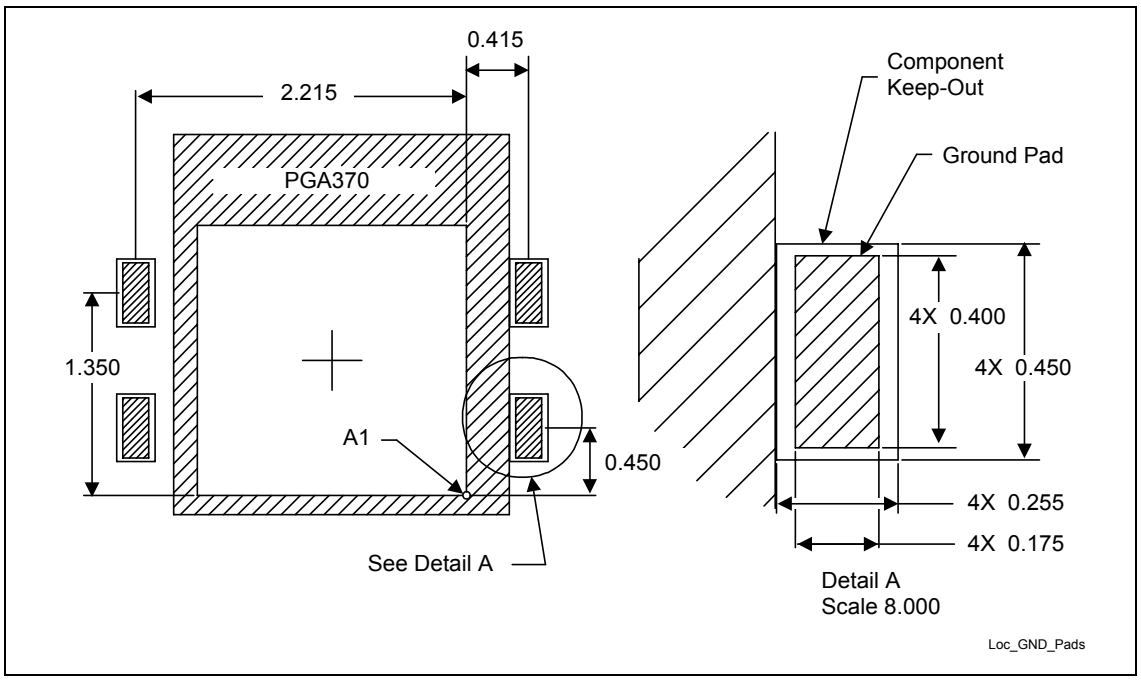
Flexible motherboard guideline for the Pentium III processor (FC-PGA) calls for 30.4 W.

- Increased power density for the Pentium III processor in FC-PGA (FMB = 41.9 W/cm^2)
- Pentium III processors in FC-PGA are specified using Tj, while PPGA processors are specified using Tcase.
- Heatsink for FC-PGA package is not compatible with PPGA processors
- New heatsink clips for FC-PGA processor heatsinks
- Option to add motherboard features to ground the processor heatsink to reduce electromagnetic interference (EMI)

4.14.1. Implementation of Optional Grounded Heatsink for EMI Reduction

The following figure illustrates the concept of providing the processor heatsink with an AC ground return path. Experiments at Intel have demonstrated improved EMI emissions with prototypes of this solution. Further details will be provided in the next revision of this design guide.

Figure 19. Location of Grounding Pads



4.14.2. Heatsink Volumetric Keep-Out Regions

Figure 20 shows the system component keep-out volume above the socket connector required for the reference design thermal solution for high frequency FC-PGA processors. This keep-out envelope provides adequate room for the heatsink, fan and attach hardware under static conditions as well as room for installation of these components on the socket.

Figure 21 shows component keep-outs on the motherboard required to prevent interference with the reference design thermal solution. Note portions of the heatsink and attach hardware hang over the motherboard.

Adhering to these keep-out areas will ensure compatibility with Intel boxed processor products and Intel enabled third party vendor thermal solutions for FC-PGA processors. While the keep-out requirements should provide adequate space for the reference design thermal solution, systems integrators should check their vendor to ensure their specific thermal solutions fit within their specific system designs. Please ensure that the thermal solutions under analysis comprehend the specific thermal design requirements for higher frequency Pentium III processors.

While thermal solutions for lower frequency FCPGA processors may not require the full keep-out area, larger thermal solutions will be required for higher frequency processors and failure to adhere to the guidelines will result in mechanical interference.

Figure 20. Heatsink Volumetric Keep-Out Regions

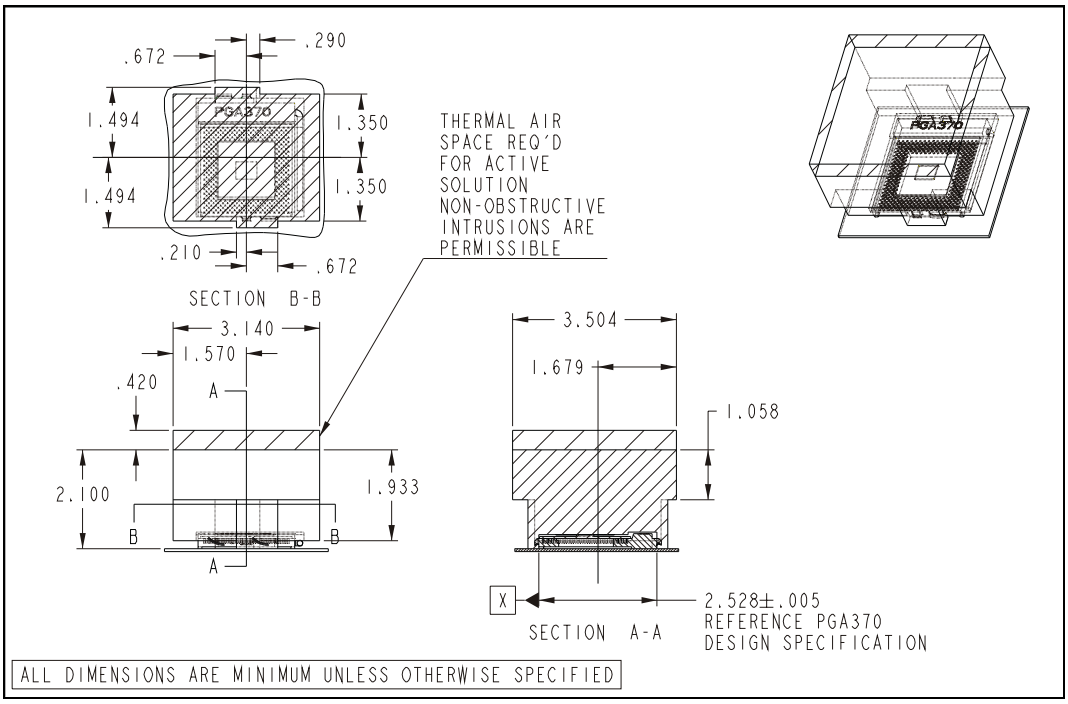
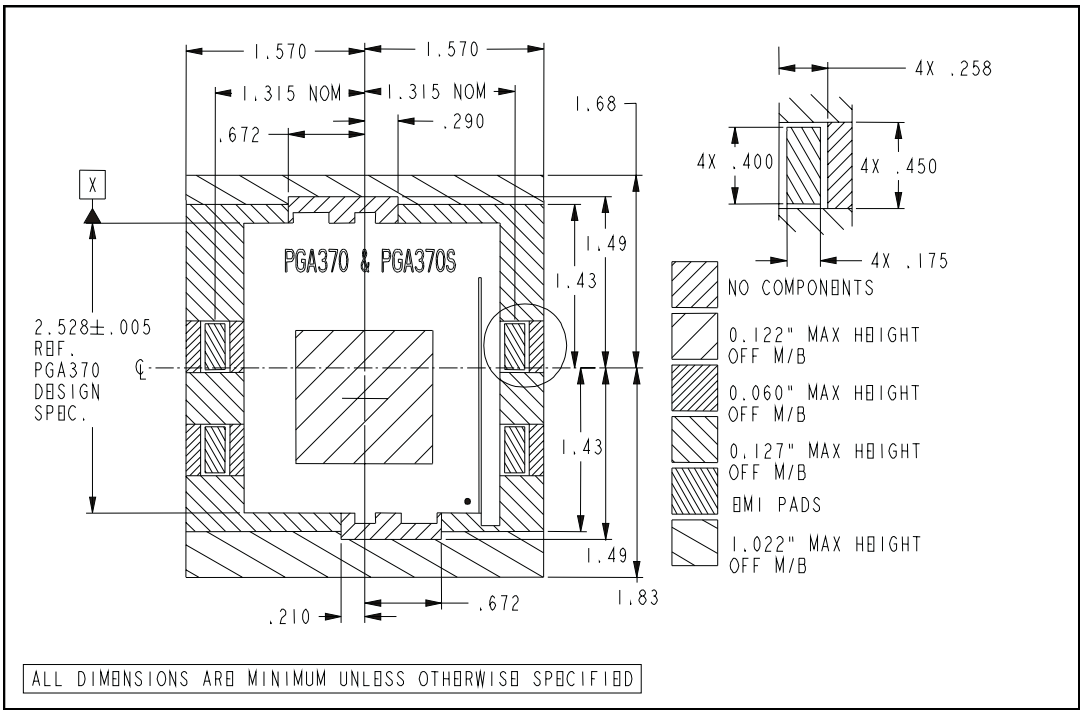


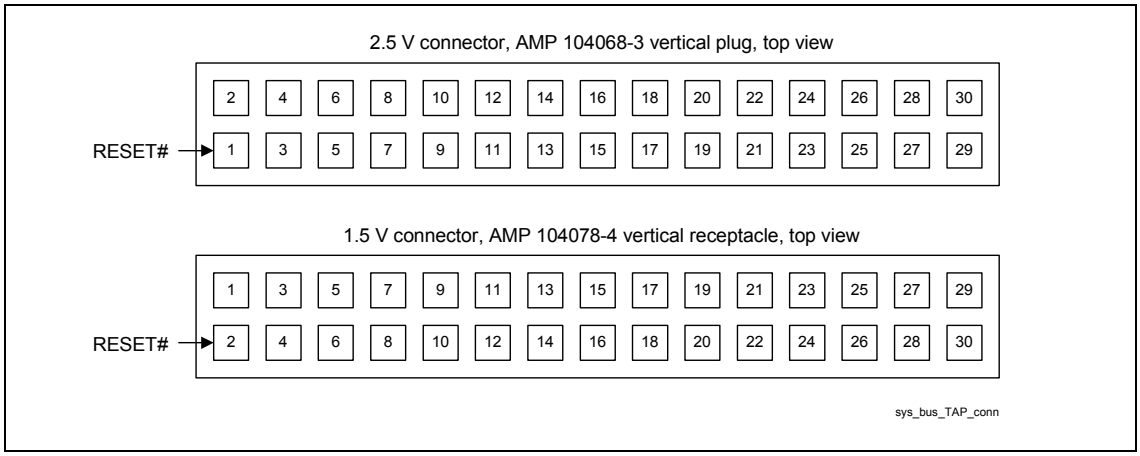
Figure 21. Motherboard Component Keep-Out Regions



4.15. Debug Port Changes

Due to the lower voltage technology employed with the FC-PGA processor, changes are required to support the debug port. Previously, test access port (TAP) signals used 2.5-V logic, as is the case with the Celeron processor in the PPGA package. FC-PGA processors utilize 1.5-V logic levels on the TAP. As a result, a new debug port connector is to be used in *flexible PGA370* designs. The new 1.5-V connector is the mirror image of the older 2.5-V connector. Either connector will fit into the same printed circuit board layout. Only the pin numbers would change, as is evident in the following drawing. Also required, along with the new connector, is an In-Target Probe (ITP) that is capable of communicating with the TAP at 1.5-V logic levels.

Figure 22. TAP Connector Comparison



Caution: FC-PGA processors require an in-target probe (ITP) compatible with 1.5-V signal levels on the TAP. Previous ITPs were designed to work with higher voltages and may damage the processor if connected to an FC-PGA processor.

See the processor datasheet for more information regarding the debug port.

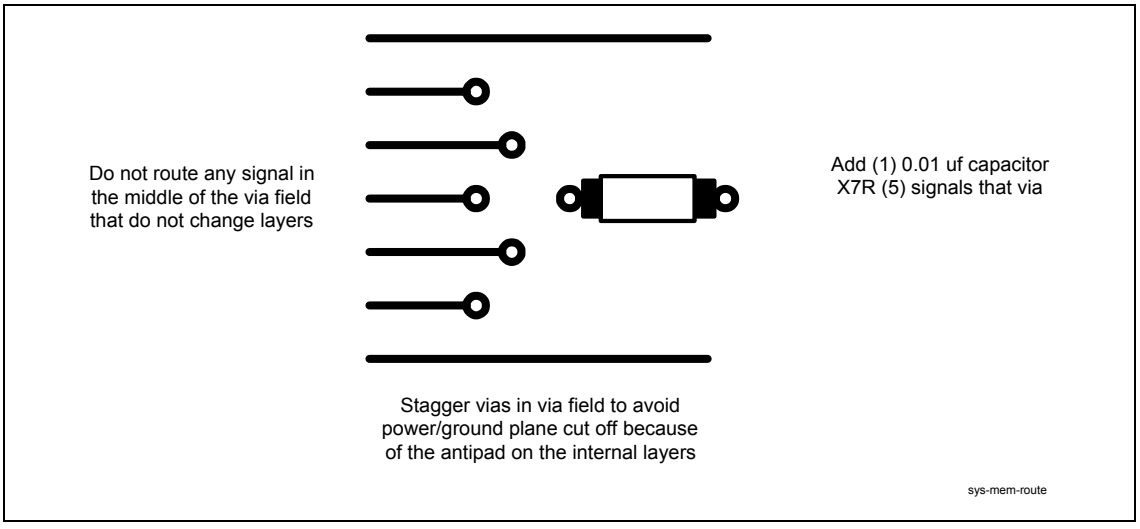
5. System Memory Design Guidelines

5.1. System Memory Routing Guidelines

Ground plane reference all system memory signals. To provide a good current return path and limit noise on the system memory signals, the signals should be ground referenced from the MCH to the DIMM connectors and from DIMM connector to DIMM connector. If ground referencing is not possible, system memory signals should be, at a minimum, referenced to a single plane. If single plane referencing is not possible, stitching capacitors should be added no more than 200 mils from the signal via field. System memory signals may via to the backside of the PCB under the MCH without a stitching capacitor as long as the trace on the topside of the PCB is less than 200 mils. Note that it is recommended that a parallel plate capacitor between VCC3.3SUS and GND be added to account for the current return path discontinuity (See Decoupling section). Use (1) .01uf X7R capacitor per every (5) system memory signals that switch plane references. No more than two vias are allowed on any system memory signal.

If a group of system memory signals need to change layers, a via field should be created and a decoupling capacitor should be added at the end of the via field. Do not route signals in the middle of a via field, this will cause noise to be generated on the current return path of these signals and can lead to issues on these signals. See diagram below. The traces shown are on layer 1 only. The diagram shows signals that are changing layer and two signals that are not changing layer. Note the two signals around the via field create a keep out zone where no signals that do not change layer should be routed.

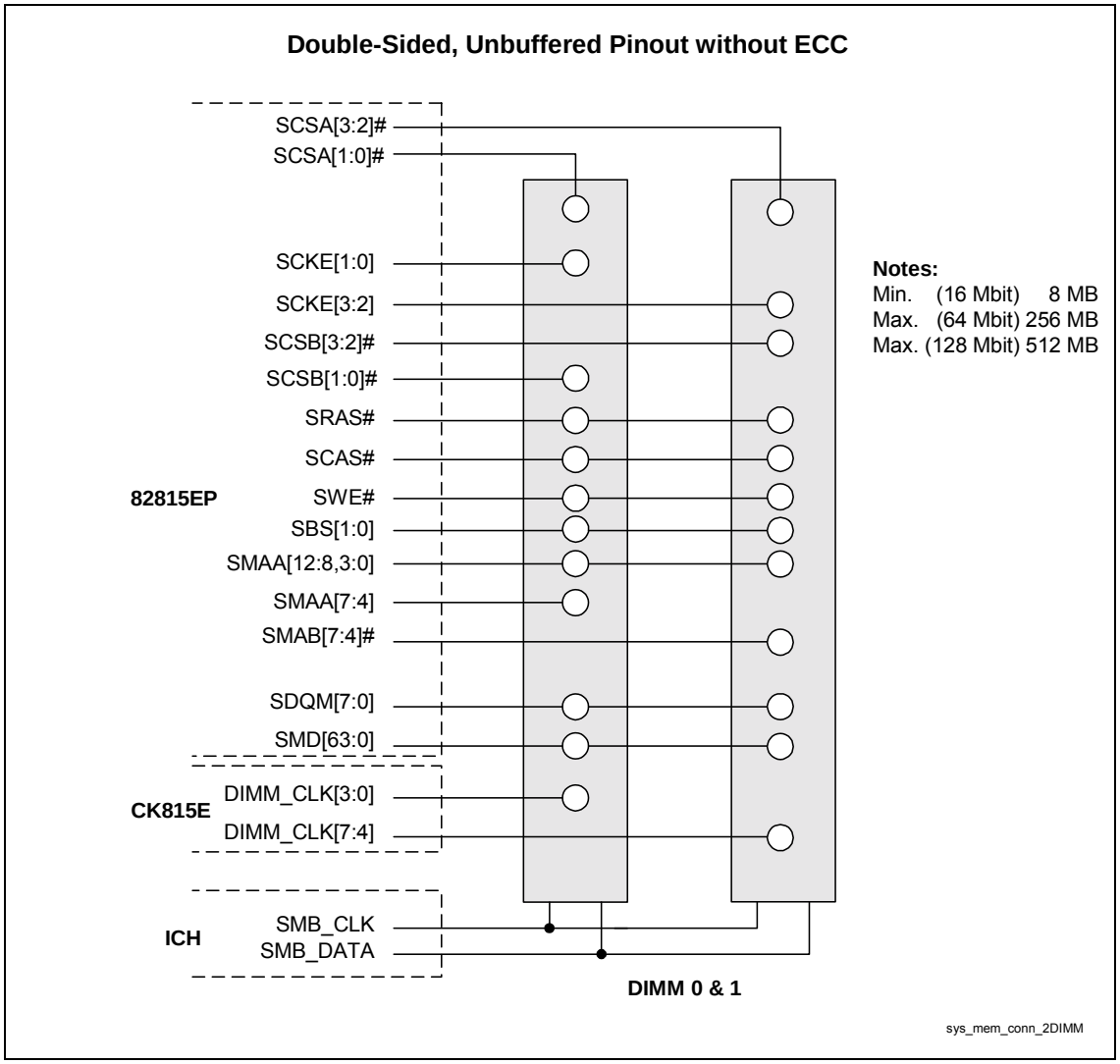
Figure 23. System Memory Routing Guidelines



5.2. System Memory 2-DIMM Design Guidelines

5.2.1. System Memory 2-DIMM Connectivity

Figure 24. System Memory Connectivity (2 DIMM)



5.2.2. System Memory 2-DIMM Layout Guidelines

Due to the lower voltage technology employed with the FC-PGA processor, changes are required to support the debug port. Previously, test access port (TAP) signals used 2.5-V logic, as is the case with the Celeron processor in the PPGA package. FC-PGA processors utilize 1.5-V logic levels on the

Figure 25. System Memory 2-DIMM Routing Topologies

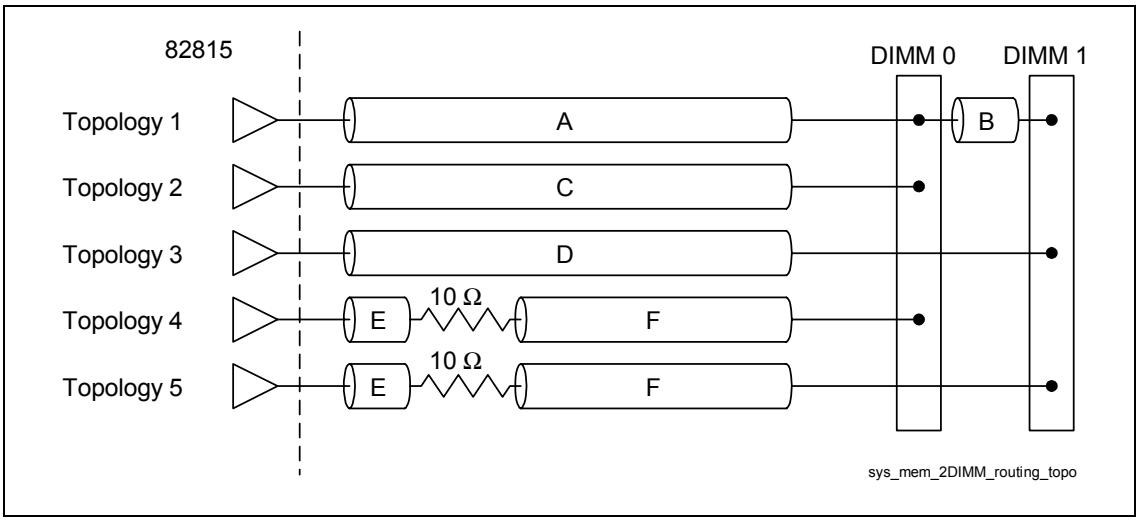
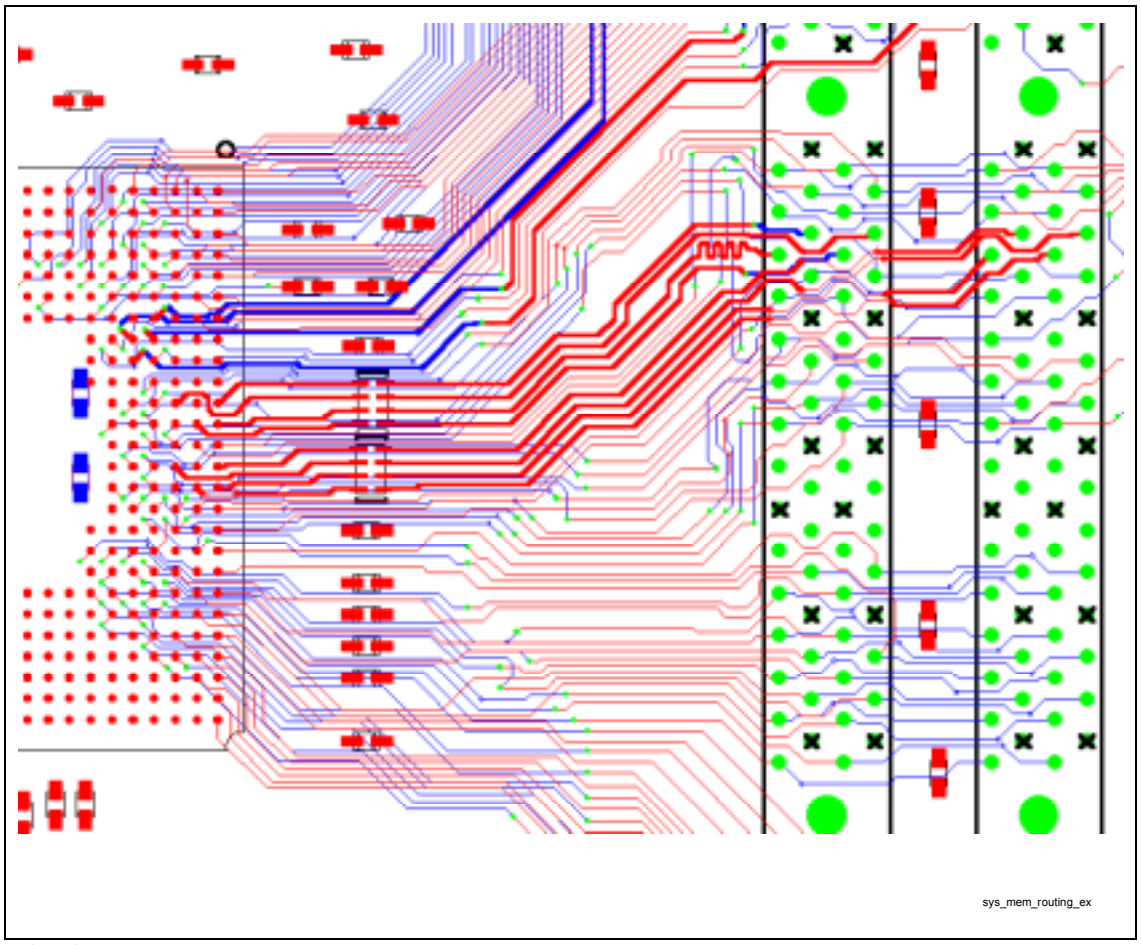


Table 16. System Memory 2-DIMM Solution Space

Signal	Top.	Trace (mils)		Trace Lengths (inches)											
				A		B		C		D		E		F	
		Width	Spacing	Min.	Max.	Min.	Max.	Min.	Max.	Min.	Max.	Min.	Max.	Min.	Max.
SCS[3:2]#	3	5	10							1	4.5				
SCS[1:0]#	2	5	10					1	4.5						
SMAA[7:4]	4	10	10									0.4	0.5	2	4
SMAB[7:4]#	5	10	10									0.4	0.5	2	4
SCKE[3:2]	3	10	10							3	4				
SCKE[1:0]	2	10	10					3	4						
SMD[63:0]	1	5	10	1.75	4	0.4	0.5								
SDQM[7:0]	1	10	10	1.5	3.5	0.4	0.5								
SCAS#, SRAS#, SWE#	1	5	10	1	4.0	0.4	0.5								
SBS[1:0], SMAA[12:8,3:0]	1	5	10	1	4.0	0.4	0.5								

In addition to meeting the spacing requirements outlined in Table 16, system memory signal trace edges must be at least 30 mils from any other non-system memory signal trace edge.

Figure 26. System Memory Routing Example



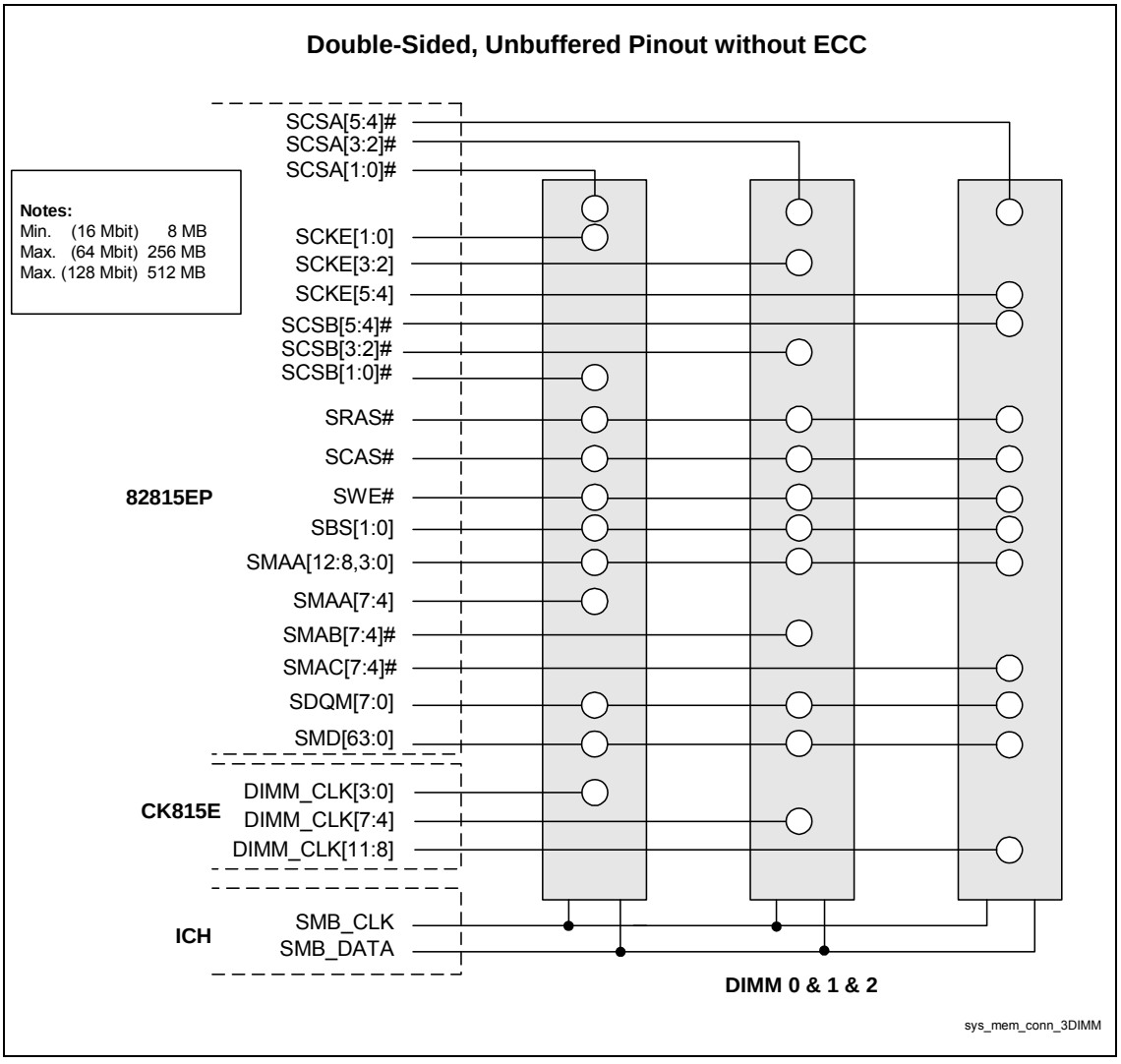
NOTES:

1. Routing in this figure is for example purposes only. It does not necessarily represent complete and correct routing for this interface.

5.3. System Memory 3-DIMM Design Guidelines

5.3.1. System Memory 3-DIMM Connectivity

Figure 27. System Memory Connectivity (3 DIMM)



5.3.2. System Memory 3-DIMM Layout Guidelines

Figure 28. System Memory 3-DIMM Routing Topologies

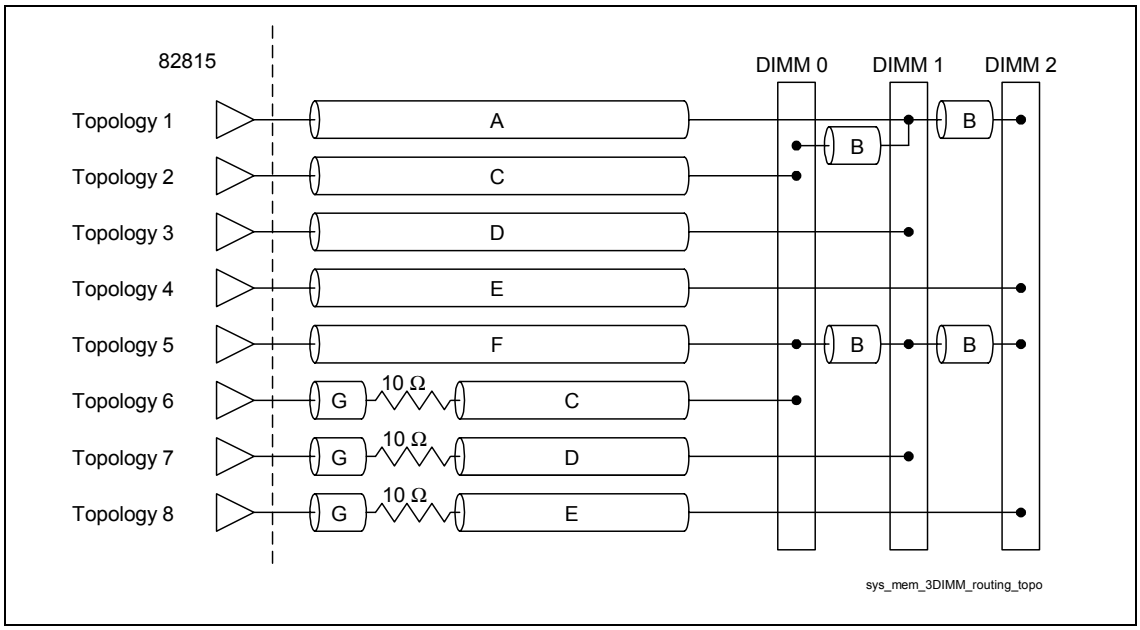


Table 17. System Memory 3-DIMM Solution Space

Signal		Trace (mils)		Trace Lengths (inches)													
				A		B		C		D		E		F		G	
	Top.	Width	Spacing	Min.	Max.	Min.	Max.	Min.	Max.	Min.	Max.	Min.	Max.	Min.	Max.	Min.	Max.
SCS[5:4]#	4	5	10									1	4.5				
SCS[3:2]#	3	5	10							1	4.5						
SCS[1:0]#	2	5	10					1	4.5								
SMAA[7:4]	6	10	10					2	4							0.4	0.5
SMAB[7:4]#	7	10	10							2	4					0.4	0.5
SMAC[7:4}	8	10	10									2	4			0.4	0.5
SCKE[5:4]	4	10	10									3	4				
SCKE[3:2]	3	10	10							3	4						
SCKE[1:0]	2	10	10					3	4								
SMD[63:0]	1	5	10	1.75	4	0.4	0.5										
SDQM[7:0]	1	10	10	1.5	3.5	0.4	0.5										
SCAS#, SRAS#, SWE#	5	5	10			0.4	0.5							1	4		
SBS[1:0], SMAA[12:8,3:0]	5	5	10			0.4	0.5							1	4		

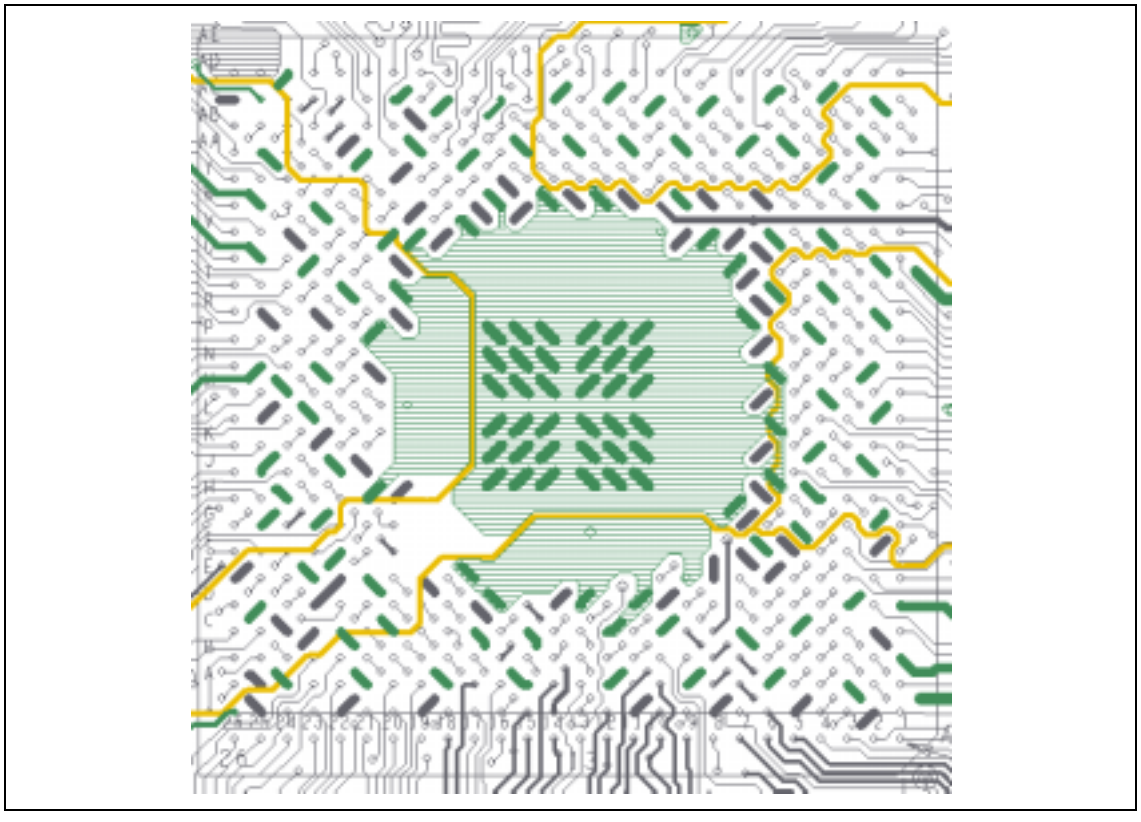
In addition to meeting the spacing requirements outlined in Table 17, system memory signal trace edges must be at least 30 mils from any other non-system memory signal trace edge.

5.4. System Memory Decoupling Guidelines

A minimum of eight 0.1-µF low-ESL ceramic capacitors (e.g., 0603 body type, X7R dielectric) are required and must be as close as possible to the MCH. They should be placed within at most 70 mils to the edge of the MCH package edge for VSUS_3.3 decoupling, and they should be evenly distributed around the system memory interface signal field including the side of the MCH where the system memory interface meets the host interface. There are power and GND balls throughout the system memory ball field of the MCH that need good local decoupling. Make sure to use at least 14 mil drilled vias and wide traces from the pads of the capacitor to the power or ground plane to create a low inductance path. If possible multiple vias per capacitor pad are recommended to further reduce inductance. To add the decoupling capacitors within 70 mils of the MCH and/or close to the vias, the trace spacing may be reduced as the traces go around each capacitor. The narrowing of space between traces should be minimal and for as short a distance as possible (500mils max).

To further de-couple the MCH and provide a solid current return path for the system memory interface signals it is recommended that a parallel plate capacitor be added under the MCH. Add a topside or bottom side copper flood under center of the MCH to create a parallel plate capacitor between VCC3.3 and GND, See following figure. The dashed lines indicate power plane splits on layer 2 or layer 3 depending on stack-up. The filled region in the middle of the MCH indicates a ground plate (on layer 1 if the power plane is on layer 2 or on layer 4 if the power layer is on layer 3).

Figure 29. Intel® 815 Chipset Decoupling Example

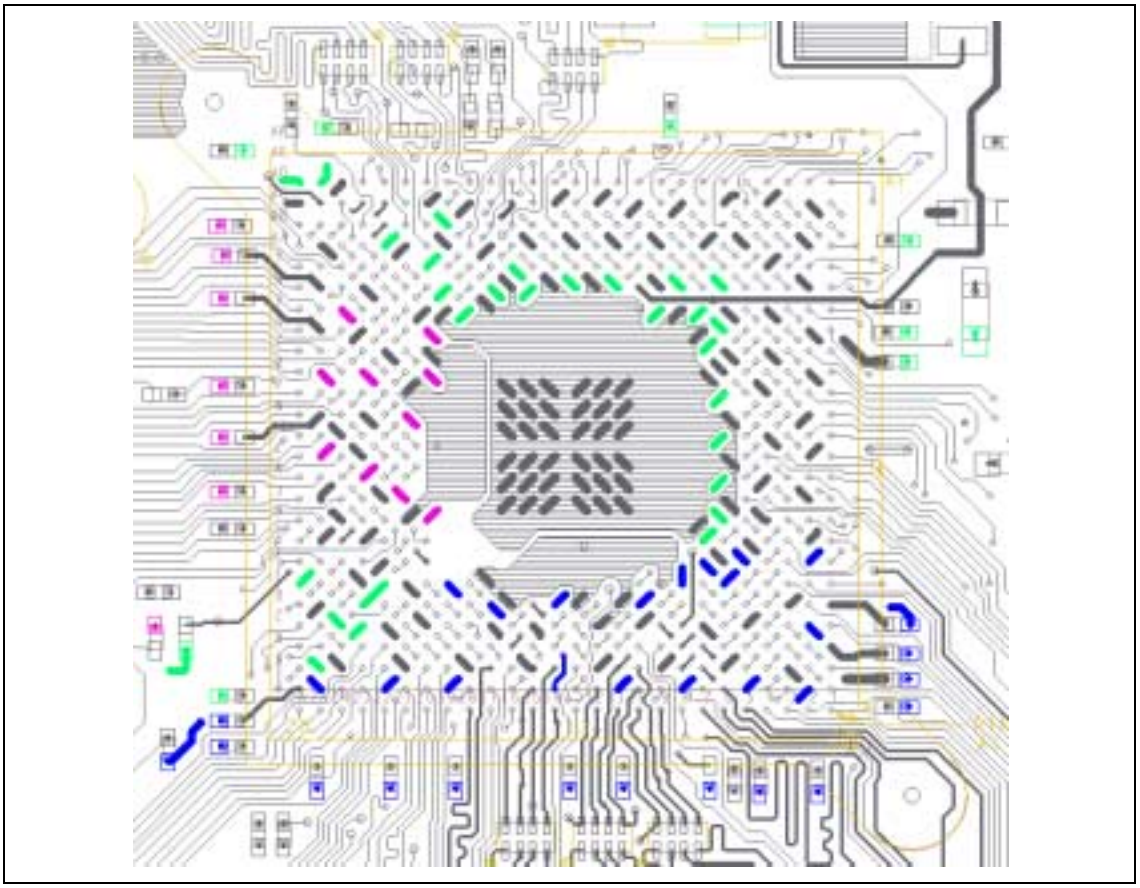


Yellow lines show layer two plane splits. Note that the layer 1 shapes do NOT cross the plane splits. The southern shape is a Vss fill over VddSDRAM. The western shape is a Vss fill over VddAGP. The larger northeastern shape is a Vss fill over VddCORE.

Additional decoupling capacitors should be added between the DIMM connectors to provide a current return path for the reference plane discontinuity created by the DIMM connectors themselves. (1) .01uf X7R capacitor should be added per every (10) SDRAM signals. Capacitors should be placed between the DIMM connectors and evenly spread out across the SDRAM interface.

For debug purposes, four or more 0603 capacitor sites should be placed on the backside of the board, evenly distributed under the 815 chipset's system memory interface signal field.

Figure 30. Intel® 815 Chipset Decoupling Example



5.5. Compensation

A system memory compensation resistor (SRCOMP) is used by the MCH to adjust the buffer characteristics to specific board and operating environment characteristics. Refer to the *Intel® 815 Chipset Family: Graphics and Memory Controller Hub (MCH) Datasheet* for details on compensation. Tie the SRCOMP pin of the MCH to a 40-Ω, 1% or 2% pull-up resistor to 3.3 Vsus (3.3-V standby) via a 10-mil-wide, 0.5" trace (targeted for a nominal impedance of 40 Ω).

This page is intentionally left blank.

6. AGP Design Guidelines

For the detailed AGP interface functionality (e.g., protocols, rules, signaling mechanisms), refer to the latest *AGP Interface Specification*, *Revision 2.0*, which can be obtained from http://www.agpforum.org. This design guide focuses only on specific 815EP chipset platform recommendations.

6.1. AGP Interface

A single AGP connector is supported by the MCH AGP interface. LOCK# and SERR#/PERR# are not supported. See the display cache discussion for a description of display cache/AGP muxing as well as a description of the AGP In-Line Memory Module (AIMM).

The AGP buffers operate in one of two selectable modes, to support the AGP universal connector:

1. 3.3-V drive, not 5-V safe. This mode is compliant with the AGP 1.0 66-MHz specification
2. 1.5-V drive, not 3.3-V safe. This mode is compliant with the AGP 2.0 specification

The AGP 4X must operate at 1.5 V. The AGP 2X can operate at 3.3 V or 1.5 V. The AGP interface supports up to 4X AGP signaling, though 4X fast writes are not supported. AGP semantic cycles to DRAM are not snooped on the host bus.

The MCH supports PIPE# or SBA[7:0] AGP address mechanisms, but not both simultaneously. Either the PIPE# or the SBA[7:0] mechanism must be selected during system initialization. The MCH contains a 32-deep AGP request queue. High-priority accesses are supported. All AGP semantic accesses hitting the graphics aperture pass through an address translation mechanism with a fully-associative, 20-entry TLB.

Accesses between AGP and the hub interface are limited to hub interface-originated memory writes to AGP. Cacheable accesses from the IOQ queue flow through one path, while aperture accesses follow another path. Cacheable AGP (SBA, PIPE# and FRAME#) reads to DRAM all snoop the cacheable global write buffer (GWB) for system data coherency. Aperture AGP (SBA, PIPE#) reads to DRAM snoop the aperture queue (GCMCRWQ). Aperture AGP (FRAME#) reads and writes to DRAM proceed through a FIFO and there is no RAW capability, so no snoop is required.

The AGP interface is clocked from the 66-MHz clock (3V66). The AGP-to-host/memory interface is synchronous with a clock ratio of 1:1 (66 MHz : 66 MHz), 2:3 (66 MHz : 100 MHz) and 1:2 (66 MHz : 133 MHz).

6.1.1. AGP Universal Retention Mechanism (RM)

Environmental testing and field reports indicate that external AGP cards may come unseated during system shipping and handling without proper retention. To avoid disengaged AGP, Intel recommends that AGP based platforms use the AGP retention mechanism (RM).

The AGP RM is a mounting bracket that is used to properly locate the card with respect to the chassis and to assist with card retention. The AGP RM is available in two different handle orientations: left-

handed (see Figure 31) and right-handed. Most system boards accommodate the left-handed AGP RM. The manufacturing capacity of the left-handed RM currently exceeds the right-handed capacity, and as a result Intel recommends that customers design their systems to insure they can use the left-handed version of the AGP RM (see Figure 32). The right-handed AGP RM is identical to the left-handed AGP RM, except for the position of the actuation handle. This handle is located on the same end as the primary design, but extends from the opposite side (mirrored about the center axis running parallel to the length of the part). Figure 32 contains keep out information for the left hand AGP retention mechanism. Use this information to make sure that your motherboard design leaves adequate space to install the retention mechanism.

The AGP interconnect design requires that the AGP card must be retained to the extent that the card not back out more than 0.99 mm (0.039 in) within the AGP connector. To accomplish this it is recommended that new cards implement an additional notch feature in the mechanical keying tab to allow an anchor point on the AGP card for interfacing with an AGP RM. The retention mechanism's round peg engages with the AGP card's retention tab and prevents the card from disengaging during dynamic loading. The additional notch feature in the mechanical keying tab is required for 1.5-volt AGP cards and is recommended for the new 3.3-volt AGP cards.

Figure 31. AGP Left-Handed Retention Mechanism

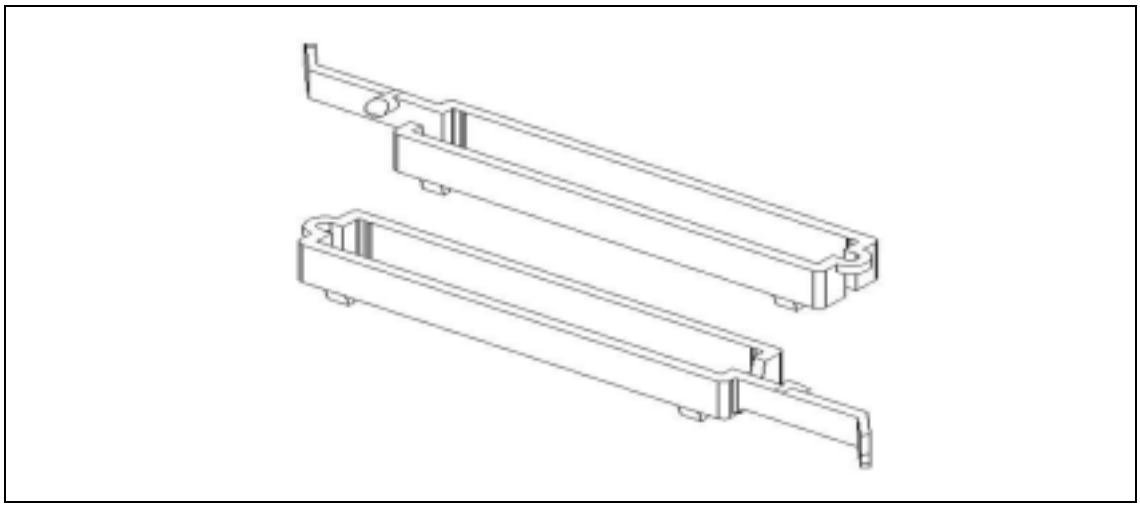
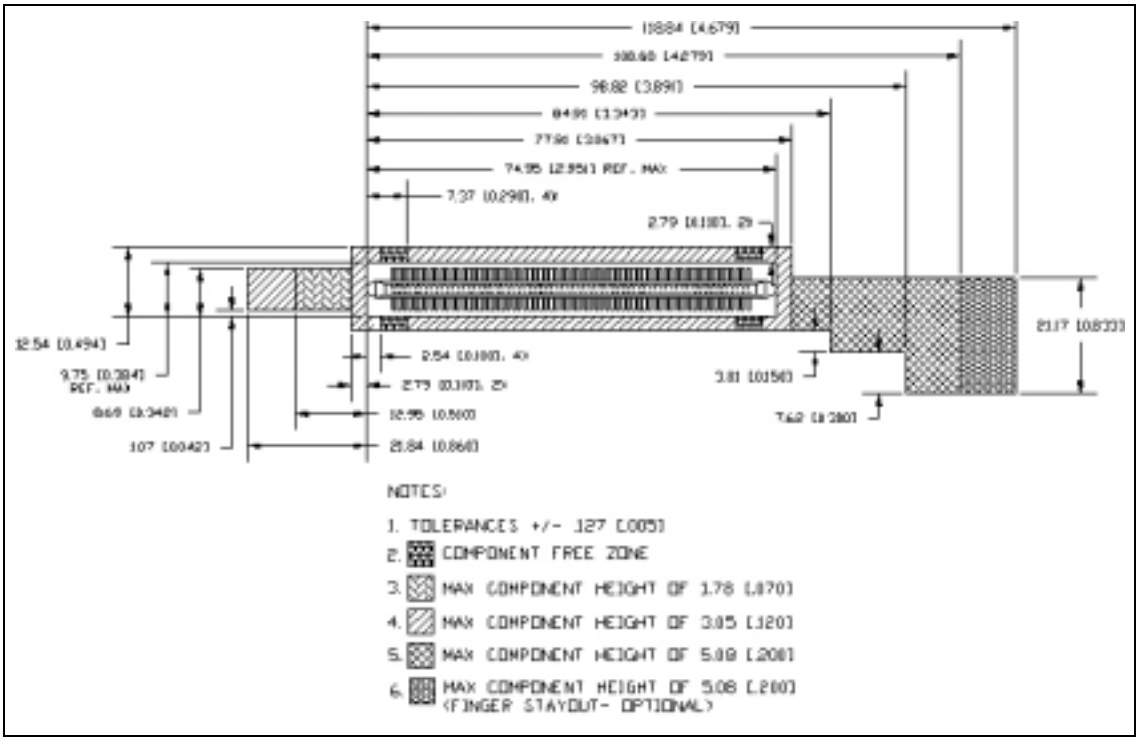


Figure 32. AGP Left-Handed Retention Mechanism Keep-Out Information



Engineering Change Request number 48 (ECR #48) of the AGP specification details the AGP RM, which is recommended for all AGP cards. These are approved changes to the Accelerated Graphics Port (AGP) Interface Specification, Revision 2.0. Intel intends to incorporate the AGP RM changes into later revisions of the AGP Interface Specification. In addition, Intel has defined a reference design of a mechanical device to utilize the features defined in ECR #48.

ECR #48 can be viewed off the Intel Website at:

http://developer.intel.com/technology/agp/ecr.htm

More information regarding this component (AGP RM) is available from the following vendors.

Resin Color	Supplier Part Number	"Left Handed" Orientation (Preferred)	"Right Handed" Orientation (Alternate)
Black	AMP P/N	136427-1	136427-2
	Foxconn P/N	006-0002-939	006-0001-939
Green	Foxconn P/N	009-0004-008	009-0003-008

Note: These vendors, devices are listed by Intel as a convenience to Intel's general customer base, but Intel does not make any representations or warranties whatsoever regarding quality, reliability, functionality, or compatibility of these devices. This list and/or these devices may be subject to change without notice.

6.2. AGP 2.0

Rev. 2.0 of the AGP Interface Specification enhances the functionality of the original (Rev. 1.0) AGP Interface Specification, by allowing 4X data transfers (4 data samples per clock) and 1.5-V operation. The 4X operation of the AGP interface provides for "quad-pumping" of the AGP AD (address/data) and SBA (side-band addressing) buses. That is, data is sampled four times during each 66-MHz AGP clock, which means that each data cycle is ¼ of a 15-ns (66-MHz) clock, or 3.75 ns. It is important to realize that 3.75 ns is the data cycle time, not the clock cycle time. During 2X operation, data is sampled twice during a 66-MHz clock cycle, so the data cycle time is 7.5 ns. To allow for such high-speed data transfers, the 2X mode of AGP operation uses source-synchronous data strobing. During 4X operation, the AGP interface uses differential source-synchronous strobing.

With data cycle times as small as 3.75 ns and setup/hold times of 1 ns, propagation delay mismatch is critical. In addition to reducing propagation delay mismatch, it is important to minimize noise. Noise on the data lines causes the settling time to be long. If the mismatch between a data line and the associated strobe is too great or if there is noise on the interface, incorrect data will be sampled. The low-voltage operation on the AGP (1.5 V) requires even more noise immunity. For example, during 1.5-V operation, $V_{il_{max}}$ is 570 mV. Without proper isolation, crosstalk could create signal integrity issues.

6.2.1. AGP Interface Signal Groups

The signals on the AGP interface are broken into three groups: *1X timing domain signals*, *2X/4X timing domain signals*, and *miscellaneous signals*. Each group has different routing requirements. In addition, within the *2X/4X timing domain signals*, there are three sets of signals. All signals in the 2X/4X timing domain must meet minimum and maximum trace length requirements as well as trace width and spacing requirements. However, trace length matching requirements only must be satisfied within each set of 2X/4X timing domain signals. The signal groups are listed in the following table.

Table 18. AGP 2.0 Signal Groups

Groups	Signal
1X Timing Domain	CLK (3.3 V), RBF#, WBF#, ST[2:0], PIPE#, REQ#, GNT#, PAR, FRAME#, IRDY#, TRDY#, STOP#, DEVSEL#
2X/4X Timing Domain	**Set #1:** AD[15:0], C/BE[1:0]#, AD_STB0, AD_STB0#[1] **Set #2:** AD[31:16], C/BE[3:2]#, AD_STB1, AD_STB1#[1] **Set #3:** SBA[7:0], SB_STB, SB_STB#[1]
Miscellaneous, async.	USB+, USB-, OVRCNT#, PME#, TYPDET#, PERR#, SERR#, INTA#, INTB#

NOTES:
1. These signals are used in 4X AGP mode ONLY.

Table 19. AGP 2.0 Data/Strobe Associations

Data	Associated Strobe in 1X	Associated Strobe in 2X	Associated Strobes in 4X
AD[15:0] and C/BE[1:0]#	Strobes are not used in 1X mode. All data is sampled on rising clock edges.	AD_STB0	AD_STB0, AD_STB0#
AD[31:16] and C/BE[3:2]#	Strobes are not used in 1X mode. All data is sampled on rising clock edges.	AD_STB1	AD_STB1, AD_STB1#
SBA[7:0]	Strobes are not used in 1X mode. All data is sampled on rising clock edges.	SB_STB	SB_STB, SB_STB#

Throughout this section the term *data* refers to AD[31:0], C/BE[3:0]#, and SBA[7:0]. The term *strobe* refers to AD_STB[1:0], AD_STB[1:0]#, SB_STB, and SB_STB#. When the term *data* is used, it refers to one of the three sets of data signals, as seen in Table 18. When the term *strobe* is used, it refers to one of the strobes as it relates to the data in its associated group.

The routing guidelines for each group of signals (*1X timing domain signals*, *2X/4X timing domain signals*, *miscellaneous signals)* will be addressed separately.

6.3. AGP Routing Guidelines

6.3.1. 1X Timing Domain Routing Guidelines

6.3.1.1. External AGP Card Motherboard Guidelines

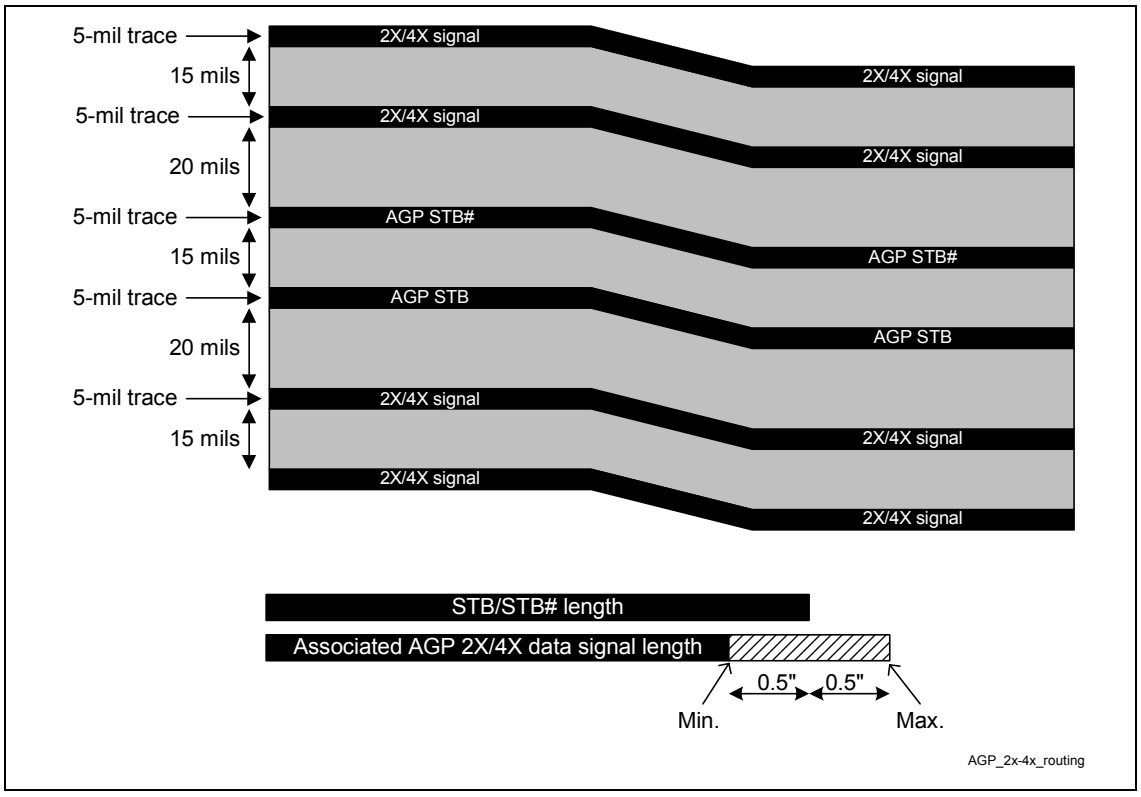
- AGP 1X timing domain signals (Table 18) have a maximum trace length of 7.5” for motherboards that will support an AGP card. This maximum applies to ALL signals listed as 1X timing domain signals in Table 18.
- All AGP 1X timing domain signals can be routed with 5-mil minimum trace separation.
- There are no trace length matching requirements for 1X timing domain signals.

6.3.2. 2X/4X Timing Domain Routing Guidelines

These trace length guidelines apply to ALL signals listed in Table 18 as 2X/4X timing domain signals. These signals should be routed using 5-mil (60-Ω) traces.

The maximum line length and length mismatch requirements depend on the routing rules used on the motherboard. These routing rules were created to provide design freedom by making trade-offs between signal coupling (trace spacing) and line lengths. The maximum length of the AGP interface defines which set of routing guidelines must be used. Guidelines for short AGP interfaces (e.g., < 6”) and long AGP interfaces (e.g., > 6” and < 7.25”) are documented separately. The maximum length allowed for the AGP interface (on external AGP card motherboards) is 7.25”.

Figure 33. AGP 2X/4X Routing Example for Interfaces < 6 Inches



6.3.2.1. External AGP Card Motherboard Guidelines

For motherboards that will use an external AGP card in the AGP slot, the maximum AGP 2X/4X signal trace length is 7.25”. However, there are different guidelines for AGP interfaces shorter than 6” (e.g., all AGP 2X/4X signals are shorter than 6”) and those longer than 6” but shorter than the 7.25” maximum.

AGP Interfaces Shorter Than 6”

The following guidelines are for designs that require less than 6” between the AGP connector and the MCH:

- 1:3 trace width-to-spacing is required for AGP 2X/4X timing domain signal traces.
- AGP 2X/4X signals must be matched with their associated strobe (as outlined in Table 18), within ±0.5”.

For example, if a set of strobe signals (e.g., AD_STB0 and AD_STB0#) are 5.3” long, the data signals associated with those strobe signals (e.g., AD[15:0] and C/BE[2:0]#) can be 4.8” to 5.8” long. Another strobe set (e.g., SB_STB and SB_STB#) could be 4.2” long, and the data signals associated with those strobe signals (e.g., SBA[7:0]) could be 3.7” to 4.7” long.

The strobe signals (AD_STB0, AD_STB0#, AD_STB1, AD_STB1#, SB_STB, and SB_STB#) act as clocks on the source-synchronous AGP interface. Therefore, special care must be taken when routing these signals. Because each strobe pair is truly a differential pair, the pair should be routed together (e.g., AD_STB0 and AD_STB0# should be routed next to each other). The two strobes in a strobe pair should be routed on 5-mil traces with at least 15 mils of space (1:3) between them. This pair should be separated from the rest of the AGP signals (and all other signals) by at least 20 mils (1:4). The strobe pair

must be length-matched to less than ±0.1". (That is, a strobe and its complement must be the same length, within 0.1".) Refer to Table 18 for an illustration of these requirements.

AGP Interfaces Longer Than 6"

Since longer lines have more crosstalk, they require wider spacing between traces to reduce the skew. The following guidelines are for designs that require more than 6" (but less than the 7.25" max.) between the AGP connector and the MCH:

- 1:4 trace width-to-spacing is required for AGP 2X/4X timing domain signal traces.
- AGP 2X/4X signals must be matched with their associated strobe (as outlined in Table 18), within ±0.125".

For example, if a set of strobe signals (e.g., AD_STB0 and AD_STB0#) are 6.5" long, the data signals associated with those strobe signals (e.g., AD[15:0] and C/BE[2:0]#) can be 6.475" to 6.625" long. Another strobe set (e.g., SB_STB and SB_STB#) could be 6.2" long, and the data signals associated with those strobe signals (e.g., SBA[7:0]) could be 6.075" to 6.325" long.

The strobe signals (AD_STB0, AD_STB0#, AD_STB1, AD_STB1#, SB_STB, and SB_STB#) act as clocks on the source-synchronous AGP interface. Therefore, special care must be taken when routing these signals. Because each strobe pair is truly a differential pair, the pair should be routed together (e.g., AD_STB0 and AD_STB0# should be routed next to each other). The two strobes in a strobe pair should be routed on 5-mil traces with at least 20 mils of space (1:4) between them. This pair should be separated from the rest of the AGP signals (and all other signals) by at least 20 mils (1:4). The strobe pair must be length-matched to less than ±0.1". (i.e., a strobe and its complement must be the same length, within 0.1".)

6.3.3. AGP Routing Guideline Considerations and Summary

This section applies to all AGP signals in any motherboard support configuration (e.g., "external AGP card"):

- The 2X/4X timing domain signals can be routed with 5-mil spacing when breaking out of the MCH. The routing must widen to the documented requirements within 0.3" of the MCH package.
- When matching trace lengths for the AGP 4X interface, all traces should be matched from the ball of the MCH to the pin on the AGP connector. It is not necessary to compensate for the length of the AGP signals on the MCH package.
- Reduce line length mismatch to ensure added margin. The trace length mismatch for all signals within a signal group should be as close as possible to zero, to provide timing margin.
- To reduce trace-to-trace coupling (i.e., crosstalk), separate the traces as much as possible.
- All signals in a signal group should be routed on the same layer.
- The trace length and trace spacing requirements ***must*** not be violated by any signal.

Table 20. AGP 2.0 Routing Summary

Signal	Maximum Length	Trace Spacing (5-Mil Traces)	Length Mismatch	Relative To	Notes
1X Timing Domain	7.5" [4]	5 mils	No requirement	N/A	None
2X/4X Timing Domain Set 1	7.25" [4]	20 mils	±0.125"	AD_STB0 and AD_STB0#	AD_STB0 and AD_STB0# must be the same length.
2X/4X Timing Domain Set 2	7.25" [4]	20 mils	±0.125"	AD_STB1 and AD_STB1#	AD_STB1 and AD_STB1# must be the same length.
2X/4X Timing Domain Set 3	7.25"[4]	20 mils	±0.125"	SB_STB and SB_STB#	SB_STB and SB_STB# must be the same length.
2X/4X Timing Domain Set 1	6" [3]	15 mils[1]	±0.5"	AD_STB0 and AD_STB0#	AD_STB0 and AD_STB0# must be the same length.
2X/4X Timing Domain Set 2	6"[3]	15 mils[1]	±0.5"	AD_STB1 and AD_STB1#	AD_STB1 and AD_STB1# must be the same length.
2X/4X Timing Domain Set 3	6"[3]	15 mils[1]	±0.5"	SB_STB and SB_STB#	SB_STB and SB_STB# must be the same length.

NOTES:
1. Each strobe pair must be separated from other signals by at least 20 mils.
2. These guidelines apply to board stack-ups with 15% impedance tolerance.
3. 4" is the maximum length for flexible motherboards.
4. Solution valid for external AGP card motherboards

6.3.4. AGP Clock Routing

The maximum total AGP clock skew, between the MCH and the graphics component, is 1 ns for all data transfer modes. This 1 ns includes skew and jitter that originates on the motherboard, add-in card, and clock synthesizer. Clock skew must be evaluated not only at a single threshold voltage, but at all points on a clock edge that falls within in the switching range. The 1-ns skew budget is divided such that the motherboard is allotted 0.9 ns of clock skew. (The motherboard designer shall determine how the 0.9 ns is allocated between the board and the synthesizer.)

For the 815EP chipset's platform AGP clock routing guidelines, refer to Section 9.3.

6.3.5. AGP Signal Noise Decoupling Guidelines

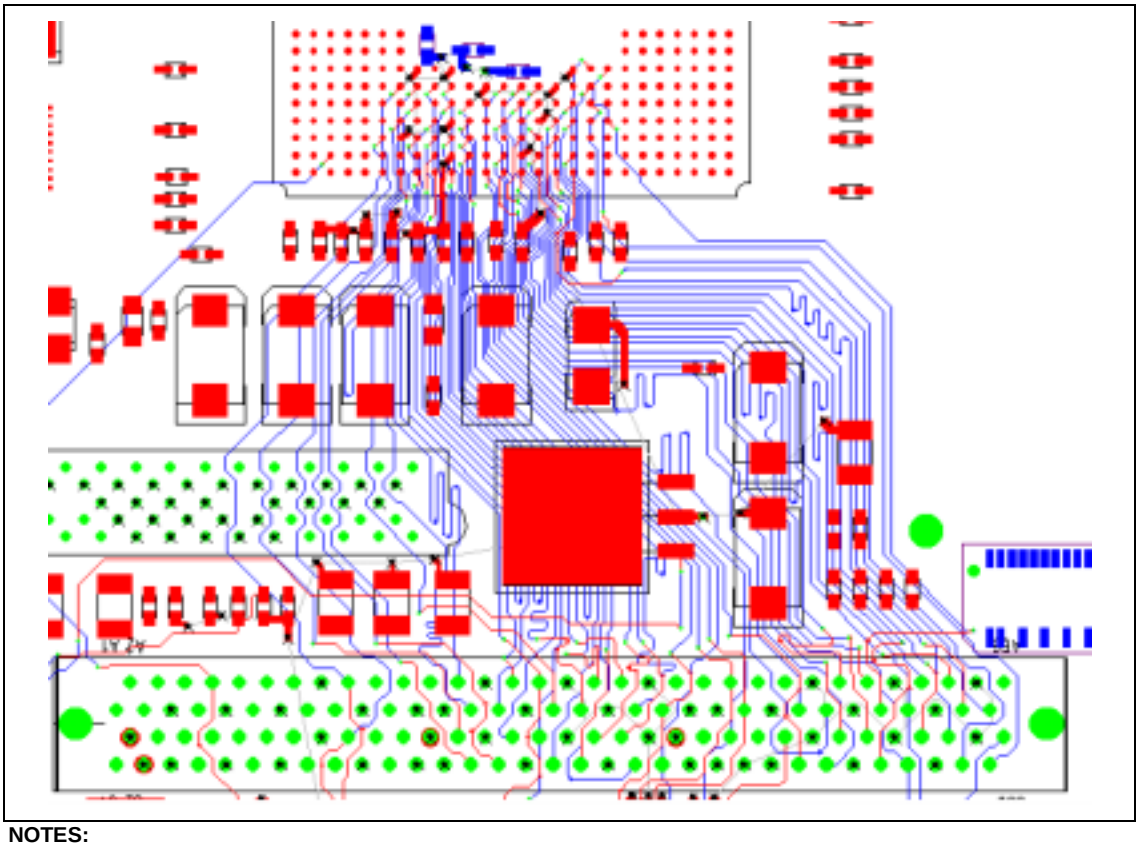
The following routing guidelines are recommended for an optimal system design. The main focus of these guidelines is to minimize signal integrity problems on the AGP interface of the MCH. The following guidelines are not intended to replace thorough system validation of 815EP chipset-based products:

- A minimum of six 0.01-µF capacitors are required and must be as close as possible to the MCH. These should be placed within 70 mils of the outer row of balls on the MCH for VDDQ decoupling. The closer the placement, the better.

- The designer should evenly distribute the placement of decoupling capacitors within the AGP interface signal field.
- It is recommended that the designer use a low-ESL ceramic capacitor, such as a 0603 body-type X7R dielectric
- To add the decoupling capacitors within 70 mils of the MCH and/or close to the vias, the trace spacing may be reduced as the traces go around each capacitor. The narrowing of the space between traces should be minimal and for as short a distance as possible (1" max.).
- In addition to the minimum decoupling capacitors, the designer should place bypass capacitors at vias that transition the AGP signal from one reference signal plane to another. *In a typical four-layer PCB design, the signals transition from one side of the board to the other.* One extra 0.01-µF capacitor is required per 10 vias. The capacitor should be placed as close as possible to the center of the via field.

The designer should ensure that the AGP connector is well decoupled, as described in the *Rev. 1.0 AGP Design Guide*, Section 1.5.3.3.

Figure 34. AGP Decoupling Capacitor Placement Example



NOTES:
1. This figure is for example purposes only. It does not necessarily represent complete and correct routing for this interface.

6.3.6. AGP Routing Ground Reference

It is strongly recommended that, at a minimum, the following critical signals be referenced to ground from the MCH to an AGP connector (or to an AGP video controller if implemented as a "down" solution), using a minimum number of vias on each net: AD_STB0, AD_STB0#, AD_STB1, AD_STB1#, SB_STB, SB_STB#, G_GTRY#, G_IRDY#, G_GNT#, and ST[2:0].

In addition to the minimum signal set listed previously, it is strongly recommended that half of all AGP signals be reference to ground, depending on board layout. In an ideal design, the entire AGP interface signal field would be referenced to ground. This recommendation is not specific to any particular PCB stack-up, but should be applied to all 815EP chipset designs.

6.4. AGP 2.0 Power Delivery Guidelines

6.4.1. VDDQ Generation and TYPEDET#

AGP specifies two separate power planes: VCC and VDDQ. VCC is the core power for the graphics controller. This voltage is **always** 3.3 V. VDDQ is the interface voltage. In AGP 1.0 implementations, VDDQ also was 3.3 V. For the designer developing an AGP 1.0 motherboard, there is no distinction between VCC and VDDQ, since both are tied to the 3.3-V power plane on the motherboard.

AGP 2.0 requires that these power planes be separate. In conjunction with the 4X data rate, the AGP 2.0 Interface Specification provides for low-voltage (1.5-V) operation. The AGP 2.0 specification implements a TYPEDET# (type detect) signal on the AGP connector that determines the operating voltage of the AGP 2.0 interface (VDDQ). The motherboard must provide either 1.5 V or 3.3 V to the add-in card, depending on the state of the TYPEDET# signal (see the following table). 1.5-V low-voltage operation applies **only** to the AGP interface (VDDQ). Vcc is always 3.3 V.

Note: The motherboard provides 3.3 V to the Vcc pins of the AGP connector. If the graphics controller needs a lower voltage, then the add-in card must regulate the 3.3-Vcc voltage to the controller's requirements. The graphics controller may **only** power AGP I/O buffers with the VDDQ power pins.

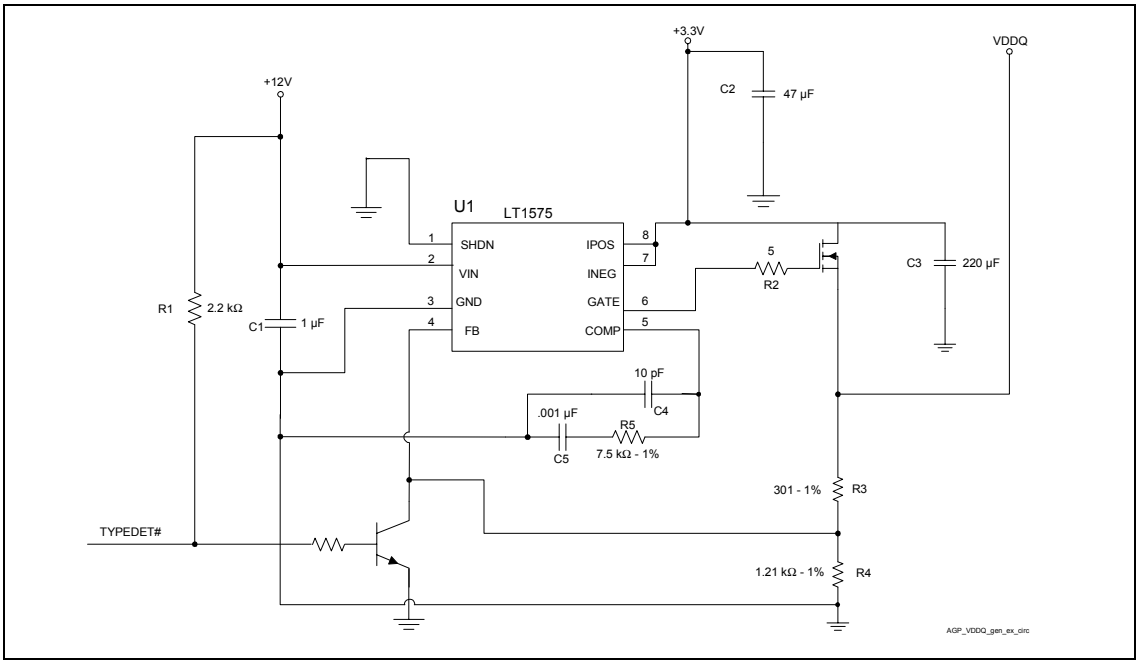
The TYPEDET# signal indicates whether the AGP 2.0 interface operates at 1.5 V or 3.3 V. If TYPEDET# is floating (i.e., No Connect) on an AGP add-in card, the interface is 3.3 V. If TYPEDET# is shorted to ground, the interface is 1.5 V.

Table 21. TYPDET#/VDDQ Relationship

TYPEDET# (on Add-in Card)	VDDQ (Supplied by MB)
GND	1.5 V
N/C	3.3 V

As a result of this requirement, the motherboard must provide a *flexible* voltage regulator or key the slot to preclude add-in cards with voltage requirements incompatible with the motherboard. This regulator must supply the appropriate voltage to the VDDQ pins on the AGP connector. For specific design recommendations, refer to the schematics in Section 12. VDDQ generation and AGP V_{REF} generation must be considered together. Before developing VDDQ generation circuitry, refer to Section 6.4.1 and the *AGP 2.0 Interface Specification*.

Figure 35. AGP VDDQ Generation Example Circuit



The previous figure demonstrates **one** way to design the VDDQ voltage regulator. This regulator is a linear regulator with an external, low-Rds_{on} FET. The source of the FET is connected to 3.3 V. This regulator converts 3.3 V to 1.5 V or passes 3.3 V, depending on the state of TYPEDET#. If a linear regulator is used, it must draw power from 3.3 V (not 5 V) to control thermals. (That is, 5 V regulated down to 1.5 V with a linear regulator will dissipate approximately 7 W at 2 A.) Because it must draw power from 3.3 V and, in some situations, must simply pass that 3.3 V to VDDQ (when a 3.3-V add-in card is placed in the system), the regulator MUST use a low-Rds_{on} FET.

AGP 1.0 ECR #44 modified VDDQ 3.3_{min} to 3.1 V. When an ATX power supply is used, the 3.3 V_{min} is 3.168. Therefore, 68 mV of drop is allowed across the FET at 2 A. This corresponds to an FET with an Rds_{on} of 34 mΩ.

How does the regulator switch? The feedback resistor divider is set to 1.5 V. When a 1.5-V card is placed in the system, the transistor is Off and the regulator regulates to 1.5 V. When a 3.3-V card is placed in the system, the transistor is On, and the feedback will be pulled to ground. When this happens, the regulator will drive the gate of the FET to nearly 12 V. This will turn on the FET and pass 3.3 V – 2 A * Rds_{on} to VDDQ.

6.4.2. VREF Generation for AGP 2.0 (2X and 4X)

VREF generation for AGP 2.0 is different, depending on the AGP card type used. 3.3-V AGP cards generate VREF locally. That is, they have a resistor divider on the card that divides VDDQ down to VREF (see Figure 36). To account for potential differences between VDDQ and GND at the MCH and graphics controller, 1.5-V cards use source-generated VREF. That is, the VREF signal is generated at the graphics controller and sent to the MCH, and another VREF is generated at the MCH and sent to the graphics controller (see Figure 36).

Both the graphics controller and the MCH must generate VREF and distribute it through the connector (1.5-V add-in cards only). The following two pins defined on the AGP 2.0 universal connector allow this VREF passing:

- VrefGC - VREF from the graphics controller to the chipset
- VrefCG - VREF from the chipset to the graphics controller

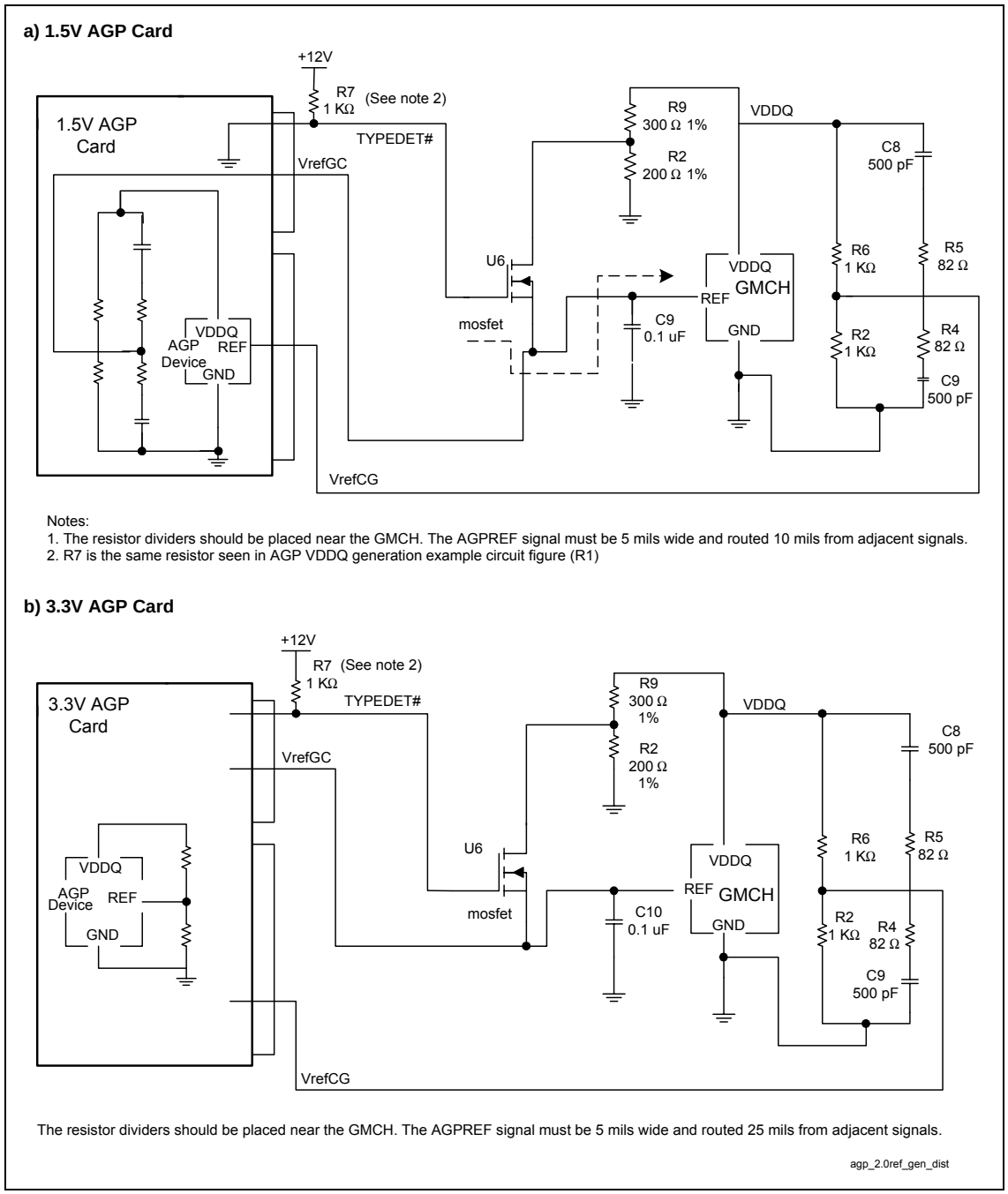
To preserve the common-mode relationship between the VREF and data signals, the routing of the two VREF signals must be matched in length to the strobe lines, within 0.5” on the motherboard and within 0.25” on the add-in card.

The voltage divider networks consist of AC and DC elements, as shown in the figure.

The VREF divider network should be placed as close as practical to the AGP interface, to get the benefit of the common-mode power supply effects. However, the trace spacing around the VREF signals must be a minimum of 25 mils to reduce crosstalk and maintain signal integrity.

During 3.3-V AGP 2.0 operation, VREF must be 0.4 VDDQ. However, during 1.5-V AGP 2.0 operation, VREF must be 0.5 VDDQ. This requires a flexible voltage divider for VREF. Various methods of accomplishing this exist, and one such example is shown in Figure 36.

Figure 36. AGP 2.0 VREF Generation and Distribution



The flexible VREF divider shown in the previous figure uses a FET switch to switch between the locally generated VREF (for 3.3-V add-in cards) and the source-generated VREF (for 1.5-V add-in cards).

Usage of the source-generated VREF at the receiver is optional and is a product implementation issue beyond the scope of this document.

6.5. Additional AGP Design Guidelines

6.5.1. Compensation

The MCH AGP interface supports resistive buffer compensation (RCOMP). Tie the GRCOMP pin to a 40-Ω, 2% (or 39-Ω, 1%) pull-down resistor (to ground) via a 10-mil-wide, very short (<0.5”) trace.

6.5.2. AGP Pull-Ups

AGP control signals require pull-up resistors to VDDQ on the motherboard, to ensure that they contain stable values when no agent is actively driving the bus. The signals requiring pull-up resistors are as follows:

1X Timing Domain Signals:

- **FRAME#**
- TRDY#
- IRDY#
- DEVSEL#
- STOP#
- SERR#
- PERR#
- RBF#
- PIPE#
- REQ#
- WBF#
- GNT#
- ST[2:0]

It is critical that these signals be pulled up to VDDQ, not 3.3 V.

The trace stub to the pull-up resistor on 1X timing domain signals should be kept shorter than 0.5” to avoid signal reflections from the stub.

The strobe signals require pull-up/pull-downs on the motherboard to ensure they contain stable values when no agent is driving the bus.

Note: INTA# and INTB# should be pulled to 3.3 V, not VDDQ.

2X/4X Timing Domain Signals:

- AD_STB[1:0] (pull-up to VDDQ)
- SB_STB (pull up to VDDQ)
- AD_STB[1:0]# (pull down to ground)
- SB_STB# (pull down to ground)

The trace stub to the pull-up/pull-down resistor on 2X/4X timing domain signals should be kept shorter than 0.1” to avoid signal reflections from the stub.

The pull-up/pull-down resistor value requirements are Rmin = 4 kΩ and Rmax = 16 kΩ. The recommended AGP pull-up/pull-down resistor value is 8.2 kΩ.

6.5.2.1. AGP Signal Voltage Tolerance List

The following signals on the AGP interface are 3.3-V tolerant during 1.5-V operation:

- PME#
- INTA#
- INTB#
- GPERR#
- GSERR#
- CLK
- RST

The following signals on the AGP interface are 5-V tolerant (see USB specification):

- USB+
- USB-
- OVRCNT#

The following signal is a special AGP signal. It is either GROUNDED or NO CONNECTED on an AGP card.

- TYPEDET#

ALL OTHER SIGNALS ON THE AGP INTERFACE ARE IN THE VDDQ GROUP. THEY ARE NOT 3.3-V TOLERANT DURING 1.5-V AGP OPERATION!

6.6. Motherboard / Add-in Card Interoperability

There are three AGP connectors: *3.3-V AGP connector*, *1.5-V AGP connector*, and *Universal AGP connector*. To maximize add-in flexibility, it is highly advisable to implement the universal connector in an 815EP chipset-based system. All add-in cards are *either* 3.3-V or 1.5-V cards. 4X transfers at 3.3 V are not allowed due to timings.

Table 22. Connector/Add-in Card Interoperability

	1.5-V Connector	3.3-V Connector	Universal Connector
1.5-V card	✔	NO	✔
3.3-V card	NO	✔	✔

Table 23. Voltage/Data Rate Interoperability

	1X	2X	4X
1.5 V VDDQ	✔	✔	✔
3.3 V VDDQ	✔	✔	NO

This page is intentionally left blank.

7. Hub Interface

The 815 chipset's MCH ball assignment and ICH2 ball assignment have been optimized to simplify hub interface routing. It is recommended that the hub interface signals be routed directly from the MCH to the ICH2 on the top signal layer. Refer to the following figure.

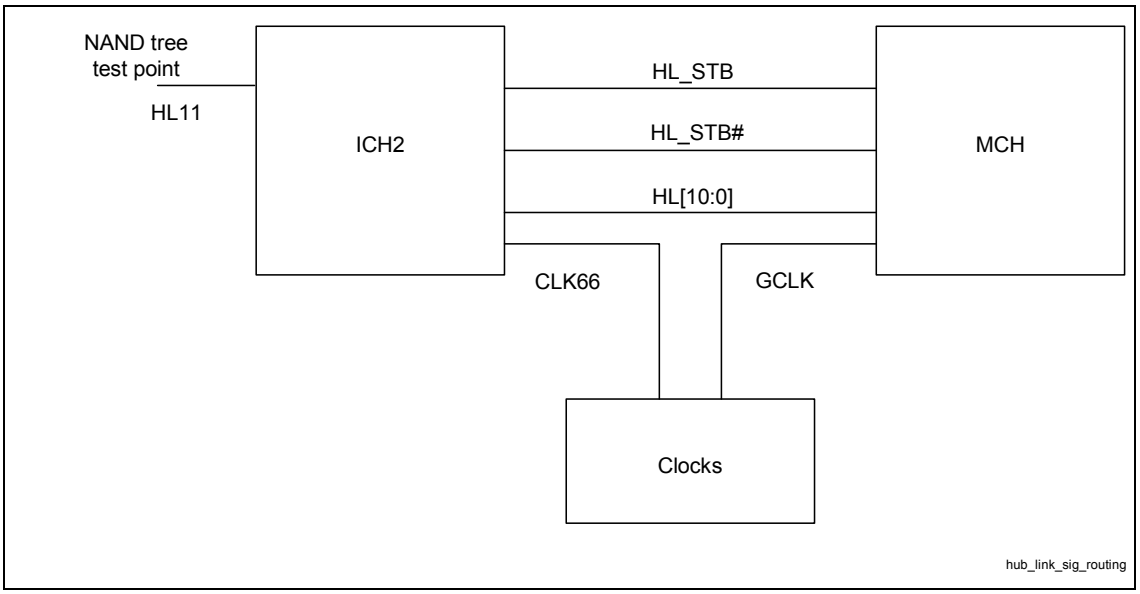
The hub interface is divided into two signal groups: data signals and strobe signals.

- Data Signals:
  - HL[10:0]
- Strobe Signals:
  - HL_STB
  - HL_STB#

Note: HL_STB/HL_STB# is a differential strobe pair.

No pull-ups or pull-downs are required on the hub interface. HL[11] on the ICH2 should be brought out to a test point for NAND Tree testing. Each signal should be routed such that it meets the guidelines documented for its signal group.

Figure 37. Hub Interface Signal Routing Example



7.1.1. Data Signals

Hub interface data signals should be routed with a trace width of *5* mils and a trace spacing of *20* mils. These signals can be routed with a trace width of *5* mils and a trace spacing of *15* mils for navigation around components or mounting holes. To break out of the MCH and the ICH2, the hub interface data signals can be routed with a trace width of *5* mils and a trace spacing of *5* mils. The signals should be separated to a trace width of *5* mils and a trace spacing of *20* mils, within 0.3" of the MCH/ICH2 components.

The maximum trace length for the hub interface data signals is 7". These signals should each be matched within ±0.1" of the HL_STB and HL_STB# signals.

7.1.2. Strobe Signals

Due to their differential nature, the hub interface strobe signals should be 5 mils wide and routed 20 mils apart. This strobe pair should be a minimum of 20 mils from any adjacent signals. The maximum length for the strobe signals is 7", and the two strobes should be the same length. Additionally, the trace length for each data signal should be matched to the trace length of the strobes, within ±0.1".

7.1.3. HREF Generation/Distribution

HREF, the hub interface reference voltage, is 0.5 * 1.85 V = 0.92 V ± 2%. It can be generated using a single HREF divider or locally generated dividers (as shown in the following two figures). The resistors should be equal in value and rated at 1% tolerance, to maintain 2% tolerance on 0.92 V. The values of these resistors must be chosen to ensure that the reference voltage tolerance is maintained over the entire input leakage specification. The recommended range for the resistor value is from a minimum of 100 Ω to a maximum of 1 kΩ (300 Ω shown in example).

The single HREF divider should not be located more than 4" away from either MCH or ICH2. If the single HREF divider is located more than 4" away, then the locally generated hub interface reference dividers should be used instead.

The reference voltage generated by a single HREF divider should be bypassed to ground at each component with a 0.01-µF capacitor located close to the component HREF pin. If the reference voltage is generated locally, the bypass capacitor must be close to the component HREF pin.

7.1.4. Compensation

Independent Hub interface compensation resistors are used by the 815 chipset's MCH and the ICH2 to adjust buffer characteristics to specific board characteristics. Refer to the *Intel® 815 Chipset Family: 82815EP Graphics and Memory Controller Hub (MCH) Datasheet* and the *Intel®* 82801BA I/O Controller Hub 2 (*ICH2) Datasheet* for details on compensation. The resistive Compensation (RCOMP) guidelines are as follows:

RCOMP: Tie the HLCOMP pin of each component to a 40-Ω, 1% or 2% pull-up resistor (to 1.8 V) via a 10-mil-wide, 0.5" trace (targeted at a nominal trace impedance of 40 Ω). The MCH and ICH2 each requires its own RCOMP resistor.

Figure 38. Single Hub Interface Reference Divider Circuit

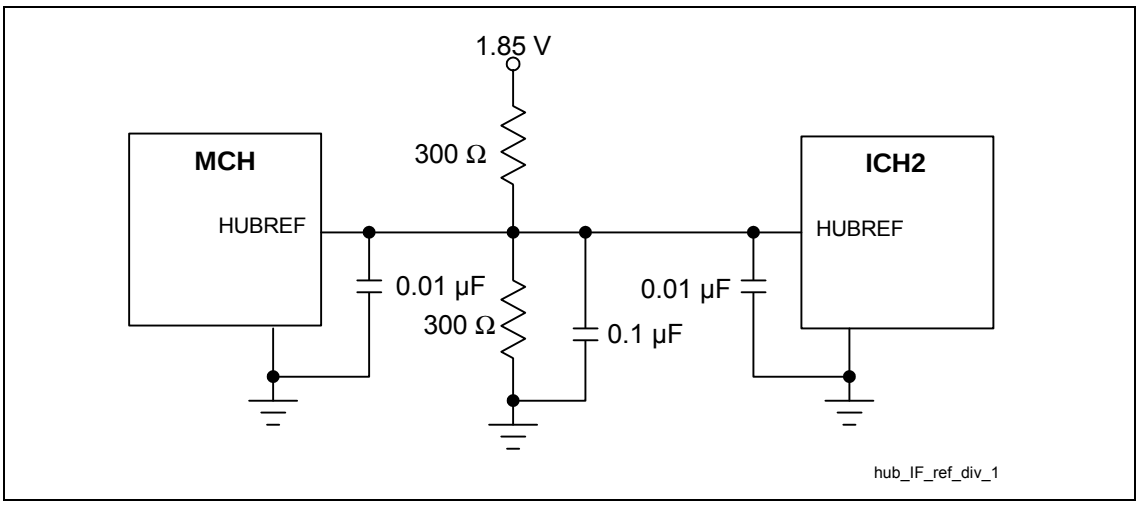
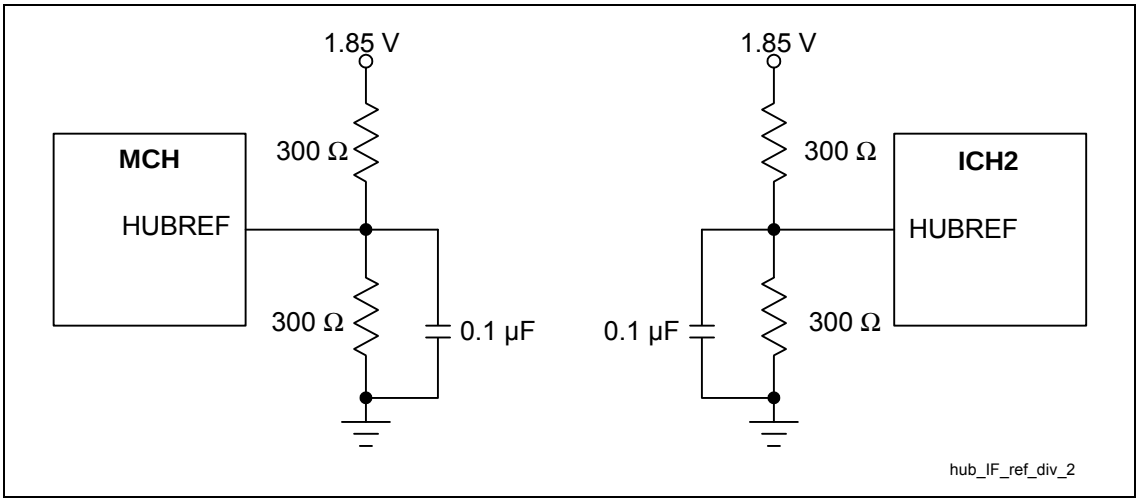


Figure 39. Locally Generated Hub Interface Reference Dividers



This page is intentionally left blank.

8. Intel® ICH2

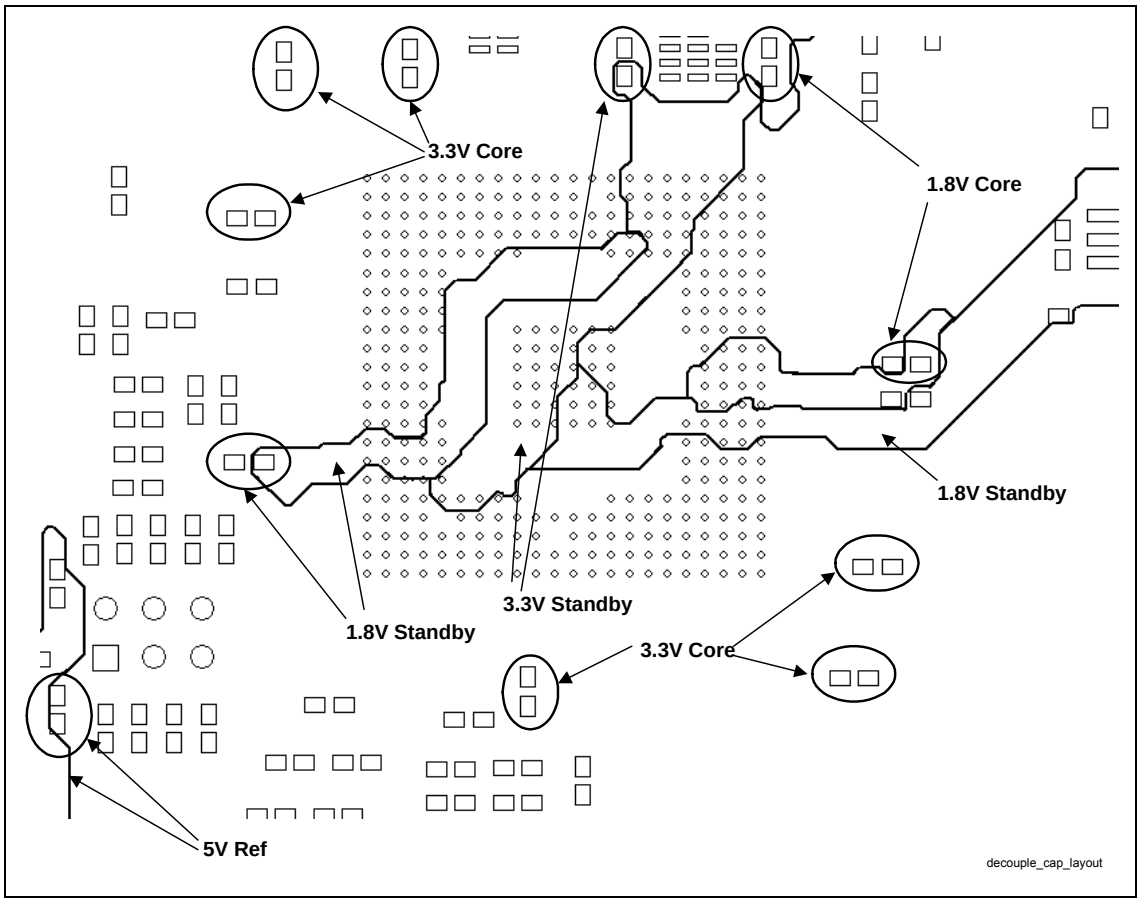
8.1. Decoupling

The ICH2 is capable of generating large current swings when switching between logic High and logic Low. This condition could cause the component voltage rails to drop below specified limits. To avoid this type of situation, ensure that the appropriate amount of bulk capacitance is added in parallel to the voltage input pins. It is recommended that the developer use the amount of decoupling capacitors specified in the following table to ensure that the component maintains stable supply voltages. The capacitors should be placed as close as possible to the package, without exceeding 400 mils (100–300 mils nominal). Note: Routing space around the ICH2 is tight. A few decoupling caps may be placed more than 300 mils away from the package. System designers should simulate the board to ensure that the correct amount decoupling is implemented. Refer to the following figure for a layout example. It is recommended that, for prototype board designs, the designer include pads for extra power plane decoupling caps.

Table 24. Decoupling Capacitor Recommendation

Power Plane/Pins	Decoupling Capacitors	Capacitor Value
3.3-V core	6	.1 µF
3.3-V standby	1	.1 µF
Processor interface (1.3 ~ 2.5 V)	1	.1 µF
1.8-V core	2	.1 µF
1.8-V standby	1	.1 µF
5-V reference	1	.1 µF
5-V reference standby	1	.1 µF

Figure 40. Intel® ICH2 Decoupling Capacitor Layout



8.2. 1.8V/3.3V Power Sequencing

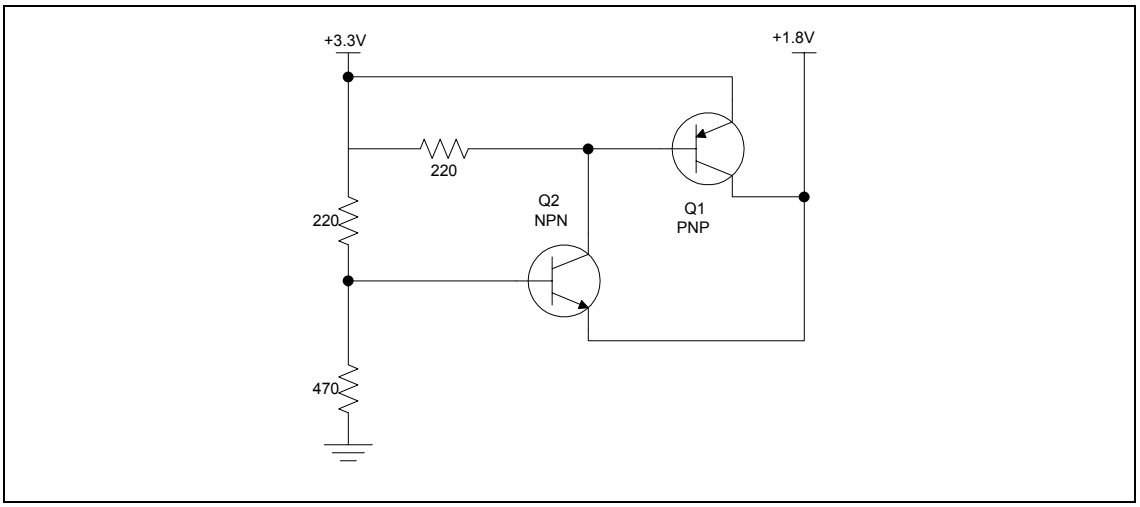
The ICH2 has two pairs of associated 1.8V and 3.3V supplies. These are {Vcc1_8, Vcc3_3} and {VccSus1_8, VccSus3_3}. These pairs are assumed to power up and power down together. **The difference between the two associated supplies must never be greater than 2.0V.** The 1.8V supply may come up before the 3.3V supply without violating this rule (though this is generally not practical in a desktop environment, since the 1.8V supply is typically derived from the 3.3V supply by means of a linear regulator).

One serious consequence of violation of this "2V Rule" is electrical overstress of oxide layers, resulting in component damage.

The majority of the ICH2 I/O buffers are driven by the 3.3V supplies, but are controlled by logic that is powered by the 1.8V supplies. Thus, another consequence of faulty power sequencing arises if the 3.3V supply comes up first. In this case the I/O buffers will be in an undefined state until the 1.8V logic is powered up. Some signals that are defined as "Input-only" actually have output buffers that are normally disabled, and the ICH2 may unexpectedly drive these signals if the 3.3V supply is active while the 1.8V supply is not.

The figure below shows an example power-on sequencing circuit that ensures the “2V Rule” is obeyed. This circuit uses a NPN (Q2) and PNP (Q1) transistor to ensure the 1.8V supply tracks the 3.3V supply. The NPN transistor controls the current through PNP from the 3.3V supply into the 1.8V power plane by varying the voltage at the base of the PNP transistor. By connecting the emitter of the NPN transistor to the 1.8V plane, current will not flow from the 3.3V supply into 1.8V plane when the 1.8V plane reaches 1.8V.

Figure 41. Example 1.8V/3.3V Power Sequencing Circuit



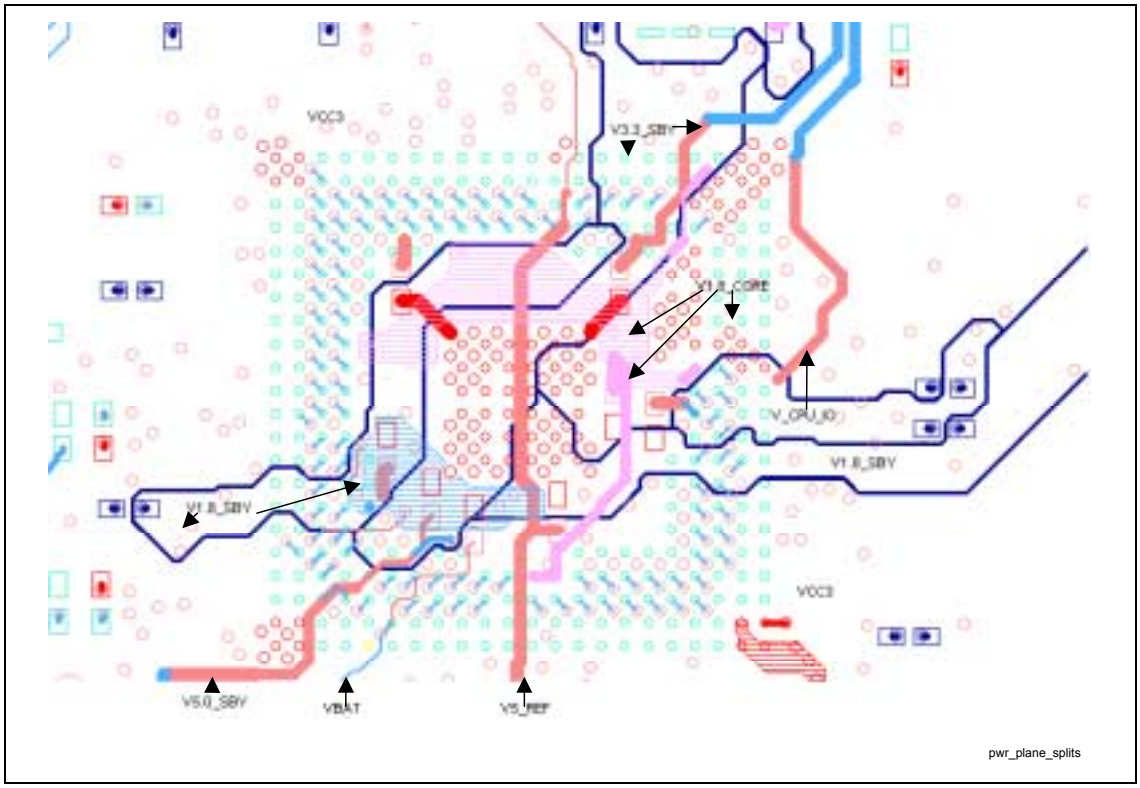
When analyzing systems that may be "marginally compliant" to the 2V Rule, pay close attention to the behavior of the ICH2's RSMRST# and PWROK signals, since these signals control internal isolation logic between the various power planes:

- RSMRST# controls isolation between the RTC well and the Resume wells.
- PWROK controls isolation between the Resume wells and Main wells

If one of these signals goes high while one of its associated power planes is active and the other is not, a leakage path will exist between the active and inactive power wells. This could result in high, possibly damaging, internal currents.

8.3. Power Plane Splits

Figure 42. Power Plane Split Example



8.4. Thermal Design Power

The thermal design power is the estimated maximum possible expected power generated in a component by a realistic application. It is based on extrapolations in both hardware and software technology over the life of the product. It does not represent the expected power generated by a power virus.

The thermal design power for the ICH2 is 1.5 W ±15%.

8.5. Power Supply PS_ON Considerations

If a pulse on SLP_S3# or SLP_S5# is short enough (~ 10–100 mS) such that PS_ON is driven active during the exponential decay of the power rails, a few power supplies may not be designed to handle this short pulse condition. In this case, the power supply will not respond to this event and never power back up. These power supplies would need to be unplugged and re-plugged to bring the system back up.

Power supplies not designed to handle this condition must have their power rails decay to a certain voltage level before they can properly respond to PS_ON. This level varies with affected power supply.

The ATX spec does not specify a minimum pulse width on PS_ON de-assertion, which means power supplies must be able to handle any pulse width. This issue can affect any power supply (beyond ATX) with similar PS_ON circuitry. Due to variance in the decay of the core power rails per platform, a single board or chipset silicon fix would be non-deterministic (may not solve the issue in all cases).

The platform designer must ensure that the power supply used with the platform is not affected by this issue.

8.6. I/O Subsystem

8.7. IDE Interface

This section contains guidelines for connecting and routing the ICH2 IDE interface. The ICH2 has two independent IDE channels. This section provides guidelines for IDE connector cabling and motherboard design, including component and resistor placement and signal termination for both IDE channels. The ICH2 has integrated the series resistors that typically have been required on the IDE data signals (PDD[15:0] and SDD[15:0]) running to the two ATA connectors. Intel does not anticipate requiring additional series termination, but OEMs should verify the motherboard signal integrity via simulation. Additional external 0-Ω resistors can be incorporated into the design to address possible noise issues on the motherboard. The additional resistor layout increases flexibility by providing future stuffing options.

The IDE interface can be routed with 5-mil traces on 7-mil spaces and must be less than 8” long (from ICH2 to IDE connector). Additionally, the shortest IDE signal (on a given IDE channel) must be less than 0.5” shorter than the longest IDE signal (on that channel).

8.7.1. Cabling

- **Length of cable:** Each IDE cable must be equal to or less than 18”.
- **Capacitance:** Less than 30 pF
- **Placement:** A maximum of 6” between drive connectors on the cable. If a single drive is placed on the cable it should be placed at the end of the cable. If a second drive is placed on the same cable it should be placed on the connector next closest to the end of the cable (6” away from the end of the cable).
- **Grounding:** Provide a direct, low-impedance chassis path between the motherboard ground and hard disk drives.
- **ICH2 Placement:** The ICH2 must be placed at most 8” from the ATA connector(s).
- **PC99 requirement:** Support Cable Select for master-slave configuration is a system design requirement of Microsoft* PC99. The CSEL signal of each ATA connector must be grounded at the host side.

8.8. Cable Detection for Ultra ATA/66 and Ultra ATA/100

The ICH2 IDE controller supports PIO, multiword (8237-style) DMA, and Ultra DMA modes 0 through 5. The ICH2 must determine the type of cable present, to configure itself for the fastest possible transfer mode that the hardware can support.

An 80-conductor IDE cable is required for Ultra ATA/66 and Ultra ATA/100. This cable uses the same 40-pin connector as the old 40-pin IDE cable. The wires in the cable alternate: ground, signal, ground, signal,…. All ground wires are tied together on the cable (and they are tied to the ground on the motherboard through the ground pins in the 40-pin connector). This cable conforms to the Small Form Factor Specification SFF-8049, which is obtainable from the Small Form Factor Committee.

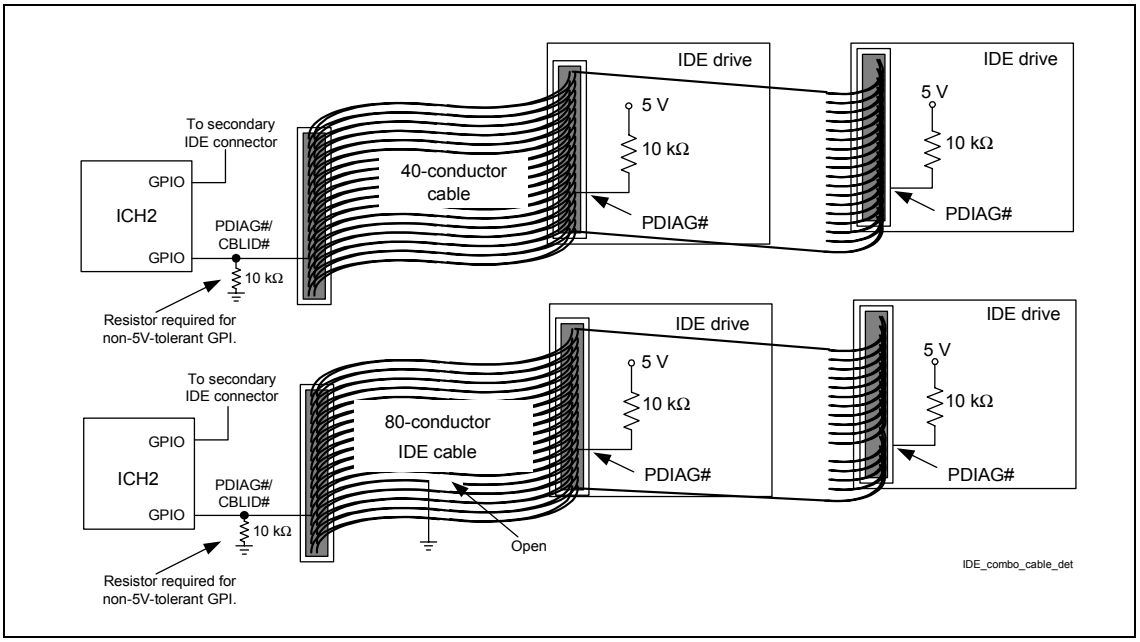
To determine whether the ATA/66 or ATA/100 mode can be enabled, the ICH2 requires that the system software attempt to determine the type of cable used in the system. If the system software detects an 80-conductor cable, it may use any Ultra DMA mode up to the highest transfer mode supported by both the chipset and the IDE device. If a 40-conductor cable is detected, the system software must not enable modes faster than Ultra DMA Mode 2 (Ultra ATA/33).

Intel recommends that cable detection be performed using a combination host-side/device-side detection mechanism. Note that host-side detection cannot be implemented on an NLX form factor system, since this configuration does not define interconnect pins for the PDIAG#/CBLID# from the riser (containing the ATA connectors) to the motherboard. These systems must rely on the device-side detection mechanism only.

8.8.1. Combination Host-Side/Device-Side Cable Detection

Host-side detection (described in the *ATA/ATAPI-4 Standard*, Section 5.2.11) requires the use of two GPI pins (one for each IDE channel). The proper way to connect the PDIAG#/CBLID# signal of the IDE connector to the host is shown in the figure below. All IDE devices have a 10-kΩ pull-up resistor to 5 V on this signal. Not all GPI and GPIO pins on the ICH2 are 5-V tolerant. If non 5-V tolerant inputs are used, a resistor divider is required to prevent 5 V on the ICH2 or FWH pins. The proper value of the divider resistor is 10 kΩ (as shown in the figure below).

Figure 43. Combination Host-Side / Device-Side IDE Cable Detection



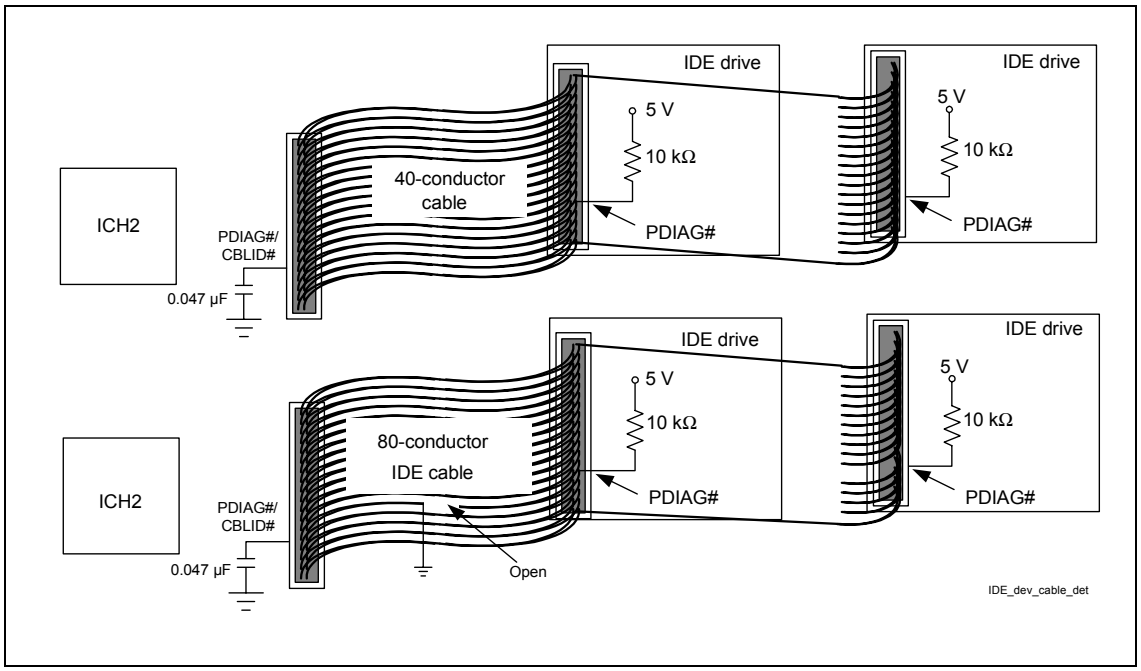
This mechanism allows the BIOS, after diagnostics, to sample PDIAG#/CBLID#. If the signal is High, then there is 40-conductor cable in the system and ATA modes 3, 4 and 5 must not be enabled.

If PDIAG#/CBLID# is detected Low, then there may be an 80-conductor cable in the system or there may be a 40-conductor cable and a legacy slave device (Device 1) that does not release the PDIAG#/CBLID# signal as required by the ATA/ATAPI-4 standard. In this case, the BIOS should check the **Identify Device** information in a connected device that supports Ultra DMA modes higher than 2. If ID Word 93, bit 13, is set to 1, then an 80-conductor cable is present. If this bit is set to 0, then a legacy slave (Device 1) is preventing proper cable detection, so the BIOS should configure the system as though a 40-conductor cable were present and then notify the user of the problem.

8.8.2. Device-Side Cable Detection

For platforms that must implement device-side detection only (e.g., NLX platforms), a 0.047-µF capacitor is required on the motherboard as shown in the figure below. This capacitor **should not be populated** when implementing the recommended combination host-side/device-side cable detection mechanism described previously.

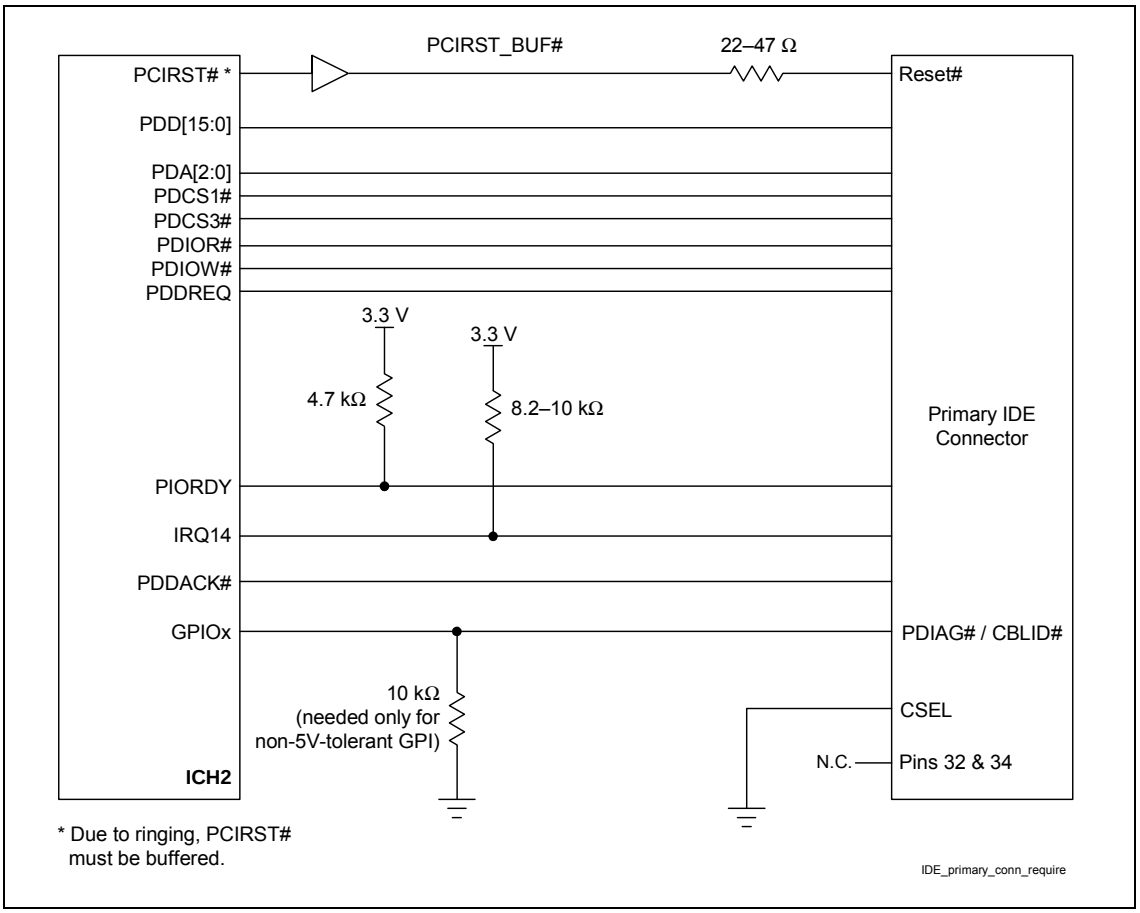
Figure 44. Device-Side IDE Cable Detection



This mechanism creates a resistor-capacitor (RC) time constant. The ATA mode 3, 4 or 5 drive will drive PDIAG#/CBLID# Low and then release it (pulled up through a 10-kΩ resistor). The drive will sample the signal after releasing it. In an 80-conductor cable, PDIAG#/CBLID# is not connected through to the host, so the capacitor has no effect. In a 40-conductor cable, the signal is connected to the host, so the signal will rise more slowly as the capacitor charges. The drive can detect the difference in rise times and will report the cable type to the BIOS when it sends the IDENTIFY_DEVICE packet during system boot, as described in the ATA/66 specification.

8.8.3. Primary IDE Connector Requirements

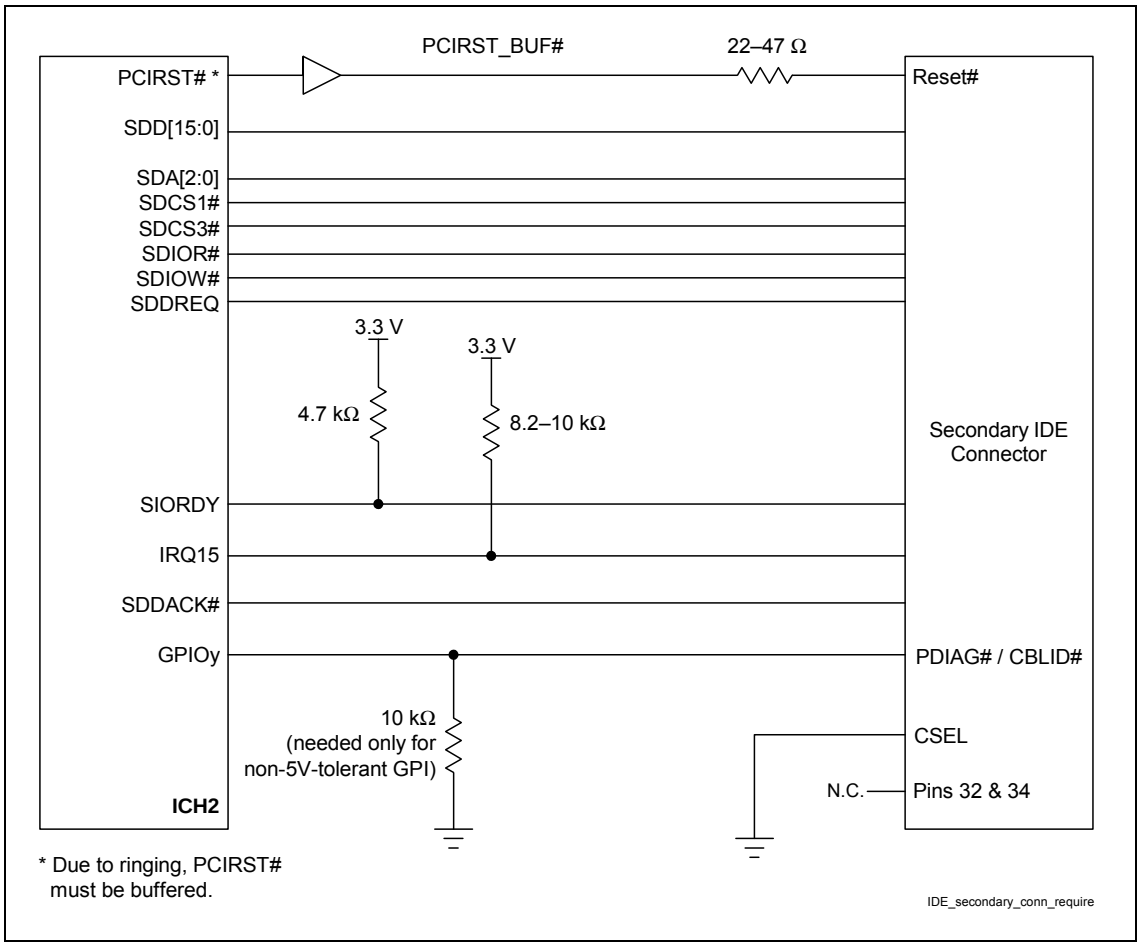
Figure 45. Connection Requirements for Primary IDE Connector



- 22-Ω to 47-Ω series resistors are required on RESET#. The correct value should be determined for each unique motherboard design, based on signal quality.
- An 8.2-kΩ to 10-kΩ pull-up resistor is required on IRQ14 and IRQ15 to VCC3.
- A 4.7-kΩ pull-up resistor to VCC3 is required on PIORDY and SIORDY.
- Series resistors can be placed on the control and data lines to improve signal quality. The resistors are placed as close as possible to the connector. Values are determined for each unique motherboard design.
- The 10 kΩ resistor to ground on the PDIAG#/CBLID# signal is now required on the Primary Connector. This change is to prevent the GPI pin from floating if a device is not present on the IDE interface.

8.8.4. Secondary IDE Connector Requirements

Figure 46. Connection Requirements for Secondary IDE Connector



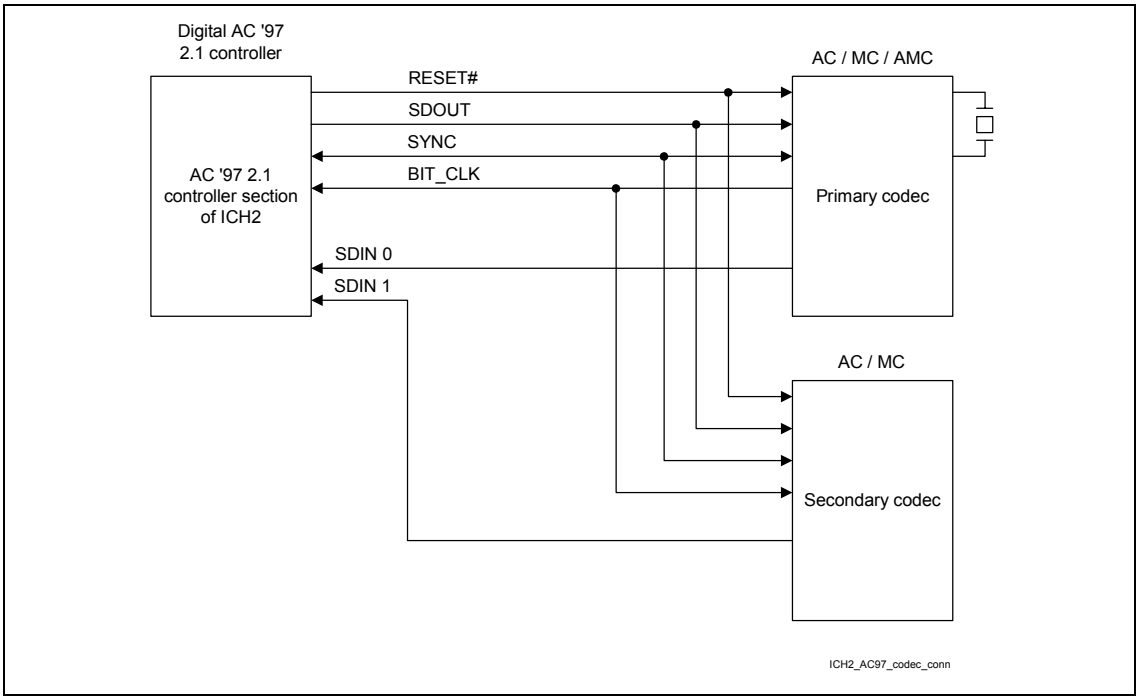
- 22-Ω to 47-Ω series resistors are required on RESET#. The correct value should be determined for each unique motherboard design, based on signal quality.
- An 8.2-kΩ to 10-kΩ pull-up resistor is required on IRQ14 and IRQ15 to VCC3.
- A 4.7-kΩ pull-up resistor to VCC3 is required on PIORDY and SIORDY
- Series resistors can be placed on the control and data lines to improve signal quality. The resistors are placed as close as possible to the connector. Values are determined for each unique motherboard design.
- The 10 kΩ resistor to ground on the PDIAG#/CBLID# signal is now required on the Primary Connector. This change is to prevent the GPI pin from floating if a device is not present on the IDE interface.

8.9. AC'97

The ICH2 implements an AC'97 2.1-compliant digital controller. Any codec attached to the ICH2 AC-link must be AC'97 2.1 compliant, as well. Contact your codec IHV for information on 2.1-compliant products. The AC'97 2.1 specification is available on the Intel website: http://developer.intel.com/pc-supp/platform/ac97/index.htm.

The AC-link is a bidirectional, serial PCM digital stream. It handles multiple input and output data streams, as well as control register accesses, by employing a time-division-multiplexed (TDM) scheme. The AC-link architecture enables data transfer through individual frames transmitted serially. Each frame is divided into 12 outgoing and 12 incoming data streams, or slots. The architecture of the ICH2 AC-link allows a maximum of two codecs to be connected. The following figure shows a two-codec topology of the AC-link for the ICH2.

Figure 47. Intel® ICH2 AC'97– Codec Connection



Intel has developed an advanced common connector for both AC'97 as well as networking options. This is known as the Communications and Network Riser (CNR). Refer to Section 8.10.

The AC '97 interface can be routed using 5 mil traces with 5 mil space between the traces. Maximum length between ICH2 to CODEC/CNR is 14 inches in a tee topology. This assumes that a CNR riser card implements its audio solution with a maximum trace length of 4 inches for the AC-link. Trace impedance should be $Z_0 = 60\ \Omega \pm 15\%$.

Clocking is provided from the primary codec on the link via BITCLK, and it is derived from a 24.576-MHz crystal or oscillator. Refer to the primary codec vendor for crystal or oscillator requirements. BITCLK is a 12.288-MHz clock driven by the primary codec to the digital controller (ICH2) and any other codec present. That clock is used as the timebase for latching and driving data.

The ICH2 supports wake-on-ring from S1-S5 via the AC-link. The codec asserts SDATAIN to wake the system. To provide wake capability and/or caller ID, standby power must be provided to the modem codec.

The ICH2 has weak pull-downs/pull-ups that are enabled only when the AC-Link Shut-off bit in the ICH2 is set. This keeps the link from floating when the AC-link is off or when there are no codecs present.

If the Shut-off bit is not set, it means that there is a codec on the link. Therefore, BITCLK and AC_SDOUT will be driven by the codec and ICH2, respectively. However, AC_SDIN0 and AC_SDIN1 may not be driven. If the link is enabled, it may be assumed that there is at least one codec. If there only is an on-board codec (i.e., no AMR), then the unused SDIN pin should have a weak (10-kΩ) pull-down to keep it from floating. If an AMR is used, any SDIN signal could be Not Connected (e.g., with no codec, both can be NC), then both SDIN pins must have a 10-kΩ pull-down.

Table 25. AC '97 SDIN Pull-Down Resistors

System Solution	Pull-up Requirements
On-board codec only	Pull-down the SDIN pin that is **not** connected to the codec.
AMR only	Pull-down **both** SDIN pins.
BOTH AMR and on-board codec	Pull-down any SDIN pin that could be NC*.

*If the on-board codec can be disabled, both SDIN pins must have pull-downs. If the on-board codec cannot be disabled, only the SDIN not connected to the on-board codec requires a pull-down.

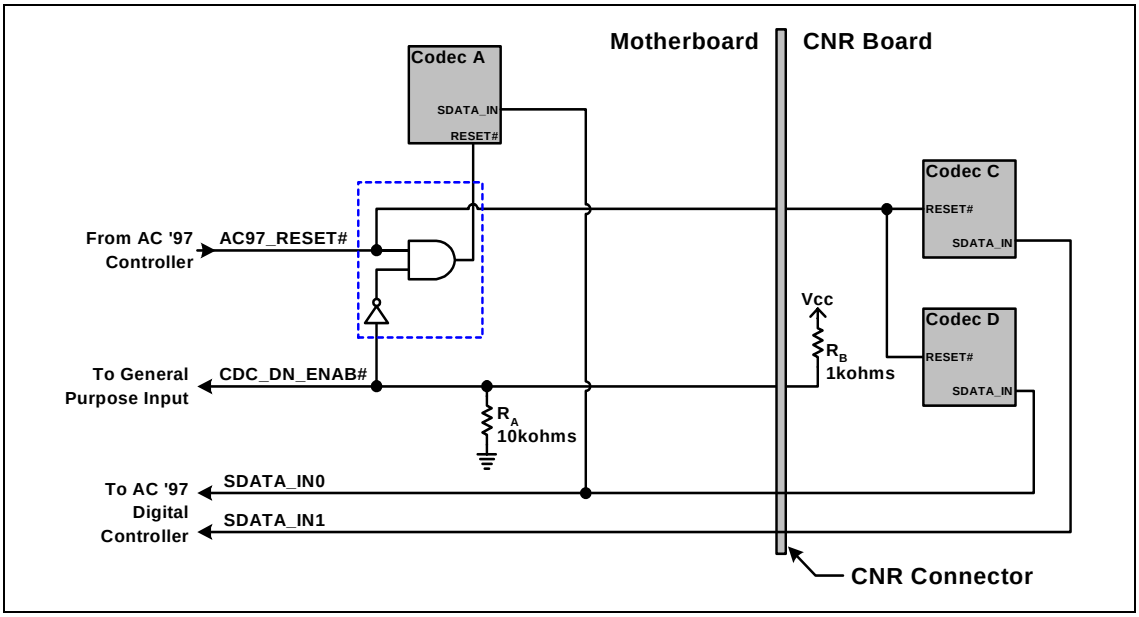
8.9.1. AC ’97 Audio Codec Detect Circuit and Configuration Options

The following provides general circuits to implement a number of different codec configurations. Please refer to Intel’s White Paper Recommendations for ICHx/AC ’97 Audio (Motherboard and Communication and Network Riser) for Intel’s recommended codec configurations.

To support more than two channels of audio output, the ICH2 allows for a configuration where two audio codecs work concurrently to provide surround capabilities. To maintain data-on-demand capabilities, the ICH2 AC ’97 controller, when configured for 4 or 6 channels, will wait for all the appropriate slot request bits to be set before sending data in the SDATA_OUT slots. This allows for simple FIFO synchronization of the attached codecs. It is assumed that both codecs will be programmed to the same sample rate, and that the codecs have identical (or at least compatible) FIFO depth requirements. It is recommended that the codecs be provided by the same vendor, upon the certification of their interoperability in an audio channel configuration.

The following circuits (Figure 48.a., Figure 48.b, Figure 48.c., and Figure 48.d.) show the adaptability of a system with the modification of R_A and R_B combined with some basic glue logic to support multiple codec configurations. This also provides a mechanism to make sure that only two codecs are enabled in a given configuration and allows the configuration of the link to be determined by the BIOS so that the correct PnP IDs can be loaded.

Figure 48. CDC_DN_ENAB# Support Circuitry for a Single Codec on Motherboard



As shown in Figure 48, when a single codec is located on the motherboard, the resistor R_A and the circuitry (AND and NOT gates) shown inside the dashed box must be implemented, on the motherboard. This circuitry is required in order to disable the motherboard codec when a CNR is installed which contains two AC ’97 codecs (or a single AC ’97 codec which must be the primary codec on the AC-Link).

By installing resistor R_B (1 kΩ) on the CNR, the codec on the motherboard becomes disabled (held in reset) and the codec(s) on the CNR take control of the AC-link. One possible example of using this architecture is a system integrator installing an audio plus modem CNR in a system already containing an audio codec on the motherboard. The audio codec on the motherboard would then be disabled, allowing all of the codecs on the CNR to be used.

The architecture shown in Figure 49. has some unique features. These include the possibility of the CNR being used as an upgrade to the existing audio features of the motherboard (by simply changing the value of resistor R_B on the CNR to 100 kΩ). An example of one such upgrade is increasing from two-channel to four or six-channel audio.

Both Figure 49 and Figure 50 show a switch on the CNR board. This is necessary to connect the CNR board codec to the proper SDATA_IN*n* line as to not conflict with the motherboard codec(s).

Figure 49. CDC_DN_ENAB# Support Circuitry for Multi-Channel Audio Upgrade

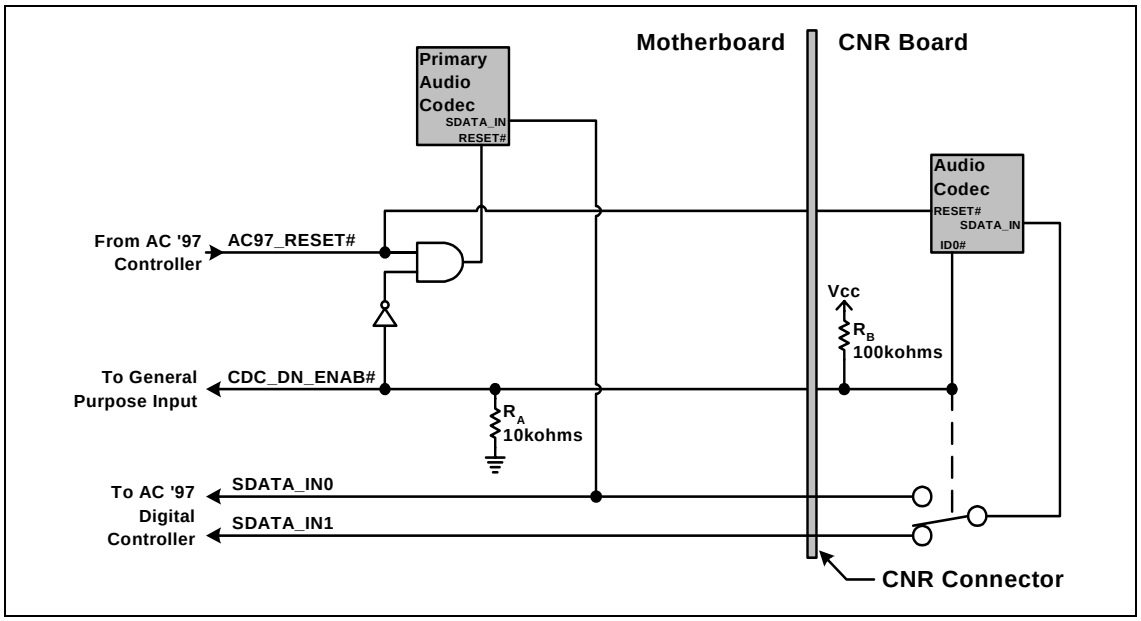


Figure 50 shows the circuitry required on the motherboard to support a two-codec down configuration. This circuitry disables the codec on a single codec CNR. Notice that in this configuration the resistor, R_B, has been changed to 100 kΩ.

Figure 50. CDC_DN_ENAB# Support Circuitry for Two-Codecs on Motherboard / One-Codec on CNR

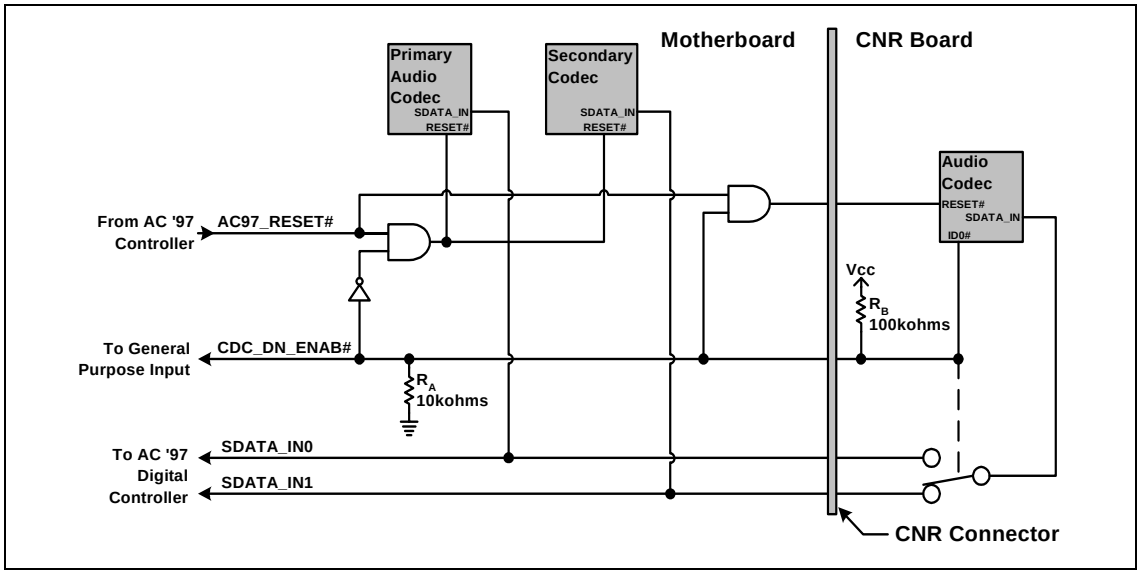
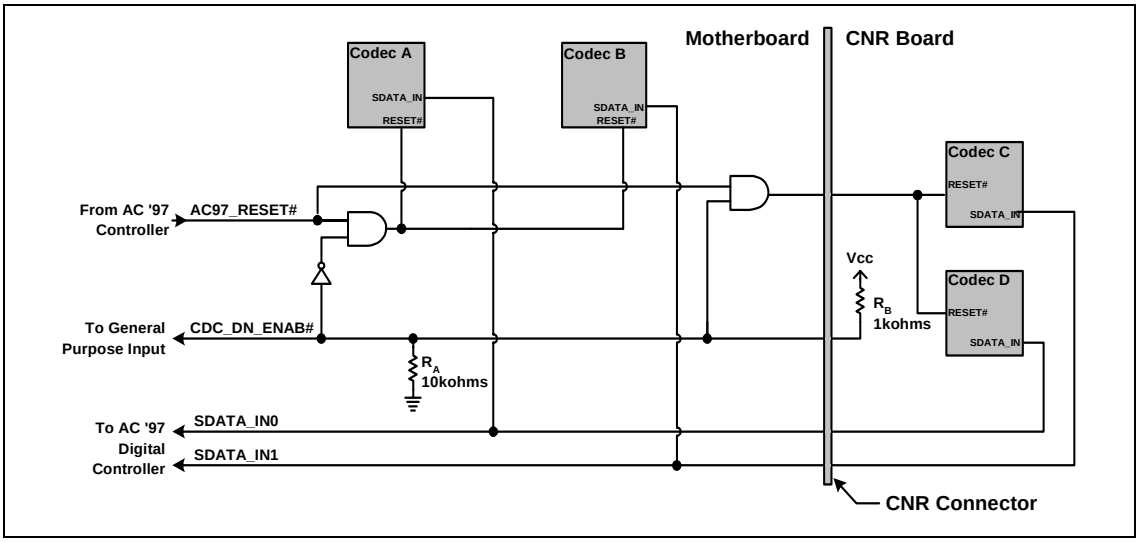


Figure 51 shows the case of two-codecs down and a dual-codec CNR. In this case, both codecs on the motherboard are disabled (while both on CNR are active) by R_A being 10 kΩ and R_B being 1 kΩ.

Figure 51. CDC_DN_ENAB# Support Circuitry for Two-Codecs on Motherboard / Two-Codecs on CNR



Circuit Notes

1. While it is possible to disable down codecs, as shown in Figure 47.a. and Figure 47.d., it is recommended against for reasons cited in the ICHx/AC'97 White Paper, including avoidance of shipping redundant and/or non-functional audio jacks.
2. All CNR designs include resistor R_B. The value of R_B is either 1 kΩ or 100 kΩ, depending on the intended functionality of the CNR (whether or not it intends to be the primary/controlling codec).
3. Any CNR with two codecs must implement R_B with value 1 kΩ. If there is one codec, use a 100 kΩ pull-up resistor. A CNR with zero codecs must not stuff R_B. If implemented, R_B must be connected to the same power well as the codec so that it is valid whenever the codec has power.
4. A motherboard with one or more codecs down must implement R_A with a value of 10 kΩ.
5. The CDC_DN_ENAB# signal must be run to a GPI so that the BIOS can sense the state of the signal. CDC_DN_ENAB# is *required* to be connected to a GPI; a connection to a GPIO is *strongly recommended* for testing purposes.

Table 26. Signal Descriptions

CDC_DN_ENAB#	When low, indicates that the codec on the motherboard is enabled and primary on the AC '97 Interface. When high, indicates that the motherboard codec(s) must be removed from the AC '97 Interface (held in reset), because the CNR codec(s) will be the primary device(s) on the AC '97 Interface.
AC97_RESET#	Reset signal from the AC '97 Digital Controller (Intel® ICH2).
SDATA_IN*n*	AC '97 serial data from an AC '97-compliant codec to an AC '97-compliant controller (i.e., the Intel® CH2).

8.9.2. Valid Codec Configurations

Table 27. Codec Configurations

Valid Codec Configurations	Invalid Codec Configurations
AC(Primary)	MC(Primary) + X(any other type of codec)
MC(Primary)	AMC(Primary) + AMC(Secondary)
AMC(Primary)	AMC(Primary) + MC(Secondary)
AC(Primary) + MC(Secondary)	
AC(Primary) + AC(Secondary)	
AC(Primary) + AMC(Secondary)	

8.9.3. SPKR Pin Considerations

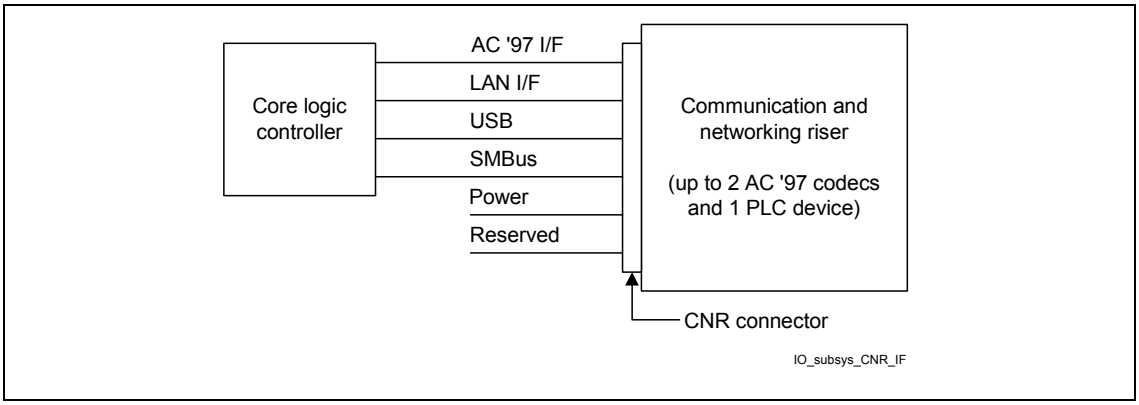
The effective impedance of the speaker and codec circuitry on the SPKR signal line must be greater than 50 kΩ. Failure to due so will cause the TCO Timer Reboot function to be erroneously disabled. SPKR is used as both the output signal to the system speaker and as a functional strap. The strap function enables or disables the "TCO Timer Reboot function" based on the state of the SPKR pin on the rising edge of POWEROK. When enabled, the ICH2 sends an SMI# to the processor upon a TCO timer timeout. The status of this strap is readable via the NO_REBOOT bit (bit 1, D31: F0, Offset D4h). The SPKR signal has a weak integrated pull up resistor (the resistor is only enabled during boot/reset). Therefore its default state when the pin is a "no connect" is a logical one or enabled. To disable the feature, a jumper can be populated to pull the signal line low. The value of the pulldown must be such that the voltage divider caused by the pull down and integrated pull up resistors will be read as logic low. When the jumper is not populated, a low can still be read on the signal line if the effective impedance due to the speaker and codec circuit is equal to or lower than the integrated pull up resistor. It is therefore strongly recommended that the effective impedance be greater than 50 kΩ and the pulldown resistor be less than 7.3 kΩ.

8.10. CNR

The Communication and Networking Riser (CNR) Specification defines a hardware-scalable Original Equipment Manufacturer (OEM) motherboard riser and interface. This interface supports multichannel audio, a V.90 analog modem, phone-line based networking, and 10/100 Ethernet based networking. The CNR specification defines the interface that should be configured before system shipment. Standard I/O expansion slots, such as those supported by the PCI bus architecture, are intended to continue serving as the upgrade medium. The CNR mechanically shares a PCI slot. Unlike in the case of the AMR, the system designer will not sacrifice a PCI slot after deciding not to include a CNR in a particular build.

The following figure indicates the interface for the CNR connector. Refer to the appropriate section of this document for the appropriate design and layout guidelines. The Platform LAN Connection (PLC) can either be an 82562EH or 82562ET component. Refer to the CNR specification for additional information.

Figure 52. CNR Interface



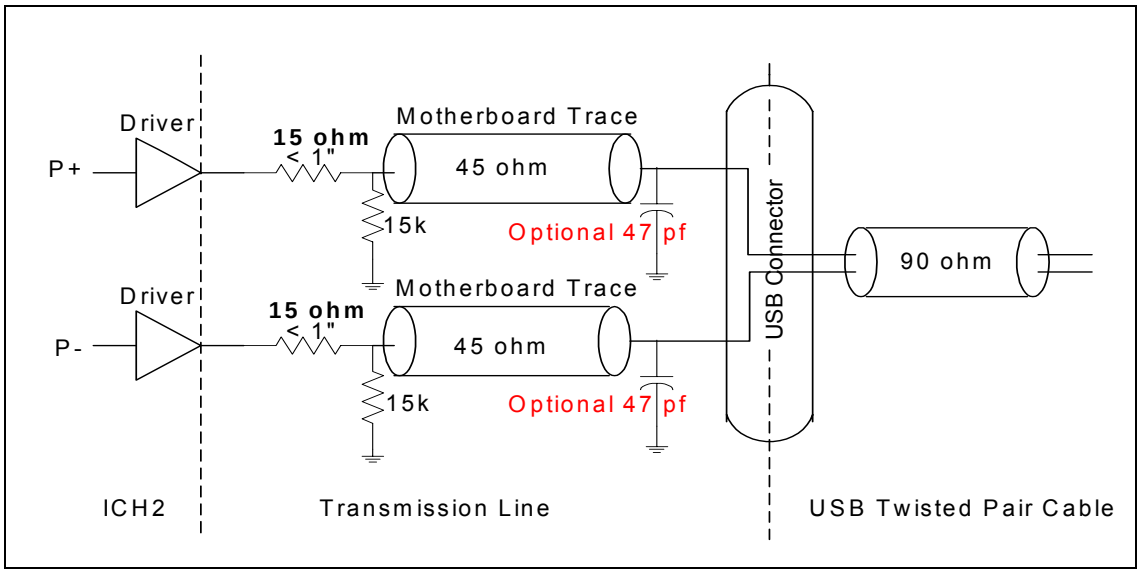
8.11. USB

The general guidelines for the USB interface are as follows:

- Unused USB ports should be terminated with 15-kΩ pull-down resistors on both P+/P- data lines.
- 15-Ω series resistors should be placed as close as possible to the ICH2 (<1”). These series resistors are required for source termination of the reflected signal.
- An optional cap (0 pF – 47pF) may be placed as close to the USB connector side of the series resistors on the USB data lines (P0+/-, P1+/-, P2+/-, P3+/-). This cap should be sized to minimize EMI radiation while still maintaining signal quality (rise/fall time, Vcrs, etc.).
- 15-kΩ ± 5% pull-down resistors should be placed on the USB side of the series resistors on the USB data lines (P0± … P3±), and they are REQUIRED for signal termination by the USB specification. The stub should be as short as possible.
- The trace impedance for the P0±…P3± signals should be 45 Ω (to ground) for each USB signal P+ or P-. When the stack-up recommended in Figure 4 is used, the USB requires 9-mil traces. The impedance is 90 Ω between the differential signal pairs P+ and P-, to match the 90-Ω USB twisted-pair cable impedance. Note that the twisted-pair’s characteristic impedance of 90 Ω is the series impedance of both wires, resulting in an individual wire presenting a 45-Ω impedance. The trace impedance can be controlled by carefully selecting the trace width, trace distance from power or ground planes, and physical proximity of nearby traces.
- USB data lines must be routed as critical signals. The P+/P- signal pair must be routed together, parallel to each other on the same layer, and not parallel with other non-USB signal traces to minimize crosstalk. Doubling the space from the P+/P- signal pair to adjacent signal traces will help to prevent crosstalk. Do not worry about crosstalk between the two P+/P- signal traces. The P+/P- signal traces must also be the same length. This will minimize the effect of common mode current on EMI. Lastly, do not route over plane splits.

The following figure illustrates the recommended USB schematic.

Figure 53. USB Data Signals



The recommended USB trace characteristics are:

- Impedance 'Z0' = 45.4 Ω
- Line Delay = 160.2 ps
- Capacitance = 3.5 pF
- Inductance = 7.3 nH
- Res @ 20° C = 53.9 mΩ

8.11.1. Disabling the Native USB Interface of the Intel® ICH2

The ICH2 native USB interface can be disabled. This can be done when an external PCI based USB controller is being implemented in the platform. To disable the native USB Interface, ensure the differential pairs are pulled down thru 15 kΩ resistors, ensure the OC[3:0]# signals are de-asserted by pulling them up weakly to VCC3SBY, and that both function 2 and 4 are disabled via the D31:F0;FUNC_DIS register. Ensure that the 48 MHz USB clock is connected to the ICH2 and is kept running. This clock must be maintained even though the internal USB functions are disabled.

8.12. ISA

Implementations that require ISA support can benefit from the enhancements of the ICH2, while "ISA-less" designs are not burdened with the complexity and cost of the ISA subsystem. For information regarding the implementation of an ISA design, contact external suppliers.

8.13. I/O APIC Design Recommendation

UP systems not using the I/O APIC should comply with the following recommendations:

- On the ICH2:
  - Tie PICCLK directly to ground.
  - Tie PICD0, PICD1 to ground through a 10-kΩ resistor.
- On the processor:
  - PICCLK must be connected from the clock generator to the PICCLK pin on the processor.
  - Tie PICD0 to 2.5 V through 10-kΩ resistors.
  - Tie PICD1 to 2.5 V through 10-kΩ resistors.

8.14. SMBus/SMLink Interface

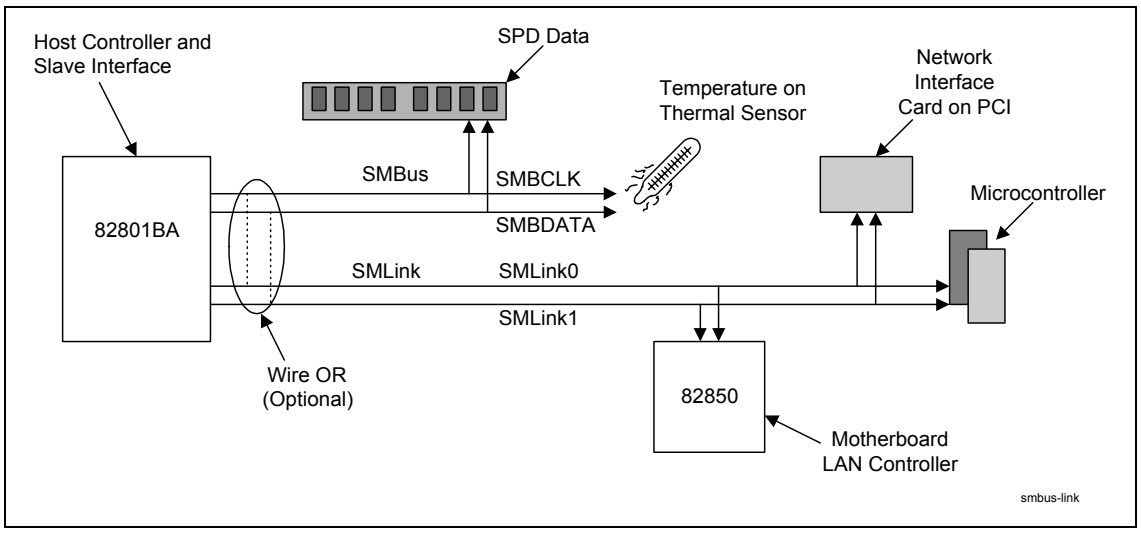
The SMBus interface on the ICH2 is the same as that on the ICH. It uses two signals, SMBCLK and SMBDATA, to send and receive data from components residing on the bus. These signals are used exclusively by the SMBus host controller. The SMBus host controller resides inside the ICH2.

The ICH2 incorporates a new SMLink interface supporting AOL*, AOL2*, and slave functionality. It uses two signals, SMLINK[1:0]. SMLINK[0] corresponds to an SMBus clock signal, and SMLINK[1] corresponds to an SMBus data signal. These signals are part of the SMB slave interface.

For Alert on LAN (AOL) functionality, the ICH2 transmits heartbeat and event messages over the interface. When the 82562EM LAN connect component is used, the ICH2's integrated LAN controller claims the SMLink heartbeat and event messages and sends them out over the network. An external, AOL2-enabled LAN controller will connect to the SMLink signals, to receive heartbeat and event messages as well to as access the ICH2 SMBus slave interface. The slave interface function allows an external microcontroller to perform various functions. For example, the slave write interface can reset or wake a system, generate SMI# or interrupts, and send a message. The slave read interface can read the system power state, read the watchdog timer status, and read system status bits.

Both the SMBus host controller and the SMBus slave interface obey the SMBus protocol, so the two interfaces can be externally wire-ORed together to allow an external management ASIC to access targets on the SMBus as well as the ICH2 slave interface. This is performed by connecting SMLink[0] to SMBCLK and SMLink[1] to SMBDATA, as shown in the following figure. Since the SMBus and SMLINK are pulled up to VCCSUS3_3, system designers must ensure that they implement proper isolation for any devices that may be powered down while VCCSUS3_3 is still active (i.e., thermal sensors).

Figure 54. SMBus/SMLink Interface



Note: Intel does not support external access to the ICH2's integrated LAN controller via the SMLink interface. Also, Intel does not support access to the ICH2's SMBus slave interface by the ICH2's SMBus host controller. The following table describes the pull-up requirements for different implementations of the SMBus and SMLink signals.

Table 28. Pull-Up Requirements for SMBus and SMLink

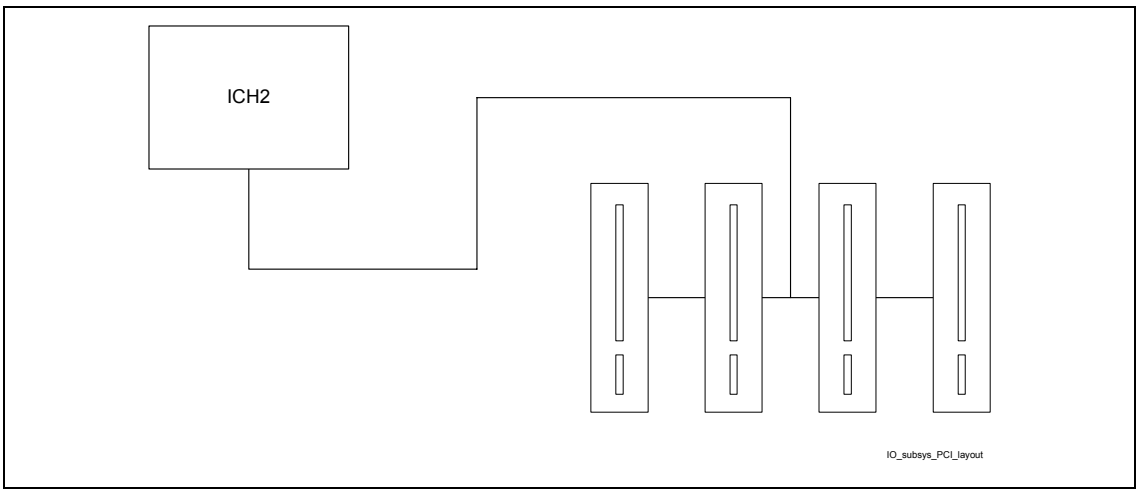
SMBus / SMLink Use	Implementation
Alert-on-LAN* signals	4.7-kΩ pull-up resistors to 3.3 VSB are required.
GPIOs	Pull-up resistors to 3.3 VSB and the signals must be allowed to change states on power-up. (For example, on power-up the Intel® ICH2 will drive *heartbeat* messages until the BIOS programs these signals as GPIOs.) The value of the pull-up resistors depends on the loading on the GPIO signal.
Not Used	4.7-kΩ pull-up resistors to 3.3 VSB are required.

8.15. PCI

The ICH2 provides a PCI Bus interface compliant with the *PCI Local Bus Specification, Revision 2.2*. The implementation is optimized for high-performance data streaming when the ICH2 is acting as either the target or the initiator on the PCI bus. For more information on the PCI Bus interface, refer to the *PCI Local Bus Specification, Revision 2.2*.

The ICH2 supports six PCI Bus masters (excluding the ICH2), by providing six REQ#/GNT# pairs. In addition, the ICH2 supports two PC/PCI REQ#/GNT# pairs, one of which is multiplexed with a PCI REQ#/GNT# pair.

Figure 55. PCI Bus Layout Example



8.16. RTC

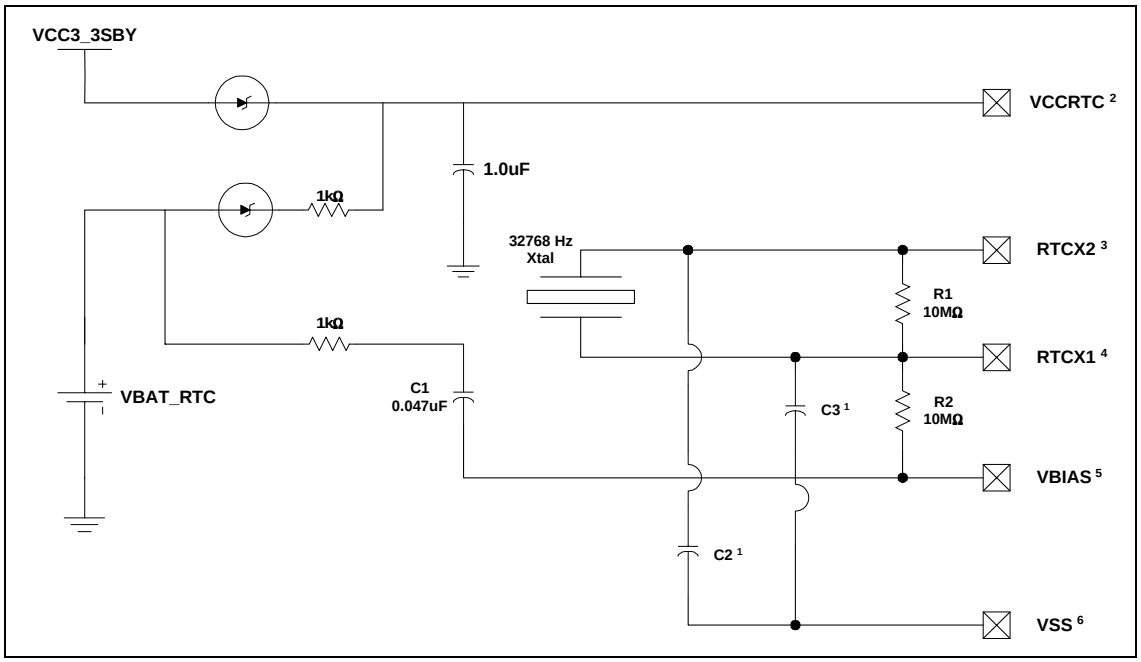
The ICH2 contains a real-time clock (RTC) with 256 bytes of battery-backed SRAM. The internal RTC module provides two key functions: keeping the date and time and storing system data in its RAM when the system is powered down.

This section will present the recommended hook-up for the RTC circuit for the ICH2.

8.16.1. RTC Crystal

The ICH2 RTC module requires an external oscillating source of 32.768 kHz connected on the RTCX1 and RTCX2 pins. The following figure documents the external circuitry that comprises the oscillator of the ICH2 RTC.

Figure 56. External Circuitry for the Intel® ICH2 RTC



NOTES:

1. The exact capacitor value must be based on the crystal maker's recommendation. (Typical values for C2 and C3 are 18 pF for a crystal with CLOAD=12.5 pF)
2. VccRTC: Power for RTC well
3. RTCX2: Crystal input 2 – Connected to the 32.768-kHz crystal.
4. RTCX1: Crystal input 1 – Connected to the 32.768-kHz crystal.
5. VBIAS: RTC BIAS voltage – This pin is used to provide a reference voltage. This DC voltage sets a current that is mirrored throughout the oscillator and buffer circuitry.
6. Vss: Ground

8.16.2. External Capacitors

To maintain RTC accuracy, the external capacitor C1 must be 0.047 µF, and the external capacitor values (C2 and C3) should be chosen to provide the manufacturer-specified load capacitance (Cload) for the crystal, when combined with the parasitic capacitance of the trace, socket (if used), and package. When the external capacitor values are combined with the capacitance of the trace, socket, and package, the closer the capacitor value can be matched to the actual load capacitance of the crystal used, the more accurate the RTC will be.

The following equation can be used to choose the external capacitance values (C2 and C3):

Cload = (C2 * C3) / (C2 + C3) + Cparasitic

C3 can be chosen such that C3 > C2. Then C2 can be trimmed to obtain 32.768 kHz.

8.16.3. RTC Layout Considerations

- Keep the RTC lead lengths as short as possible. Approximately 0.25” is sufficient.
- Minimize the capacitance between Xin and Xout in the routing.
- Put a ground plane under the XTAL components.
- Don’t route switching signals under the external components (unless on the other side of the board).
- The oscillator VCC should be clean. Use a filter, such as an RC low-pass or a ferrite inductor.

8.16.4. RTC External Battery Connection

The RTC requires an external battery connection to maintain its functionality and its RAM while the ICH2 is not powered by the system.

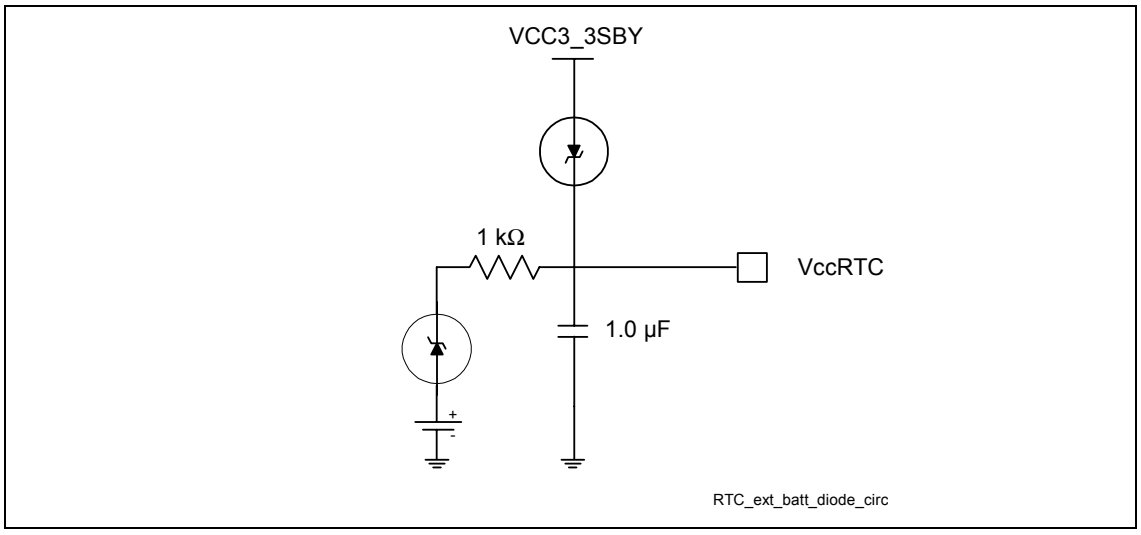
Example batteries include the Duracell* 2032, 2025 or 2016 (or equivalent), which give many years of operation. Batteries are rated by storage capacity. The battery life can be calculated by dividing the capacity by the average current required. For example, if the battery storage capacity is 170 mAh (assumed usable) and the average current required is 3 µA, the battery life will be at least:

170,000 µAh / 3 µA = 56,666 h = 6.4 years

The voltage of the battery can affect the RTC accuracy. In general, when the battery voltage decays, the RTC accuracy also decreases. High accuracy can be obtained when the RTC voltage is within the range 3.0 V to 3.3 V.

The battery must be connected to the ICH2 via an isolation Schottky diode circuit. The Schottky diode circuit allows the ICH2 RTC well to be powered by the battery when system power is unavailable, but by system power when it is available. So, the diodes are set to be reverse-biased when system power is unavailable. The following figures shows an example of the used diode circuitry.

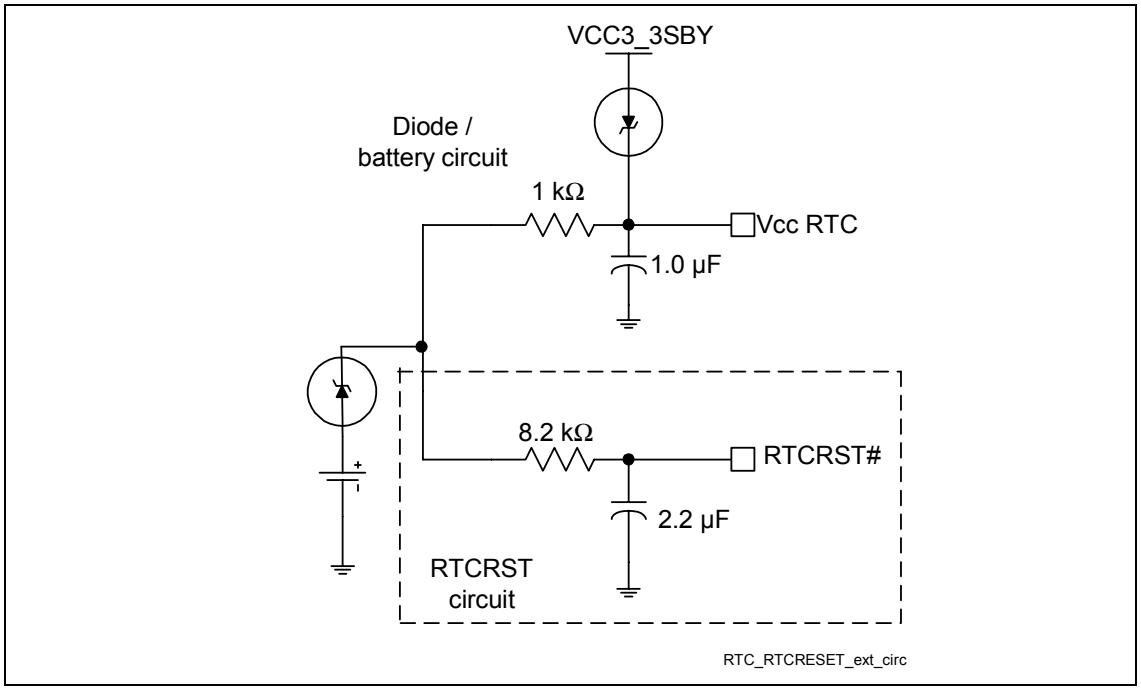
Figure 57. Diode Circuit to Connect RTC External Battery



A standby power supply should be used in a desktop system, to provide continuous power to the RTC when available, which will significantly increase the RTC battery life and thereby the RTC accuracy.

8.16.5. RTC External RTCRST Circuit

Figure 58. RTCRST External Circuit for Intel® ICH2 RTC



The ICH2 RTC requires some additional external circuitry. The RTCRST# signal is used to reset the RTC well. The external capacitor and the external resistor between RTCRST# and the RTC battery (Vbat) were selected to create an RC time delay, such that RTCRST# will go High some time after the battery voltage is valid. The RC time delay should be within the range of 10–20 ms. When RTCRST# is asserted, bit 2 (RTC_PWR_STS) in the GEN_PMCON_3 (General PM Configuration 3) register is set to 1 and remains set until cleared by software. As a result, when the system boots, the BIOS knows that the RTC battery has been removed.

This RTCRST# circuit is combined with the diode circuit (see previous figure), which allows the RTC well to be powered by the battery when system power is unavailable. The previous figure shows an example of this circuitry when used in conjunction with the external diode circuit.

8.16.6. RTC Routing Guidelines

- All RTC OSC signals (RTCX1, RTCX2, VBIAS) should all be routed with trace lengths less than 1”. The shorter, the better.
- Minimize the capacitance between RTCX1 and RTCX2 in the routing. (Optimally, there would be a ground line between them.)
- Put a ground plane under all external RTC circuitry.
- Don’t route any switching signals under the external components (unless on the other side of the ground plane).

8.16.7. VBIAS DC Voltage and Noise Measurements

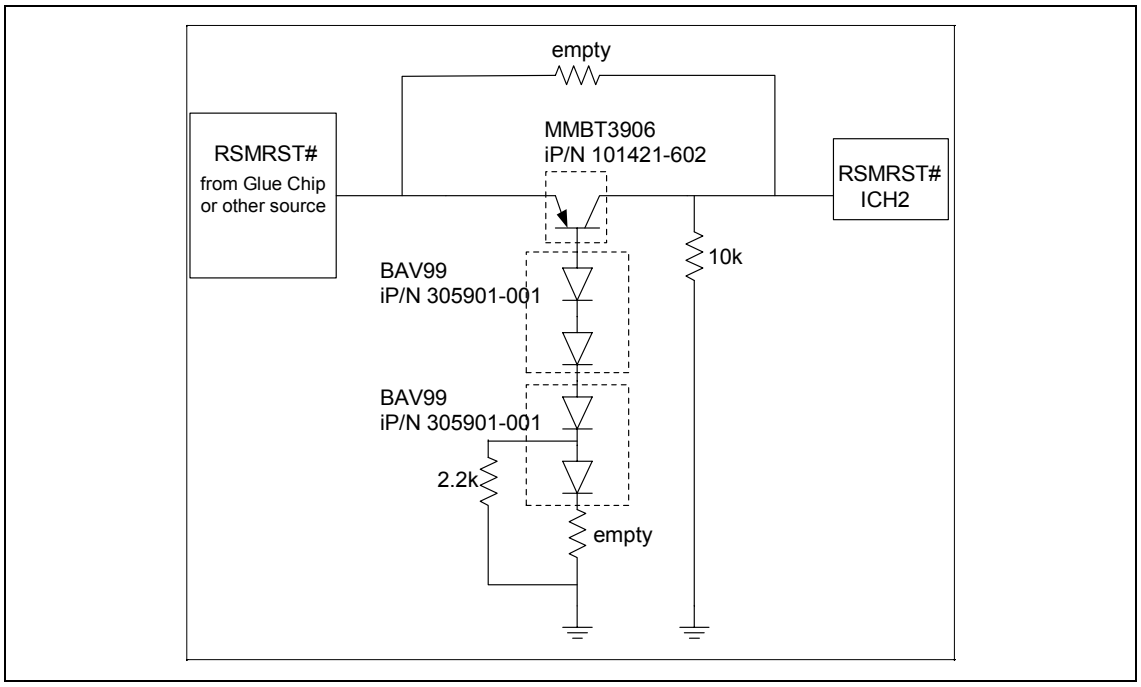
- All RTC OSC signals (RTCX1, RTCX2, VBIAS) should all be routed with trace lengths less than 1”. The shorter, the better.
- Steady-state VBIAS is a DC voltage of about 0.38 V ± .06 V.
- When the battery is inserted, VBIAS will be “kicked” to about 0.7–1.0 V, but it will return to its DC value within a few ms.
- Noise on VBIAS must be kept to a minimum (200 mV or less).
- VBIAS is very sensitive and cannot be directly probed, but it can be probed through a .01-µF capacitor.
- Excessive noise on VBIAS can cause the ICH2 internal oscillator to misbehave or even stop completely.
- To minimize VBIAS noise, it is necessary to implement the routing guidelines described previously as well as the required external RTC circuitry, as described in the *Intel® 82801BA I/O Controller Hub 2 (ICH2) Datasheet*.

8.16.8. Power-Well Isolation Control

All RTC-well inputs (RSMRST#, RTCRST#, INTRUDER#) must be either pulled up to VCCRTC or pulled down to ground while in G3 state. RTCRST# when configured as shown in Figure 54 meets this requirement. RSMRST# should have a weak external pull-down to ground and INTRUDER# should have a weak external pull-up to VCCRTC. This will prevent these nodes from floating in G3, and correspondingly will prevent ICCRTC leakage that can cause excessive coin-cell drain. The PWROK input signal should also be configured with an external weak pull-down.

The circuit shown in the figure below should be implemented to control well isolation between the 3.3V resume and RTC power-wells. Failure to implement this circuit may result in excessive droop on the VCCRTC node during Sx-to-G3 power state transitions (removal of AC power).

Figure 59. RTC – Power-Well Isolation Control Circuit



8.17. LAN Layout Guidelines

The ICH2 provides several options for integrated LAN capability. The platform supports several components, depending on the target market.

LAN Connect Component	Connection	Features
Intel® 82562EM	Advanced 10/100 Ethernet	AOL* & Ethernet 10/100 connection
Intel® 82562ET	10/100 Ethernet	Ethernet 10/100 connection
Intel® 82562EH	1-Mb HomePNA* LAN	1-Mb HomePNA connection

Intel developed a dual footprint for 82562ET and 82562EH, to minimize the required number of board builds. A single layout with the specified dual footprint allows the OEM to install the appropriate LAN connect component to satisfy market demand. Design guidelines are provided for each required interface and connection. Refer to the following figure and table for the corresponding section of the design guide.

Figure 60. Intel® ICH2 / LAN Connect Section

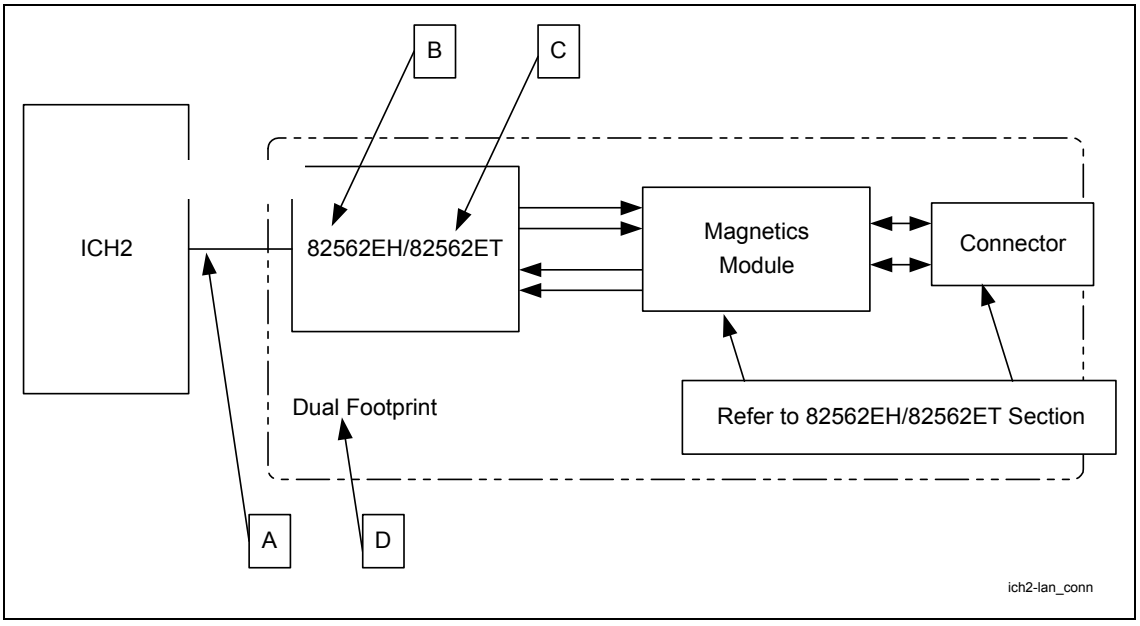


Table 29. LAN Design Guide Section Reference

Layout Section	Figure Ref.	Design Guide Section
Intel® ICH2 – LAN interconnect	A	8.17.1 Intel® ICH2 – LAN Interconnect Guidelines
General routing guidelines	B,C,D	Section 8.17.2 General LAN Routing Guidelines and Considerations
Intel® 82562EH	B	Section 8.17.3 Intel® 82562EH Home/PNA* Guidelines
Intel® 82562ET /82562EM	C	Section 8.17.4 Intel® 82562ET / 82562EM Guidelines
Dual layout footprint	D	Section 8.17.4.6 Intel® 82562ET/EM Disable Guidelines

8.17.1. Intel® ICH2 – LAN Interconnect Guidelines

This section contains the guidelines for the design of motherboards and riser cards that comply with LAN connect. It should not be considered a specification, and the system designer must ensure through simulations or other techniques that the system meets the specified timings. Special care must be taken to match the **LAN_CLK** traces with those of the other signals, as follows. The following guidelines are for the ICH2-to-LAN component interface. The following signal lines are used on this interface:

- LAN_CLK
- LAN_RSTSYNC
- LAN_RXD[2:0]
- LAN_TXD[2:0]

This interface supports both 82562EH and 82562ET/82562EM components. Both components share signal lines LAN_CLK, LAN_RSTSYNC, LAN_RXD[0], and LAN_TXD[0]. Signal lines LAN_RXD[2:1] and LAN_TXD[2:1] are not connected when 82562EH is installed.

The AC Characteristics for this interface are found in the ICH2 EDS Addendum (Doc# FM-1720). Dual footprint guidelines are found in Section 4.17.5.

8.17.1.1. Bus Topologies

The LAN connect interface can be configured in several topologies:

- Direct point-to-point connection between the ICH2 and the LAN component
- Dual footprint
- LOM/CNR implementation

8.17.1.2. Point-to-Point Interconnect

The following guidelines are for a single-solution motherboard. Either 82562EH, 82562ET or CNR is installed.

Figure 61. Single-Solution Interconnect

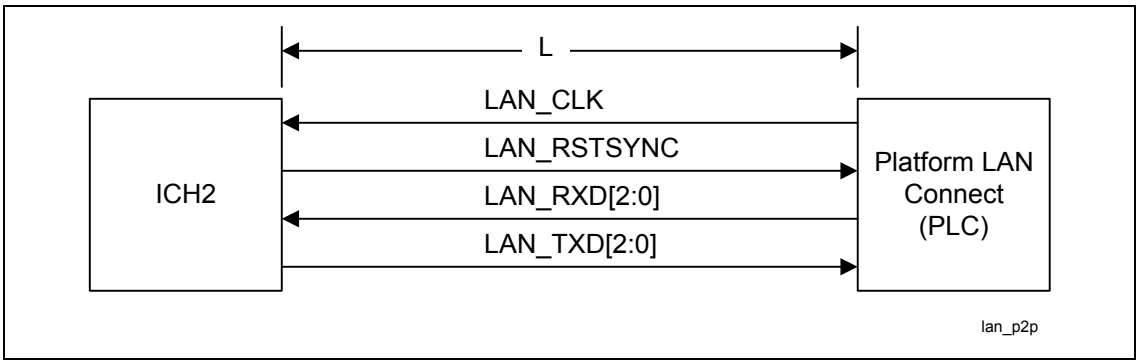


Table 30. Single-Solution Interconnect Length Requirements

Configuration	L	Comment
Intel® 82562EH	4.5” to 10.0”	Signal lines LAN_RXD[2:1] and LAN_TXD[2:1] are not connected.
Intel® 82562ET	3.5” to 10.0”	
CNR	4.5” to 8.5”	The trace length from the connector to LOM should be 0.5” to 3.0”

8.17.1.3. LOM/CNR Interconnect

The following guidelines allow for an all-inclusive motherboard solution. This layout combines the LOM, dual-footprint, and CNR solutions. The resistor pack ensures that either a CNR option or a LAN on Motherboard option can be implemented at one time, as shown in the following figure, which shows the recommended trace routing lengths.

Figure 62. LOM/CNR Interconnect

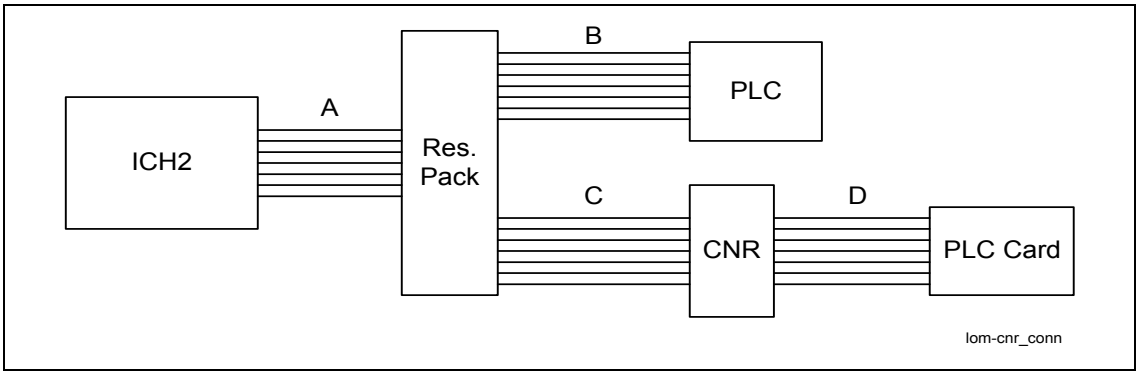


Table 31. LOM/CNR Length Requirements

Configuration	A	B	C	D
Intel® 82562EH	0.5” to 6.0”	4.0” to (10.0” – A)		
Intel® 82562ET	0.5” to 7.0”	3.0” to (10.0” – A)		
Dual footprint	0.5” to 6.5”	4.0” to (10.0” – A)		
Intel® 82562ET/EH card*	0.5” to 6.5”		2.5” to (9” – A)	0.5” to 3.0”

NOTES:
1. The total trace length should not exceed 13”.

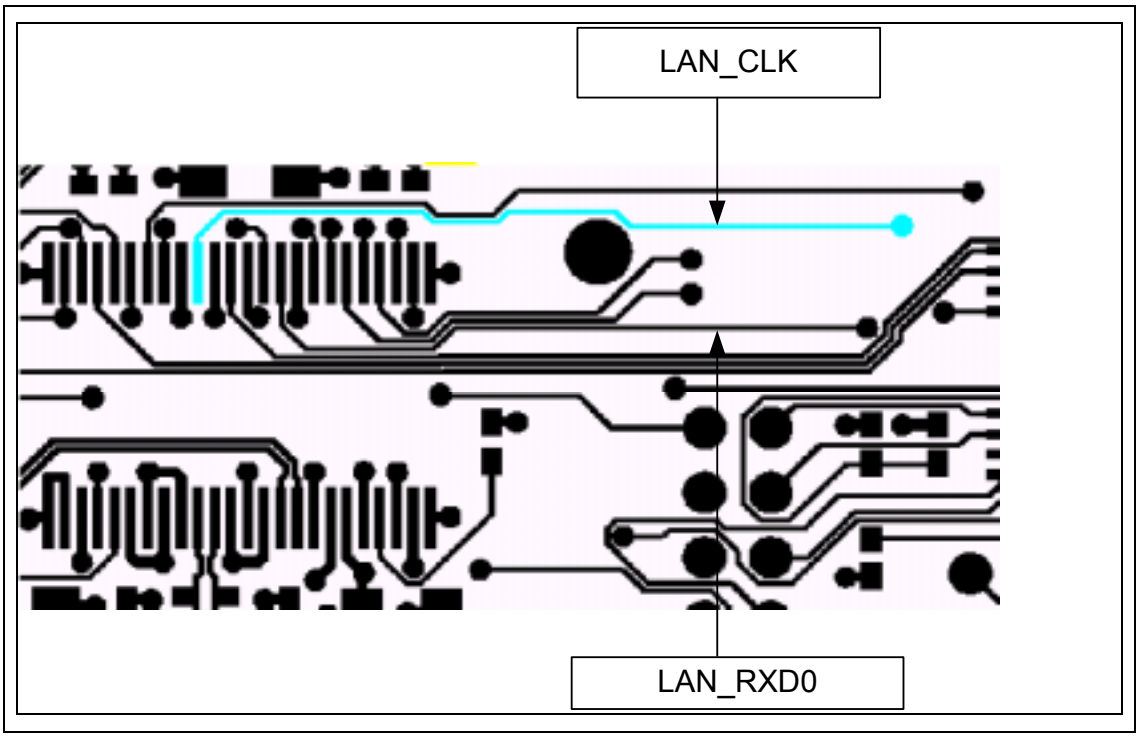
Additional guidelines for this configuration are as follows:

- Stubs due to the resistor pack should not be present on the interface.
- The resistor pack value can be 0 Ω or 22 Ω.
- LAN on Motherboard PLC can be a dual-footprint configuration.

8.17.1.4. Signal Routing and Layout

LAN connect signals must be carefully routed on the motherboard to meet the timing and signal quality requirements of this interface specification. The following are some of the general guidelines that should be followed. It is recommended that the board designer simulate the board routing, to verify that the specifications are met for flight times and skews due to trace mismatch and crosstalk. On the motherboard, the length of each data trace is either equal in length to the **LAN_CLK** trace or up to 0.5” shorter than the **LAN_CLK** trace. (**LAN_CLK** should always be the longest motherboard trace in each group.)

Figure 63. LAN_CLK Routing Example



8.17.1.5. Crosstalk Consideration

Noise due to crosstalk must be carefully minimized. Crosstalk is the main cause of timing skews and is the largest part of the t_{RMATCH} skew parameter.

8.17.1.6. Impedances

Motherboard impedances should be controlled to minimize the impact of any mismatch between the motherboard and the add-in card. An impedance of 60 Ω ± 15% is strongly recommended. Otherwise, signal integrity requirements may be violated.

8.17.1.7. Line Termination

Line termination mechanisms are not specified for the LAN connect interface. Slew-rate-controlled output buffers achieve acceptable signal integrity by controlling signal reflection, over/undershoot, and ringback. A 33-Ω series resistor can be installed at the driver side of the interface, if the developer has concerns about over/undershoot. Note that the receiver must allow for any drive strength and board impedance characteristic within the specified ranges.

8.17.2. General LAN Routing Guidelines and Considerations

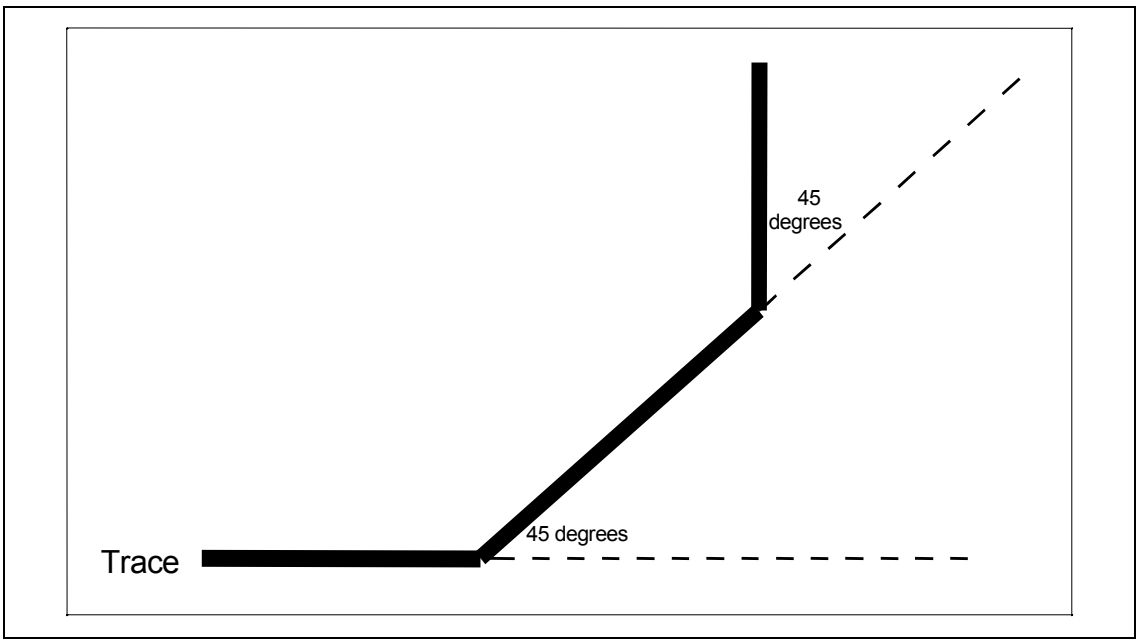
8.17.2.1. General Trace Routing Considerations

Trace routing considerations are important to minimize the effects of crosstalk and propagation delays on sections of the board where high-speed signals exist. Signal traces should be kept as short as possible to decrease interference from other signals, including those propagated through power and ground planes.

Observe the following suggestions to help optimize board performance:

- The maximum mismatch between the clock trace length and the length of any data trace is 0.5”.
- Maintain constant symmetry and spacing between the traces within a differential pair.
- Keep the signal trace lengths of a differential pair equal to each other.
- Keep the total length of each differential pair under 4”. (Many customer designs with differential traces longer than 5” have had one or more of the following issues: IEEE phy conformance failures, excessive EMI, and/or degraded receive BER.)
- Do not route the transmit differential traces closer than 70 mils from the receive differential traces.
- Do not route any other signal traces both parallel to the differential traces, and closer than 100 mils to the differential traces (300 mils recommended).
- Keep to 7 mils the maximum separation between differential pairs.
- For high-speed signals, the number of corners and vias should be kept to a minimum. If a 90° bend is required, it is recommended to use two 45 degree bends instead. Refer to Figure 64, *Trace Routing*
- Traces should be routed away from board edges by a distance greater than the trace height above the ground plane. This allows the field around the trace to couple more easily to the ground plane rather than to adjacent wires or boards.
- Do not route traces and vias under crystals or oscillators. This will prevent coupling to or from the clock. And as a general rule, place traces from clocks and drives at a minimum distance from apertures, by a distance exceeding the largest aperture dimension.

Figure 64. Trace Routing



8.17.2.1.1. Trace Geometry and Length

The key factors in controlling trace EMI radiation are the trace length and the ratio of trace width to trace height above the ground plane. To minimize trace inductance, high-speed signals and signal layers close to a ground or power plane should be as short and wide as practical. Ideally, this ratio of trace width to height above the ground plane is between 1:1 and 3:1. To maintain trace impedance, the width of the trace should be modified when changing from one board layer to another, if the two layers are not equidistant from the power or ground plane. Differential trace impedances should be controlled to ~100 Ω. It is necessary to compensate for trace-to-trace edge coupling, which can lower the differential impedance by 10 Ω, when the traces within a pair are closer than 0.030" (edge to edge).

Traces between decoupling and I/O filter capacitors should be as short and wide as practical. Long-and-thin traces are more inductive and would reduce the intended effect of the decoupling capacitors. For similar reasons, traces to I/O signals and signal terminations should be as short as possible. Vias to the decoupling capacitors should have diameters sufficiently large to decrease series inductance. Additionally, the PLC should not be closer than one inch to the connector/magnetics/edge of the board.

8.17.2.1.2. Signal Isolation

Comply with the following rules for signal isolation:

- Separate and group signals by function on separate layers if possible. Maintain a gap of 100 mils between all differential pairs (Phoneline and Ethernet) and other nets, but group associated differential pairs together.

Note: Over the length of the trace run, each differential pair should be at least 0.3" away from any parallel signal trace.

- Physically group together all components associated with one clock trace, to reduce trace length and radiation.
- Isolate I/O signals from high-speed signals to minimize crosstalk, which can increase EMI emission and susceptibility to EMI from other signals.
- Avoid routing high-speed LAN or Phoneline traces near other high-frequency signals associated with a video controller, cache controller, processor or other similar device.

8.17.2.2. Power and Ground Connections

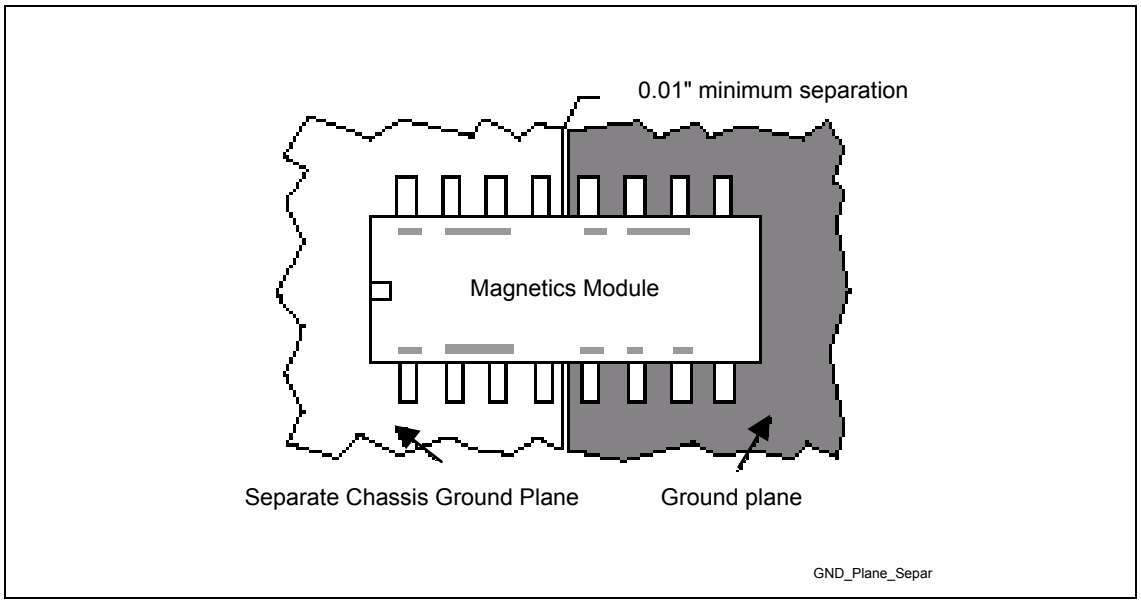
Comply with the following rules and guidelines for power and ground connections:

- All VCC pins should be connected to the same power supply.
- All VSS pins should be connected to the same ground plane.
- Use one decoupling capacitor per power pin for optimized performance.
- Place decoupling as close as possible to power pins.

8.17.2.2.1. General Power and Ground Plane Considerations

To properly implement the common-mode choke functionality of the magnetics module, the chassis or output ground (secondary side of transformer) should be physically separated from the digital or input ground (primary side) by at least 100 mils.

Figure 65. Ground Plane Separation



Good grounding requires minimizing inductance levels in the interconnections. Keeping ground returns short, signal loop areas small, and power inputs bypassed to signal return will significantly reduce EMI radiation.

Comply with the following rules to help reduce circuit inductance in both backplanes and motherboards:

- *Route traces over a continuous plane with no interruptions (i.e., do not route over a split plane).* If there are vacant areas on a ground or power plane, avoid routing signals over the vacant area. This will increase inductance and EMI radiation levels.
- To reduce coupling, separate noisy digital grounds from analog grounds.
- 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.
- *The ground plane beneath the filter/transformer module should be split. The RJ45 and/or RJ11 connector side of the transformer module should have chassis ground beneath it.* Splitting the ground planes beneath the transformer minimizes noise coupling between the primary and secondary sides of the transformer and between adjacent coils in the transformer. There should not be a power plane under the magnetics module.
- *Create a spark gap between pins 2 through 5 of the Phoneline connector(s) and shield ground of 1. 6 mm (59.0 mil).* This is a ***critical*** requirement needed to past FCC part 68 testing for Phoneline connection. Note: For worldwide certification a trench of 2.5 mm is required. In North America, the spacing requirement is 1.6 mm. However, home networking can be used in other parts of the world, including Europe, where some Nordic countries require the 2.5-mm spacing.

8.17.2.3. A 4-Layer Board Design

Top-Layer Routing

Sensitive analog signals are routed completely on the top layer without the use of vias. This allows tight control of signal integrity and removes any impedance inconsistencies due to layer changes.

Ground Plane

A layout split (100 mils) of the ground plane under the magnetics module between the primary and secondary side of the module is recommended. It is also recommended to minimize the digital noise injected into the 82562 common ground plane. Suggestions include optimizing decoupling on neighboring noisy digital components, isolating the 82562 digital ground using a ground cutout, etc.

Power Plane

Physically separate digital and analog power planes must be provided to prevent digital switching noise from being coupled into the analog power supply planes VDD_A. Analog power may be a metal fill “island,” separated from digital power, and better filtered than digital power.

Bottom-Layer Routing

Digital high-speed signals, which include all LAN interconnect interface signals, are routed on the bottom layer.

8.17.2.4. Common Physical Layout Issues

Common physical layer design and layout mistakes in LAN On Motherboard designs are as follows:

1. Unequal length of the two traces within a differential pair. Inequalities create common-mode noise and distort the transmit or receive waveforms.
2. Lack of symmetry between the two traces within a differential pair. (For each component and/or via that one trace encounters, the other trace must encounter the same component or a via at the same distance from the PLC.) Asymmetry can create common-mode noise, and distort the waveforms.
3. Excessive distance between the PLC and the magnetics or between the magnetics and the RJ-45/11 connector. Beyond a total distance of about 4 inches, it can become extremely difficult to design a spec-compliant LAN product. Long traces on FR4 (fiberglass epoxy substrate) will attenuate the analog signals. Also any impedance mismatch in the traces will be aggravated if they are longer (see #9 below). The magnetics should be as close to the connector as possible (<= 1 inch).
4. Routing any other trace parallel to and close to one of the differential traces. Crosstalk on the receive channel will induce degraded long-cable BER. When crosstalk gets onto the transmit channel, it can cause excessive emissions (below the FCC standard) and can cause poor transmit BER on long cables. Other signals should be kept at least 0.3” from the differential traces.
5. Routing the transmit differential traces next to the receive differential traces. The transmit trace closest to one of the receive traces will put more crosstalk onto the closest receive trace, which can greatly degrade the receiver's BER over long cables. After exiting the PLC, the transmit traces should be kept 0.3” or more away from the nearest receive trace. In the vicinities where the traces enter or exit the magnetics, the RJ-45/11 and the PLC are the only possible exceptions.
6. Use of an inferior magnetics module. The magnetics modules used by Intel have been fully tested for IEEE PLC conformance, long-cable BER problems, and emissions and immunity. (Inferior magnetics modules often have less common-mode rejection and/or no auto-transformer in the transmit channel.)
7. Another common mistake is using an 82555 or 82558 physical layer schematic in a PLC design. The transmit terminations and decoupling are different, and there also are differences in the receive circuit. Use the appropriate reference schematic or application notes.
8. Not using (or incorrectly using) the termination circuits for the unused pins at the RJ-45/11 and for the wire-side center-taps of the magnetics modules. These unused RJ pins and wire-side center-taps must be correctly referenced to chassis ground via the proper value resistor and capacitor or termination plane. If these are not terminated properly, there can be emission (FCC) problems, IEEE conformance issues, and long-cable noise (BER) problems. The application notes contain schematics that illustrate the proper termination for these unused RJ pins and the magnetics center-taps.
9. Incorrect differential trace impedances. It is important to have ~100 Ω impedance between the two traces within a differential pair. This becomes even more important as the differential traces become longer. It is very common to see customer designs that have differential trace impedances between 75 Ω and 85 Ω, even when the designers think they have designed for 100 Ω. (To calculate the differential impedance, many impedance calculators only multiply the single-ended impedance by two. This does not take into account edge-to-edge capacitive coupling between the two traces. When the two traces within a differential pair are kept close (see Note) to each other, the edge coupling can lower the effective differential impedance by 5 Ω to 20 Ω. A 10-Ω to 15-Ω drop in impedance is common.) Short traces will have fewer problems if the differential impedance is a little off.
10. Another common problem is to use a too-large capacitor between the transmit traces and/or too much capacitance from the magnetic's transmit center-tap (on the 82562ET side of the magnetics)

to ground. Using capacitors with capacitances exceeding a few pF in either of these locations can slow the 100-Mbps rise and fall times so much that they fail the IEEE rise time and fall time specs, which will cause the return loss to fail at higher frequencies and will degrade the transmit BER performance. Caution should be exercised if a cap is put in either of these locations. If a cap is used, it should almost certainly be less than 22 pF. (Reasonably good success has been achieved by using 6-pF to 12-pF values in past designs.) Unless there is some overshoot in the 100-Mbps mode, these caps are not necessary.

Note: It is important to keep the two traces within a differential pair close to each other. Keeping them close helps to make them more immune to crosstalk and other sources of common-mode noise. This also means lower emissions (i.e., FCC compliance) from the transmit traces, and better receive BER for the receive traces. Close should be considered to be less than 0.030 inch between the two traces within a differential pair. 0.007 inch trace-to-trace spacing is recommended.

8.17.3. Intel® 82562EH Home/PNA* Guidelines

Related Documents

Title	Location
Intel® 82562EH HomePNA 1-Mb/s Physical Layer Interface Datasheet (document number: 278313)	Intel's website for developers is at: http://developer.intel.com
Intel® 82562EH HomePNA 1-Mb/s Physical Layer Interface Brief Datasheet (document number: 278314)	Intel's website for developers is at: http://developer.intel.com

For correct LAN performance, designers must follow the general guidelines outlined in Section 8.17.2. Additional guidelines for implementing an 62EH Home/PNA* LAN connect component are as follows.

8.17.3.1. Power and Ground Connections

Obey the following rule for power and ground connections:

- For best performance, place decoupling capacitors on the back side of the PCB, directly under the 82562EH, with equal distance from both pins of the capacitor to power/ground.

The analog power supply pins for 82562EH (VCCA, VSSA) should be isolated from the digital VCC and VSS through the use of ferrite beads. In addition, adequate filtering and decoupling capacitors should be provided between VCC and VSS as well as the VCCA and VSSA power supplies.

8.17.3.2. Guidelines for Intel® 82562EH Component Placement

Component placement can affect the signal quality, emissions, and temperature of a board design. This section discusses guidelines for component placement.

Careful component placement can:

- Decrease potential problems directly related to electromagnetic interference (EMI), which could result in failure to meet FCC specifications.

- Simplify the task of routing traces. To some extent, component orientation will affect the complexity of trace routing. The overall objective is to minimize turns and crossovers between traces.

It is important to minimize the space needed for the HomePNA LAN interface, because all other interfaces will compete for physical space on a motherboard near the connector edge. As with most subsystems, the HomePNA LAN circuits must be as close as possible to the connector. Thus, it is imperative that all designs be optimized to fit in a very small space.

8.17.3.3. Crystals and Oscillators

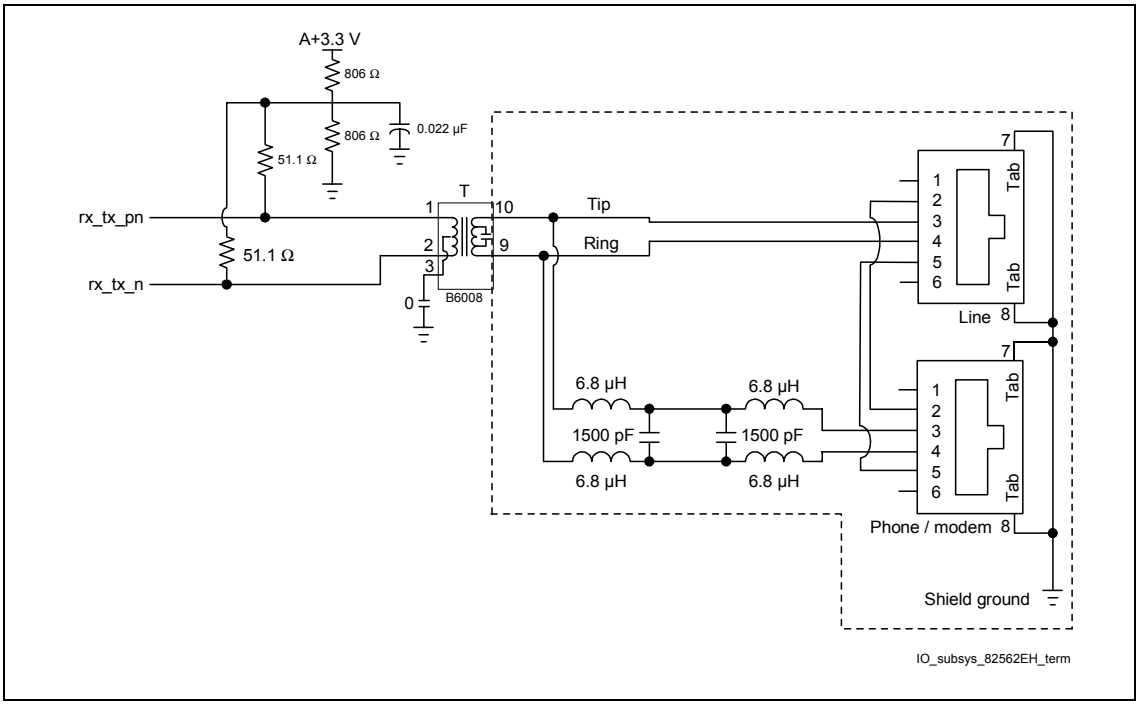
To minimize the effects of EMI, clock sources should not be placed near I/O ports or board edges. Radiation from these devices may be coupled onto the I/O ports or out of the system chassis. Crystals should also be kept away from the HomePNA magnetics module to prevent communication interference. If they exist, the crystal's retaining straps should be grounded to prevent the possibility of radiation from the crystal case, and the crystal should lie flat against the PC board to provide better coupling of the electromagnetic fields to the board.

For noise-free and stable operation, place the crystal and associated discrete components as close as possible to the 82562EH. Minimize the length and do not route any noisy signals in this area.

8.17.3.4. Phoneline HPNA Termination

The transmit/receive differential signal pair is terminated with a pair of 51.1-Ω (1%) resistors. This parallel termination should be placed close to the 82562EH. The center, common point between the 51.1-Ω resistors is connected to a voltage-divider network. The termination is shown in the following figure.

Figure 66. Intel® 82562EH Termination



The filter and magnetics component T1 integrates the required filter network, high-voltage impulse protection, and transformer to support the HomePNA LAN interface.

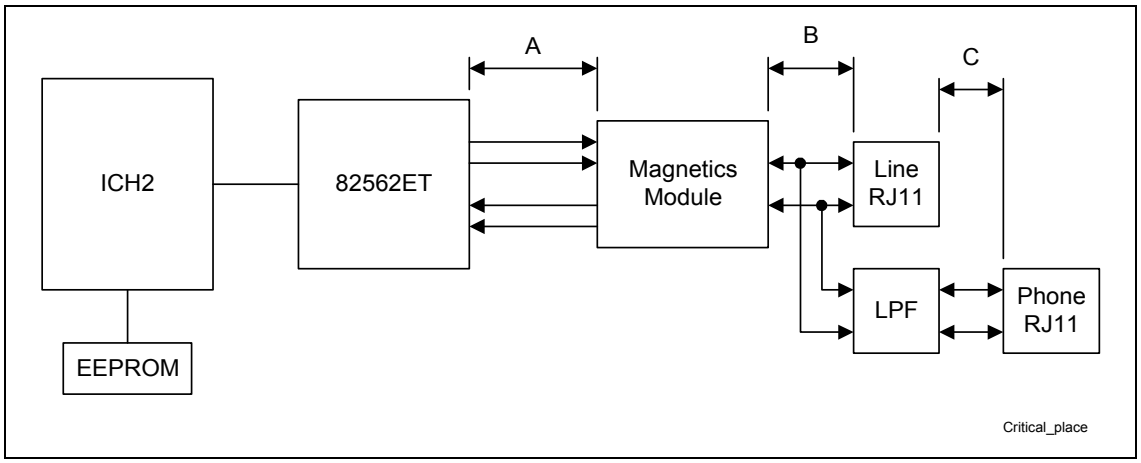
One RJ-11 jack (labeled “LINE” in the previous figure) allows the node to be connected to the Phoneline, and the second jack (labeled “PHONE” in the previous figure) allows other down-line devices to be connected at the same time. This second connector is not required by the HomePNA. However, typical PCI adapters and PC motherboard implementations are likely to include it for user convenience.

A low-pass filter, setup in-line with the second RJ-11 jack, also is recommended by the HomePNA to minimize interference between the HomeRun connection and a POTs voice or modem connection on the second jack. This restricts of the type of devices connected to the second jack as the pass-band of this filter is set approximately at 1.1 MHz. Refer to the HomePNA website (www.homepna.org) for up-to-date information and recommendations regarding the use of this low-pass filter to meet HomePNA certifications.

8.17.3.5. Critical Dimensions

There are three dimensions to consider during layout. Distance ‘B’ from the line RJ11 connector to the magnetics module, distance ‘C’ from the phone RJ11 to the LPF (if implemented), and distance ‘A’ from 82562EH to the magnetics module (see the following figure).

Figure 67. Critical Dimensions for Component Placement



Distance	Priority	Guideline
B	1	< 1”
A	2	< 1”
C	3	< 1”

8.17.3.5.1. Distance from Magnetics Module to Line RJ11

This distance ‘B’ should be given highest priority and should be less then 1”. Regarding trace symmetry, route differential pairs with consistent separation and with exactly the same lengths and physical dimensions.

Asymmetrical and unequally long differential pairs contribute to common-mode noise. This can degrade the receive circuit performance and contribute to emissions radiated from the transmit side.

8.17.3.5.2. Distance from Intel® 82562EH to Magnetics Module

Due to the high speed of signals present, distance 'A' between the 82562EH and the magnetics should also be less than 1", but should be second priority relative to distance from connects to the magnetics module.

Generally speaking, any section of trace intended for use with high-speed signals should be subject to proper termination practices. Proper signal termination can reduce reflections caused by impedance mismatches between the device and traces route. The reflections of a signal may have a high-frequency component that may contribute more EMI than the original signal itself.

8.17.3.5.3. Distance from LPF to Phone RJ11

This distance 'C' should be less then 1". Regarding trace symmetry, route differential pairs with consistent separation and with exactly the same lengths and physical dimensions.

Asymmetrical and unequally long differential pairs contribute to common-mode noise. This can degrade the receive circuit performance and contribute to emissions radiated from the transmit side

8.17.4. Intel® 82562ET / 82562EM Guidelines

Related Documents

- *82562ET Platform LAN Connect (PLC) Datasheet*
- *PCB Design for the 82562 ET/EM Platform LAN Connect*

For correct LAN performance, designers must follow the general guidelines outlined in Section 8.17.2. Additional guidelines for implementing an 82562ET or 82562EM LAN connect component are as follows.

8.17.4.1. Guidelines for Intel® 82562ET / 82562EM Component Placement

Component placement can affect the signal quality, emissions, and temperature of a board design. This section provides guidelines for component placement.

Careful component placement can:

- Decrease potential problems directly related to electromagnetic interference (EMI), which could result in failure to meet FCC and IEEE test specifications.
- Simplify the task of routing traces. To some extent, component orientation will affect the complexity of trace routing. The overall objective is to minimize turns and crossovers between traces.

It is important to minimize the space needed for the Ethernet LAN interface, because all other interfaces will compete for physical space on a motherboard near the connector edge. As with most subsystems, the Ethernet LAN circuits must be as close as possible to the connector. Thus, it is imperative that all designs be optimized to fit in a very small space.

8.17.4.2. Crystals and Oscillators

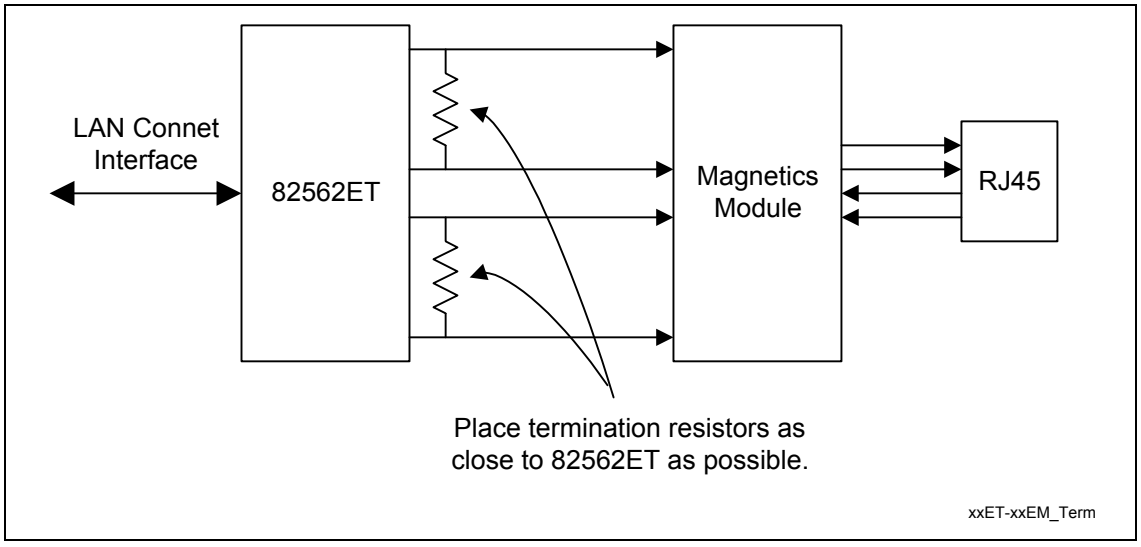
To minimize the effects of EMI, clock sources should not be placed near I/O ports or board edges. Radiation from these devices may be coupled onto the I/O ports or out of the system chassis. Crystals should also be kept away from the Ethernet magnetics module to prevent interference with communication. If they exist, the retaining straps of the crystal should be grounded to prevent possible radiation from the crystal case. Also, the crystal should lie flat against the PC board to provide better coupling of the electromagnetic fields to the board.

For noise-free and stable operation, place the crystal and associated discrete components as close as possible to the 82562ET or 82562EM. Keep the trace length as short as possible and do not route any noisy signals in this area.

8.17.4.3. Intel® 82562ET / 82562EM Termination Resistors

The 100-Ω (1%) resistor used to terminate the differential transmit pairs (TDP/TDN) and the 100-Ω (1%) receive differential pairs (RDP/RDN) should be placed as close as possible to the LAN connect component (82562ET or 82562EM). This is due to the fact that these resistors terminate the entire impedance seen at the termination source (i.e., 82562ET), including the wire impedance reflected through the transformer.

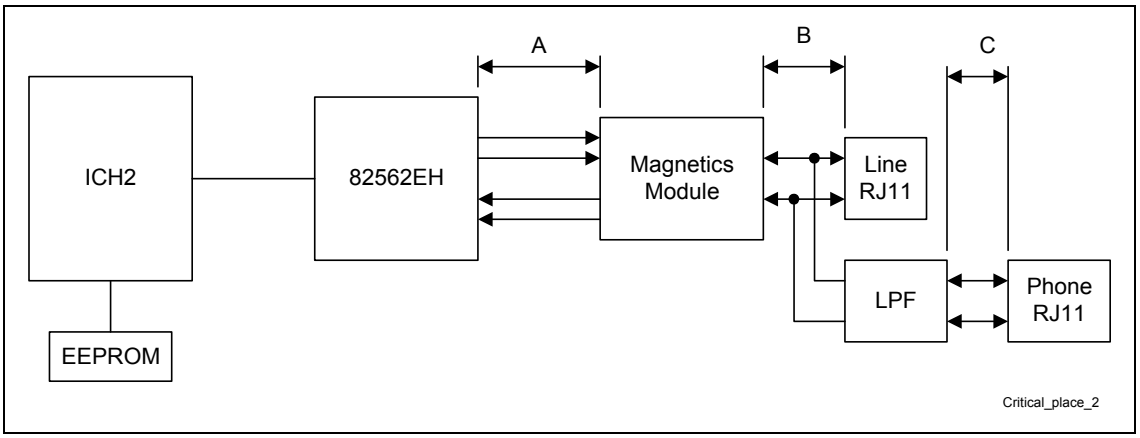
Figure 68. Intel® 82562ET/82562EM Termination



8.17.4.4. Critical Dimensions

Figure 69. Critical Dimensions for Component Placement

There are two dimensions to consider during layout. Distance ‘B’ from the line RJ45 connector to the magnetics module and distance ‘A’ from the 82562ET or 82562EM to the magnetics module (see the following figure).



Distance	Priority	Guideline
A	1	< 1”
B	2	< 1”

Distance from Magnetics Module to RJ45

The distance A in the previous figure should be given the highest priority in board layout. The separation between the magnetics module and the RJ45 connector should be kept less than 1”. The following trace characteristics are important and should be observed:

- Differential impedance: The differential impedance should be 100 Ω. The single-ended trace impedance will be approximately 50 Ω. However, the differential impedance can also be affected by the spacing between the traces.
- Trace Symmetry: Differential pairs (e.g., TDP and TDN) should be routed with consistent separation and with exactly the same lengths and physical dimensions (e.g., width).

Caution: Asymmetric and unequal length traces in the differential pairs contribute to common-mode noise. This can degrade the receive circuit’s performance and contribute to emissions radiated from the transmit circuit. If the 82562ET must be placed farther than a couple of inches from the RJ45 connector, distance B can be sacrificed. It should be a priority to keep the total distance between the 82562ET and RJ-45 as short as possible.

Note: The measured trace impedance for layout designs targeting 100 Ω often result in lower actual impedance. OEMs should verify actual trace impedance and adjust their layouts accordingly. If the actual impedance is consistently low, a target of 105–110 Ω should compensate for second-order effects.

Distance from Intel® 82562ET to Magnetics Module

Distance B should also be designed to be less than 1” between devices. The high-speed nature of the signals propagating through these traces requires that the distance between these components be closely observed. In general, any section of traces intended for use with high-speed signals should be subject to proper termination practices. Proper termination of signals can reduce reflections caused by impedance mismatches between device and traces. The reflections of a signal may have a high-frequency component that contributes more EMI than the original signal itself. For this reason, these traces should be designed to a 100-Ω differential value. These traces should also be symmetric and of equal length within each differential pair.

8.17.4.5. Reducing Circuit Inductance

The following guidelines show how to reduce circuit inductance in both backplanes and motherboards. Traces should be routed over a continuous ground plane with no interruptions. If there are vacant areas on a ground or power plane, the signal conductors should not cross the vacant area. This increases inductance and associated radiated noise levels. Noisy logic grounds should be separated from analog signal grounds to reduce coupling. Noisy logic 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. Similarly, every power via should be connected to all power planes at equal potential. This helps reduce circuit inductance. Another recommendation is to physically locate grounds so as to minimize the loop area between a signal path and its return path. Rise and fall times should be as slow as possible. Because signals with fast rise and fall times contain many high-frequency harmonics, that can radiate significantly. The most sensitive signal returns closest to the chassis ground should be connected together. This will result in a smaller loop area and reduce the likelihood of crosstalk. The effect of different configurations on the amount of crosstalk can be studied using electronics modeling software.

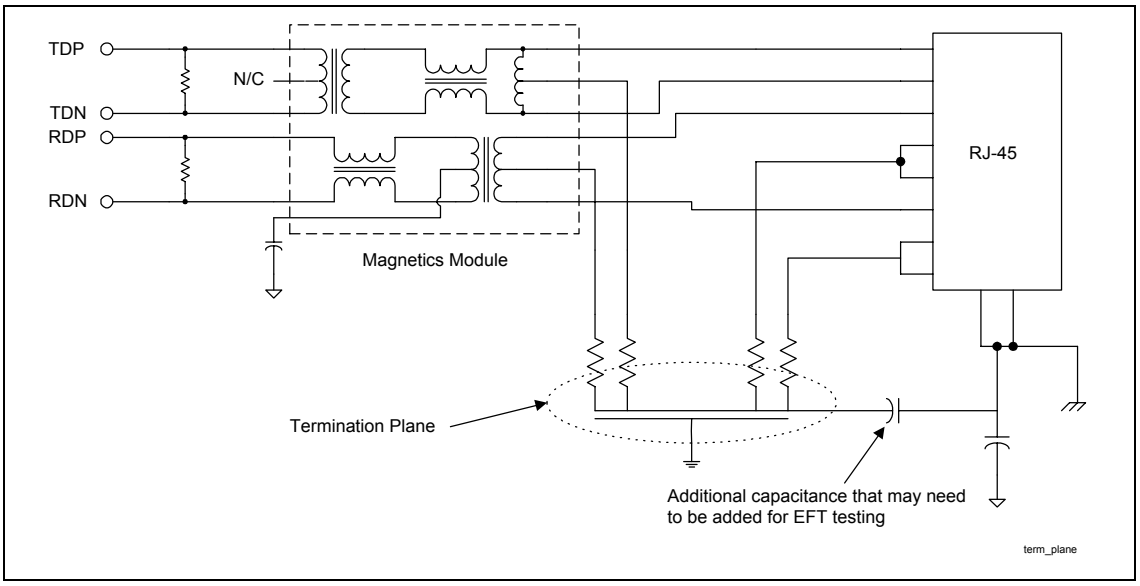
Terminating Unused Connections

In Ethernet designs, it is common practice to terminate to ground both unused connections on the RJ-45 connector and the magnetics module. Depending on the overall shielding and grounding design, this may be done to the chassis ground, signal ground or a termination plane. Care must be taken when using various grounding methods to ensure that emission requirements are met. The method most often implemented is called the “Bob Smith” termination. In this method, a floating termination plane is cut out of a power plane layer. This floating plane acts as a plate of a capacitor with an adjacent ground plane. The signals can be routed through 75-Ω resistors to the plane. Stray energy on unused pins is then carried to the plane.

Termination Plane Capacitance

The recommended minimum termination plane capacitance is 1500 pF. This helps reduce the amount of crosstalk on the differential pairs (TDP/TDN and RDP/RDN) from the unused pairs of the RJ45. Pads may be placed for an additional capacitance to chassis ground, which may be required if the termplane capacitance is not large enough to pass EFT (electrical fast transient) testing. If a discrete capacitor is used, it should be rated for at least 1000 Vac, to satisfy the EFT requirements.

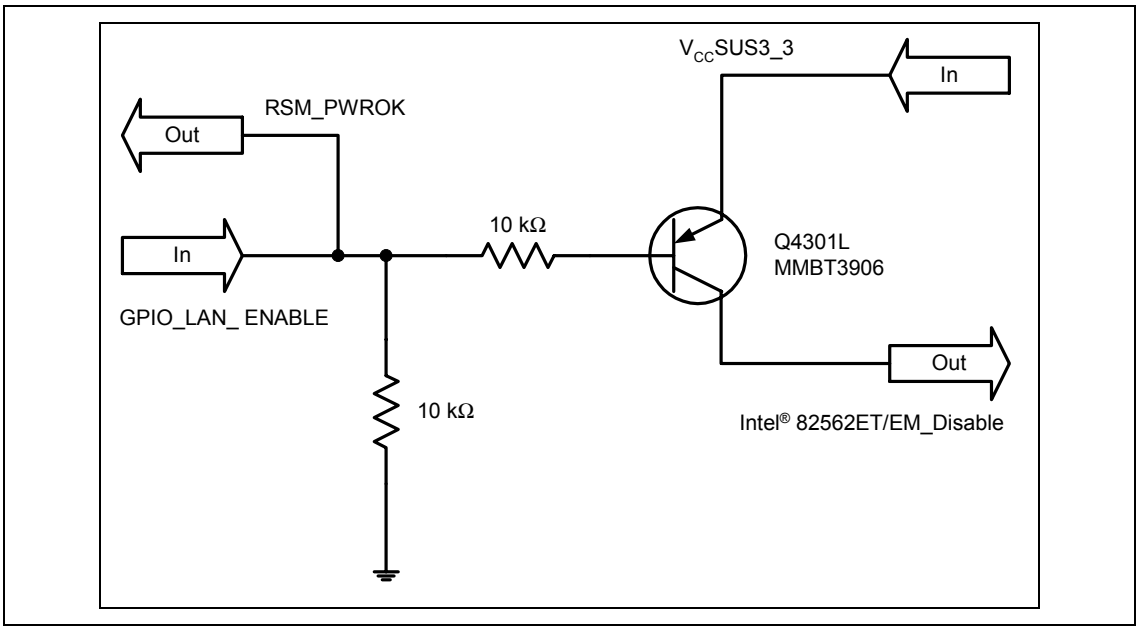
Figure 70. Termination Plane



8.17.4.6. Intel® 82562ET/EM Disable Guidelines

To disable the 82562ET/EM, the device must be isolated (disabled) prior to reset (RSM_PWROK) asserting. Using a GPIO, such as GPO28 to be LAN_Enable (enabled high), LAN will default to enabled on initial power-up and after an AC power loss. This circuit shown below will allow this behavior. The BIOS, by controlling the GPIO, can disable the LAN microcontroller.

Figure 71. Intel® 82562ET/EM Disable Circuit



There are four pins which are used to put 82562ET/EM controller in different operating states: Test_En, Isol_Tck, Isol_Ti, and Isol_Tex. The table below describes the operational/disable features for this design.

Table 32. Intel® 82562ET/EM Control Signals

Test_En	Isol_Tck	Isol_Ti	Isol_Tex	State
0	0	0	0	Enabled
0	1	1	1	Disabled w/ Clock (low power)
1	1	1	1	Disabled w/out Clock (lowest power)

The four control signals shown in the above table should be configured as follows: Test_En should be pulled-down through a 100 Ω resistor. The remaining three control signals should each be connected through 100 Ω series resistors to the common node "82562ET/EM_Disable" of the disable circuit.

8.17.5. Intel® 82562ET / 82562EH Dual Footprint Guidelines

These guidelines characterize the proper layout for a dual-footprint solution. This configuration enables the developer to install either the 82562EH or the 82562ET/82562EM components, while using only one motherboard design. The following guidelines are for the 82562ET/82562EH dual-footprint option. The guidelines called out in Section 8.17.2 apply to this configuration. The dual footprint for this particular solution uses a SSOP footprint for 82562ET and a TQFP footprint for 82562EH. The combined footprint for this configuration is shown in the following two figures.

Figure 72. Dual-Footprint LAN Connect Interface

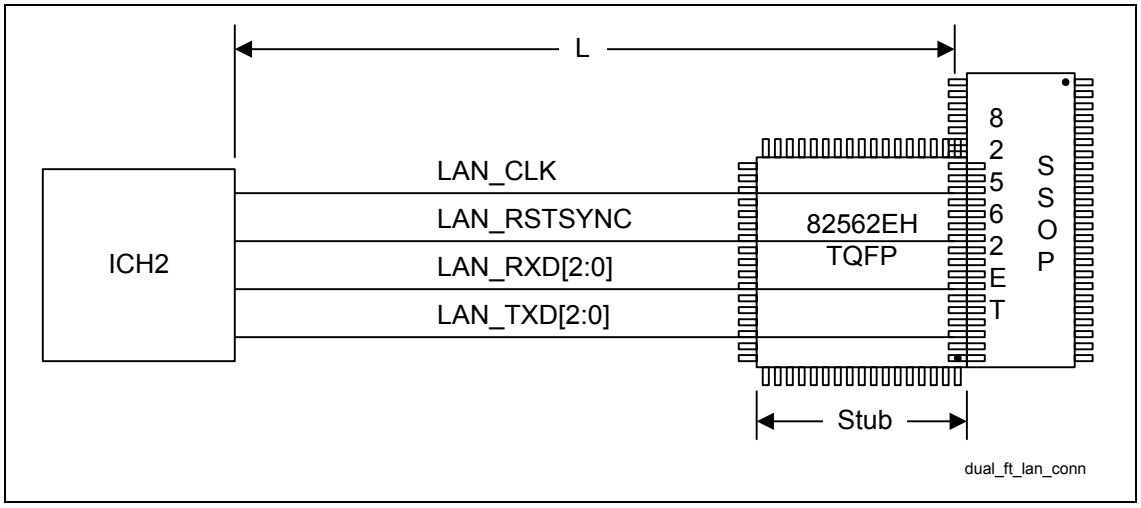
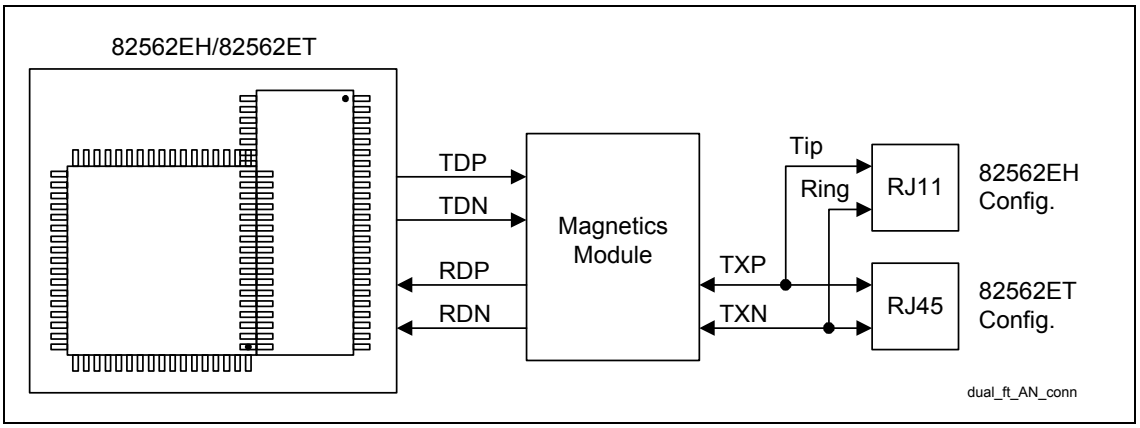


Figure 73. Dual-Footprint Analog Interface



The following are additional guidelines for this configuration:

- L = 3.5” to 10.0”
- Stub < 0.5”
- Either 82562EH or 82562ET/82562EM can be installed, but not both.
- 82562ET pins 28,29, and 30 overlap with 82562EH pins 17,18, and 19.
- Overlapping pins are tied to ground.
- No other signal pads should overlap or touch.
- Signal lines LAN_CLK, LAN_RSTSYNC, LAN_RXD[0], LAN_TXD[0], RDP, RDN, RXP/Ring, and RXN/Tip are shared by the 82562EH and 82562ET configurations.
- No stubs should be present when 82562ET is installed.
- Packages used for the dual footprint are TQFP for 82562EH and SSOP for 82562ET.
- A 22-Ω resistor can be placed at the driving side of the signal line to improve signal quality on the LAN connect interface.
- Resistor should be placed as close as possible to the component.
- Use components that can satisfy both the 82562ET and 82562EH configurations (i.e., magnetics module).
- Install components for either the 82562ET or the 82562EH configuration. Only one configuration can be installed at a time.
- Route shared signal lines such that stubs are not present or are kept to a minimum.
- Stubs may occur on shared signal lines (i.e RDP and RDN). These stubs are due to traces routed to an uninstalled component.
- Use 0-Ω resistors to connect and disconnect circuitry not shared by both configurations. Place resistor pads along the signal line to reduce stub lengths.

8.18. LPC/FWH

The following provides general guidelines for compatibility and design recommendations for supporting the FWH device. The majority of the changes will be incorporated in the BIOS. Refer to the FWH BIOS Specification or equivalent

8.18.1. In-Circuit FWH Programming

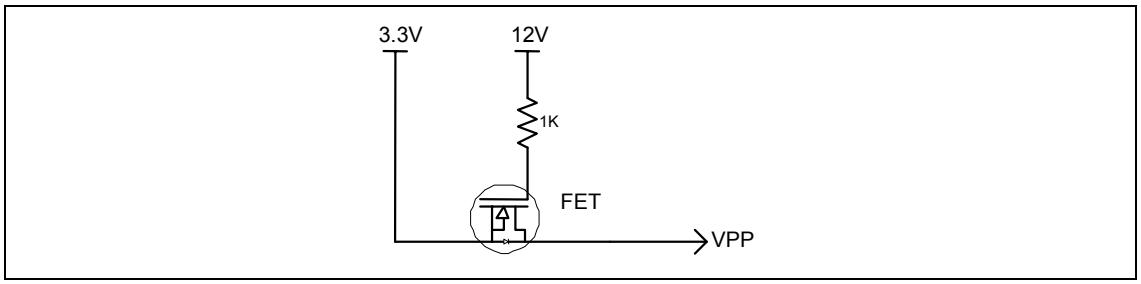
All cycles destined for the FWH will appear on the PCI. The ICH2 hub interface-to-PCI Bridge puts all processor boot cycles out on the PCI (before sending them out on the FWH interface). If the ICH2 is set for subtractive decode, these boot cycles can be accepted by a positive decode agent out on PCI. This enables booting from a PCI card that positively decodes these memory cycles. To boot from a PCI card, it is necessary to keep the ICH2 in the subtractive decode mode. If a PCI boot card is inserted and the ICH2 is programmed for positive decode, two devices will positively decode the same cycle. In systems with the 82380AB (ISA bridge), it is also necessary to keep the NOGO signal asserted when booting from a PCI ROM. Note that it is not possible to boot from a ROM behind the 82380AB. Once you have booted from the PCI card, you potentially could program the FWH in circuit and program the ICH2 CMOS.

8.18.2. FWH Vpp Design Guidelines

The Vpp pin on the FWH is used for programming the flash cells. The FWH supports a Vpp of 3.3 V or 12 V. If Vpp is 12 V, the flash cells will program about 50% faster than at 3.3 V. However, the FWH only supports 12 Vpp for 80 hours. The 12 Vpp would be useful in a programmer environment that is typically an event that occurs very infrequently (much less than 80 hours). The VPP pin MUST be tied to 3.3 V on the motherboard.

In some instances, it is desirable to program the FWH during assembly with the device soldered down on the board. In order to decrease programming time it becomes necessary to apply 12 V to the V_{PP} pin. The following circuit will allow testers to put 12 V on the V_{PP} pin while keeping this voltage separated from the 3.3 V plane to which the rest of the power pins are connected. This circuit also allows the board to operate with 3.3 V on this pin during normal operation.

Figure 74. FWH VPP Isolation Circuitry



8.18.3. FWH Decoupling

A 0.1 µF capacitor should be placed between the Vcc supply pins and the Vss ground pins to decouple high frequency noise, which may affect the programmability of the device. Additionally, a 4.7 µF

capacitor should be placed between the Vcc supply pins and the Vss ground pin to decouple low frequency noise. The capacitors should be placed no further than 390 mils from the Vcc supply pins.

This page is intentionally left blank.

9. Clocking

For an 815EP chipset-based system, there are two clock specifications. One is for a two-DIMM solution, and the other is for a three-DIMM solution.

9.1. 2-DIMM Clocking

9.1.1. Clock Generation

Table 33. Intel® CK815E (2-DIMM) Clocks

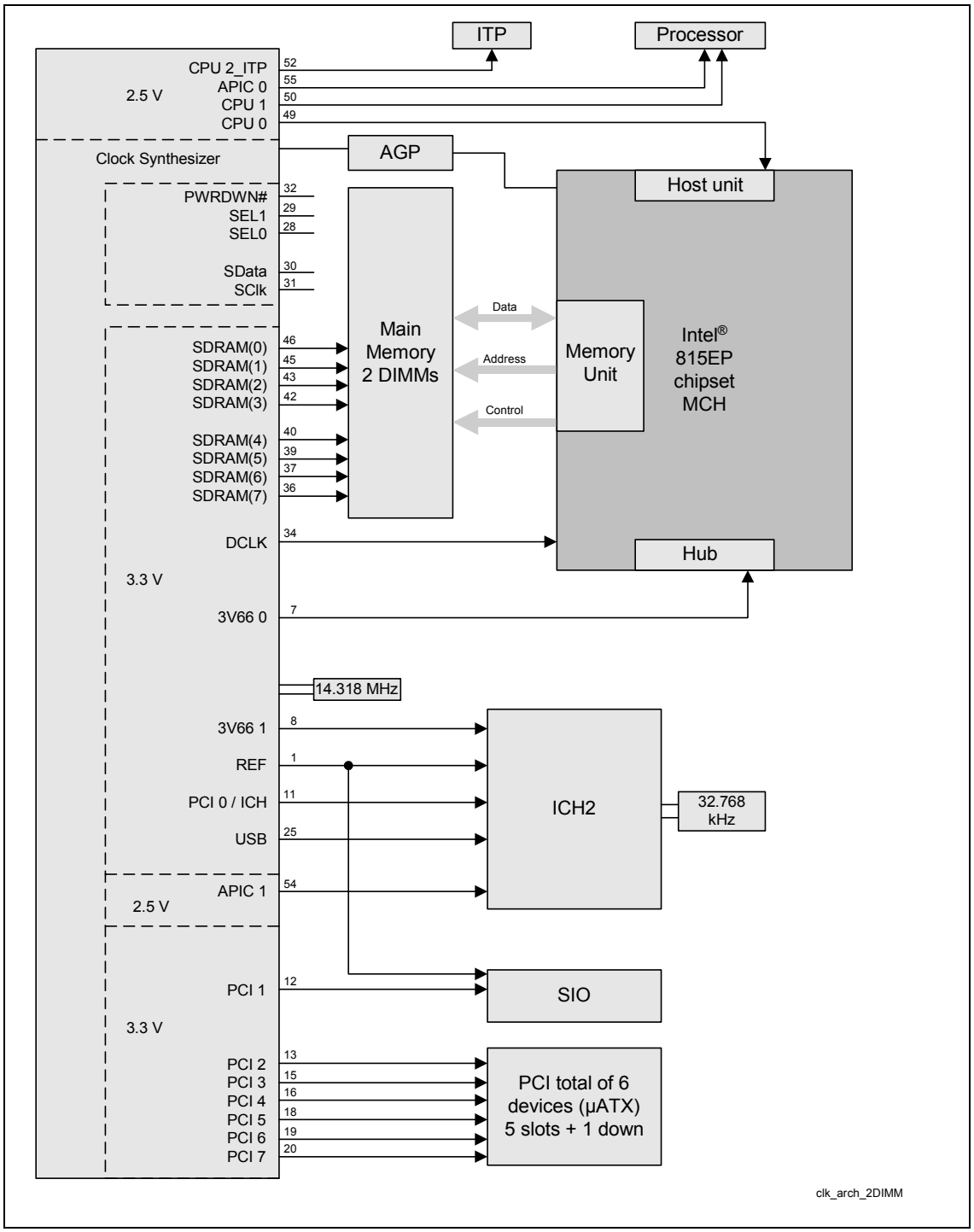
Number	Clock	Frequency
3	processor clocks	66/100/133 MHz
9	SDRAM clocks	100 MHz
7	PCI clocks	33 MHz
2	APIC clocks	16.67/33 MHz
2	48-MHz clocks	48 MHz
3	3-V, 66-MHz clocks	66 MHz
1	REF clock	14.31818 MHz

Features (56-pin SSOP package)

- 9 copies of 100-MHz SDRAM clocks (3.3 V) [SDRAM0…7, DClk]
- 7 copies of PCI clock (33 MHz) (3.3 V)
- 2 copies of APIC clock @ 33 MHz, synchronous to processor clock (2.5 V)
- 1 copy of 48-MHz USB clock (3.3 V) (non-SSC) (type 3 buffer)
- 1 copy of 48-MHz DOT clock (3.3 V) (non-SSC) (see DOT details)
- 3 copies of 3-V, 66-MHz clock (3.3 V)
- 1 copy of REF clock @ 14.31818 MHz (3.3 V)
- Ref. 14.31818-MHz xtal oscillator input
- Power-down pin
- Spread-spectrum support
- IIC support for turning off unused clocks

9.1.2. 2-DIMM Clock Architecture

Figure 75. Intel® 815EP Chipset Clock Architecture



9.2. 3-DIMM Clocking

9.2.1. Clock Generation

Table 34. Intel® CK815E (3-DIMM) Clocks

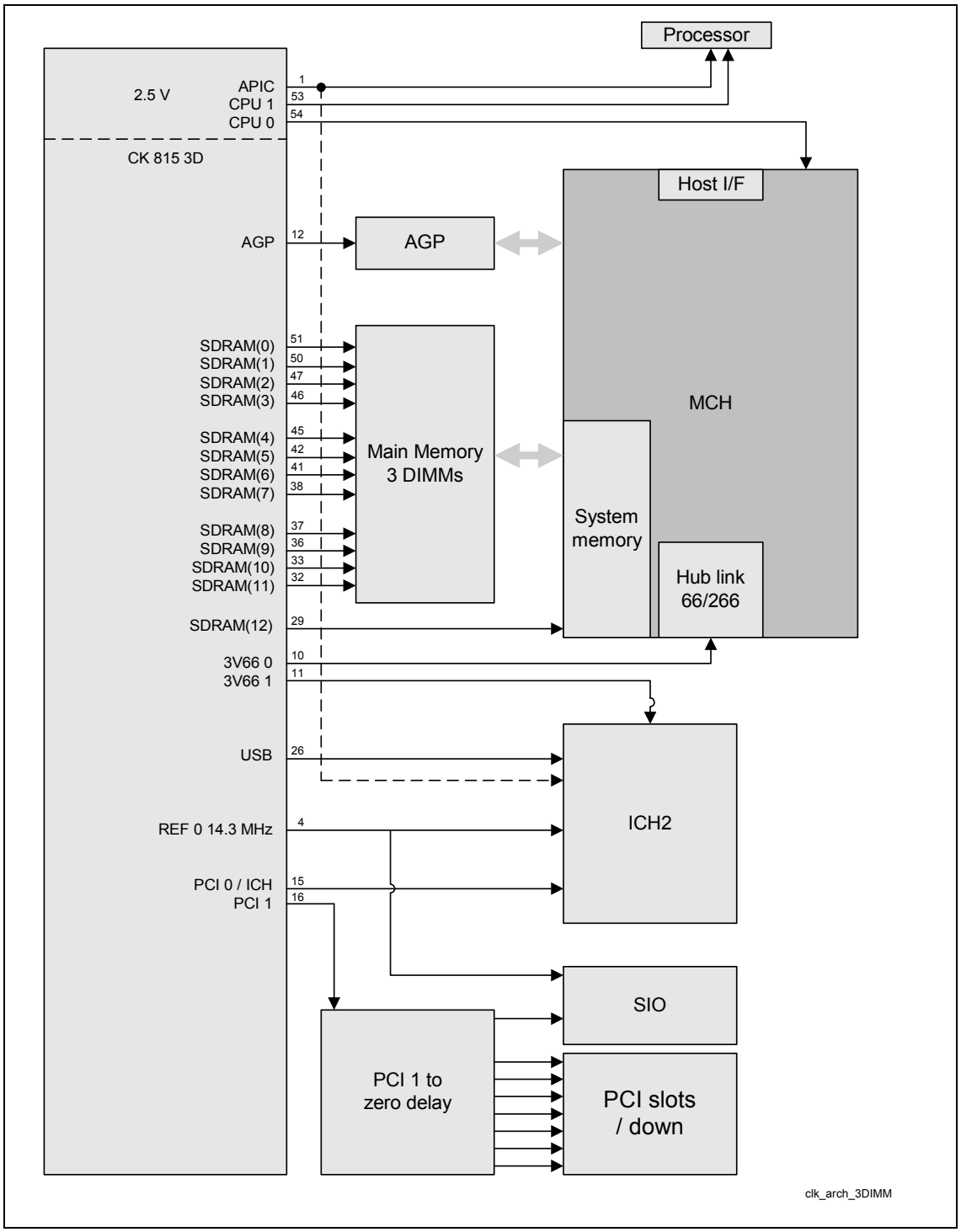
Number	Clock	Frequency
2	processor clocks	66/100/133 MHz
13	SDRAM clocks	100 MHz
2	PCI clocks	33 MHz
1	APIC clocks	33 MHz
2	48-MHz clocks	48 MHz
3	3-V, 66-MHz clocks	66 MHz
1	REF clock	14.31818 MHz

Features (56-pin SSOP package)

- 13 copies of SDRAM clocks
- 2 copies of PCI clock
- 1 copy of APIC clock
- 1 copy of 48-MHz USB clock (3.3 V) (non-SSC) (type 3 buffer)
- 1 copy of 48-MHz DOT clock (3.3 V) (non-SSC) (see DOT details)
- 3 copies of 3-V, 66-MHz clock (3.3 V)
- 1 copy of ref. clock @ 14.31818 MHz (3.3 V)
- Ref. 14.31818-MHz xtal oscillator input
- Spread-spectrum support
- IIC support for turning off unused clocks

9.2.2. 3-DIMM Clock Architecture

Figure 76. Intel® 815EP Chipset Clock Architecture



9.3. Clock Routing Guidelines

This section presents the generic clock routing guidelines for both 2-DIMM and 3-DIMM boards. For 3-DIMM boards, additional analysis must be performed by the motherboard designer to ensure that the clocks generated by the external PCI clock buffer meet the PCI specifications for clock skew at the receiver, when compared with the PCI clock at the ICH2.

Figure 77. Clock Routing Topologies

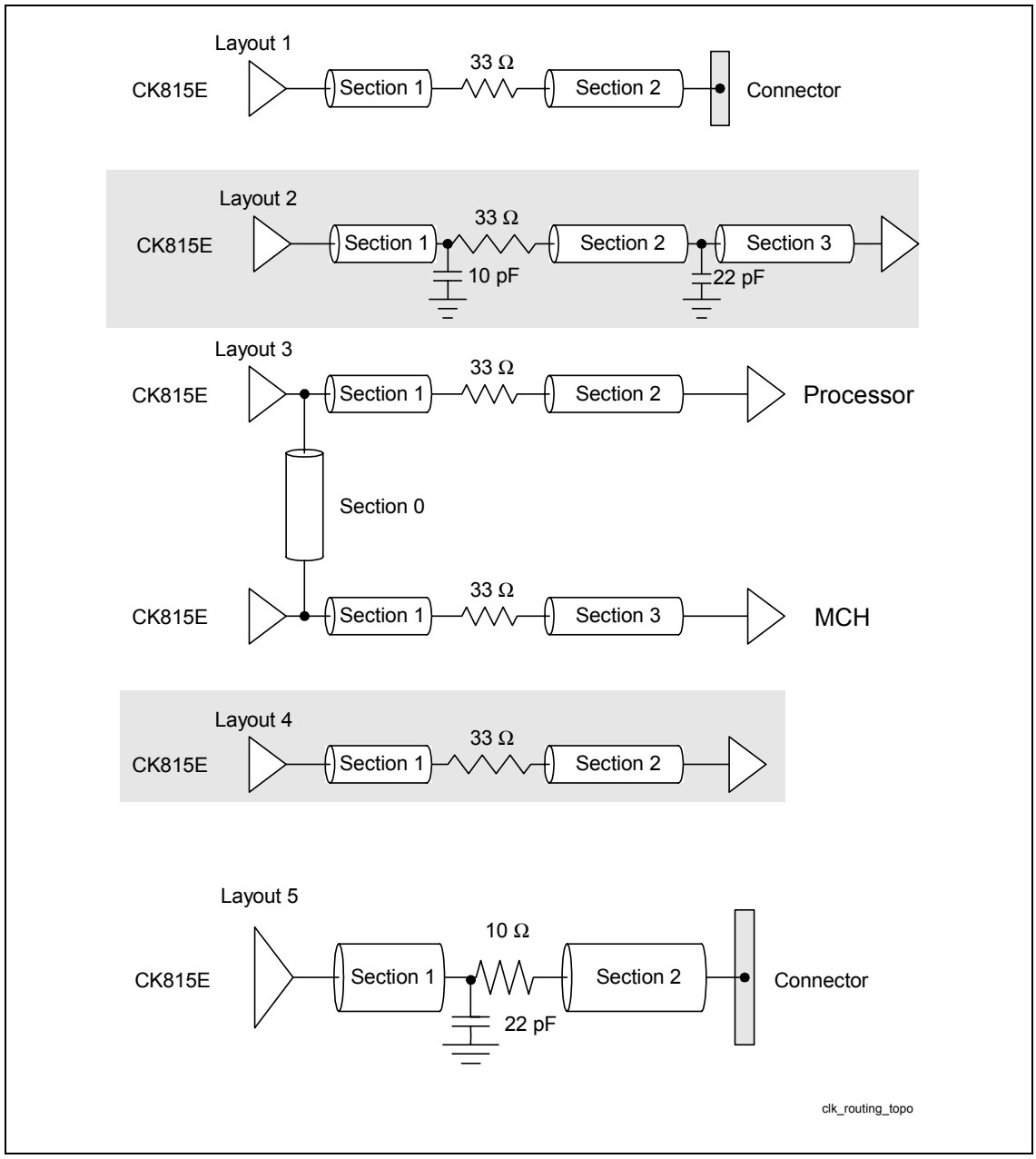


Table 35. Simulated Clock Routing Solution Space

Destination	Topology from Previous Figure	Section 0 Length	Section 1 Length	Section 2 Length	Section 3 Length
SDRAM MCLK	Layout 5	N/A	< 0.5”	A[1]	N/A
MCH SCLK[3]	Layout 2	N/A	< 0.5”=L1	A + 3.5”-L1	0.5”
Processor BCLK	Layout 3	< 0.1”	< 0.5”	A + 5.2”	A + 8”
MCH HCLK			<0.5”		
MCH HUBCLK	Layout 4	N/A	<0.5”	A + 8”	N/A
Intel® ICH2 HUBCLK	Layout 4	N/A	<0.5”	A + 8”	N/A
ICH2 PCICLK	Layout 4	N/A	<0.5”	A + 8”	N/A
AGP CLK	Layout 4	N/A	<0.5”	A + 3" to A + 4"	N/A
PCI down[2]	Layout 4	N/A	<0.5”	A + 8.5” to A + 14”	N/A
PCI slot[2]	Layout 1	N/A	<0.5”	A + 5” to A + 11”	

NOTES:
1. Length “A” has been simulated up to 6”.
2. All PCI clocks must be within 6” of the ICH2 PCICLK route length. Routing on PCI add-in cards must be included in this length. In the presented solution space, ICH2 PCICLK was considered to be the shortest in the 6” trace routing range, and other clocks were adjusted from there. The system designer may choose to alter the relationship of PCI device and slot clocks, as long as all PCI clock lengths are within 6”. Note that the ICH2 PCICLK length is fixed to meet the skew requirements of ICH2 PCICLK to ICH2 HUBCLK.
3. 22pF Load cap should be placed 0.5” from the MCH pin.

General Clock Layout Guidelines

- All clocks should be routed 5 mils wide with 15-mil spacing to any other signals.
- It is recommended to place capacitor sites within 0.5” of the receiver of all clocks. They are useful in system debug and AC tuning.
- Series resistor for clock guidelines: 22 Ω for MCH SCLK and 10 Ohms SDRAM clocks. All other clocks use 33 Ω.
- Each DIMM clock should be matched within ± 10 mils.

Clock Decoupling

Several general layout guidelines should be followed when laying out the power planes for the CK815E clock generator, as follows:

- Isolate power planes to the each of the clock groups.
- Place local decoupling as close as possible to power pins, and connect with short, wide traces and copper.
- Connect pins to appropriate power plane with power vias (larger than signal vias).
- Bulk decoupling should be connected to a plane with 2 or more power vias.
- Minimize clock signal routing over plane splits.
- Do not route any signals underneath the clock generator on the component side of the board.
- An example signal via is a 14-mil finished hole with a 24-mil to 26-mil path. An example power via is an 18-mil finished hole with a 33-mil to 38-mil path. For large decoupling or power planes with large current transients, a larger power via is recommended.

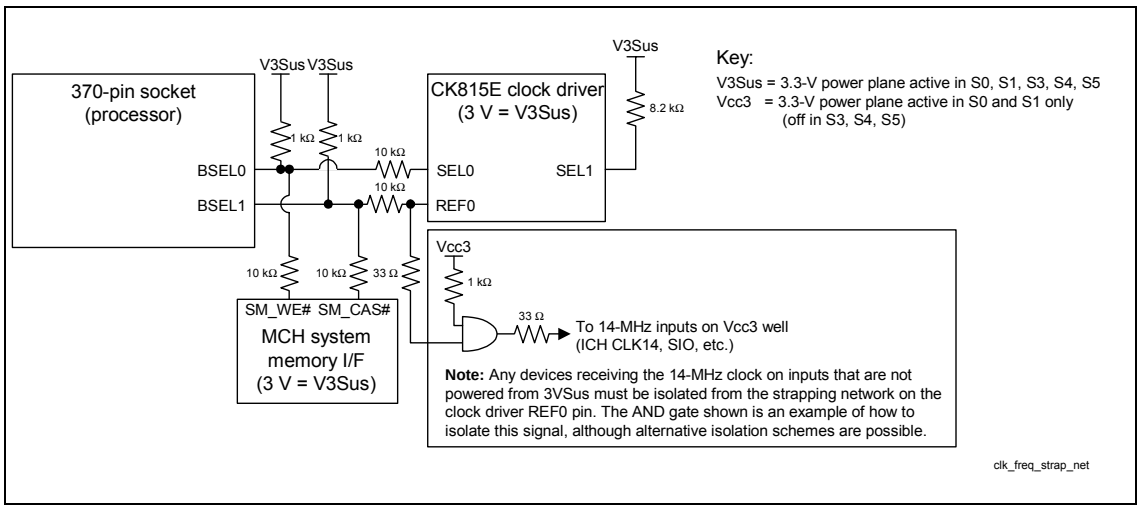
9.4. Clock Driver Frequency Strapping

A CK-815E-compliant clock driver device uses two of its pins to determine whether processor clock outputs should run at 133 MHz , 100 MHz or 66 MHz. The pin names are SEL0 and REF0. In addition, a third strapping pin is defined (SEL1), which must be pulled High for normal clock driver operation. Refer to the appropriate CK-815E clock driver specification for detailed strap timings and the logic encoding of straps.

SEL0 and REF0 are driven by either the processor, which depends on the processor populated in the 370-pin socket, or pull-up resistors on the motherboard. While SEL0 is a pure input to a CK-815E-compliant clock driver, REF0 is also the 14-MHz output that drives the ICH2 and other devices on the platform. In addition to sampling straps at reset, CK-815E-compliant clock drivers are configured by the BIOS via a two-wire interface to drive SDRAM clock outputs at either 100 MHz (default) or 133 MHz (if all system requirements are met).

If ACPI power management is supported on an 815EP chipset platform, the motherboard designer should power the clock chip and input straps from the 3.3-Vsus (e.g., active in S0, S1, S3, S4, S5) power supply. This enables the clock driver to seamlessly maintain its configuration register settings while switching between ACPI sleep and wake states. The following figure shows the block diagram of the recommended clock frequency strapping network, with implementation considerations.

Figure 78. Recommended Clock Frequency Strapping Network



For the platform to properly come out of reset, the clock driver straps powered from the standby supply must be isolated from any logic that powers off in ACPI sleep states.

9.5. Clock Skew Assumptions

The clock skew assumptions in the following table are used in the system clock simulations.

Table 32a. Simulated Clock Skew Assumptions

Skew Relationships	Target	Tolerance (±)	Notes
HCLK @ MCH to HCLK @ processor	0 ns	150 ps	• Assumes ganged clock outputs will allow max. of 50-ps skew
HCLK @ MCH to SCLK @ MCH	0 ns	600 ps	• 500-ps pin-to-pin skew • 100-ps board/package skew
SCLK @ MCH to SCLK @ SDRAM	0 ns	630 ps	• 250-ps pin-to-pin skew • 380-ps board + DIMM variation
HLCLK @ MCH to SCLK @ MCH	0 ns	900 ps	• 500-ps pin-to-pin skew • 400-ps board/package skew
HLCLK @ MCH to HCLK @ MCH	0 ns	700 ps	• 500-ps pin-to-pin skew • 200-ps board/package skew
HLCLK @ MCH to HLCLK @ ICH	0 ns	375 ps	• 175-ps pin-to-pin skew • 200-ps board/package skew
HLCLK @ ICH to PCICLK @ ICH	0 ns	900 ps	• 500-ps pin-to-pin skew • 400-ps board/package skew
PCICLK @ ICH to PCICLK @ other PCI devices	0 ns	2.0-ns window	• 500-ps pin-to-pin skew • 1.5-ns board/add-in skew
HLCLK @ MCH to AGPCLK @ connector			• Total electrical length of AGP connector + add-in card is 750 ps (according to AGP2.0 spec and AGP design guide 1.0). • Motherboard clock routing must account for this additional electrical length. Therefore, AGPCLK routed to the connector must be shorter than HLCLK to the GMCH, to account for this additional 750 ps.

This page is intentionally left blank.

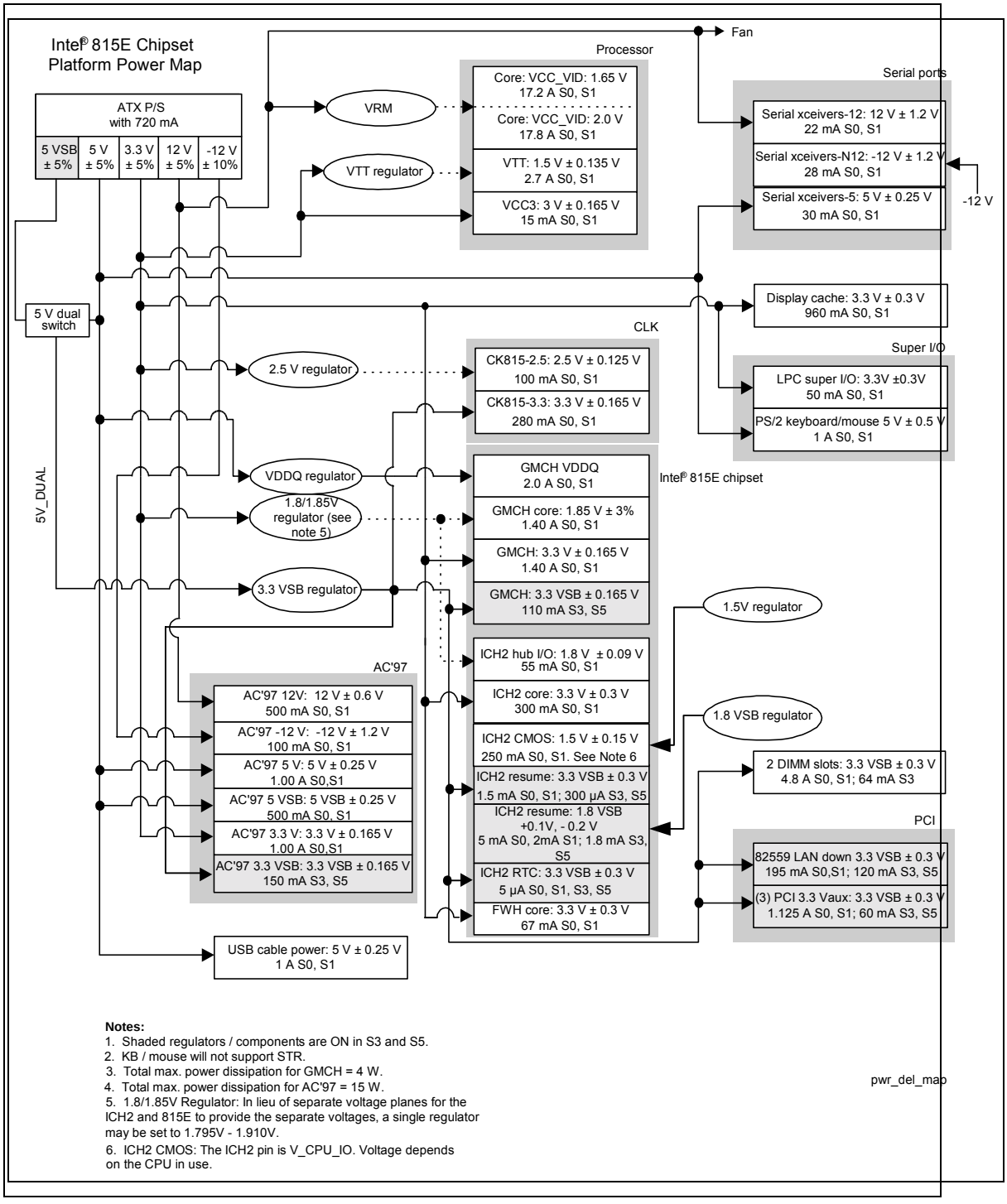
10. Power Delivery

The following figure shows the power delivery architecture for an example 815EP chipset platform. This power delivery architecture supports the "Instantly Available PC Design Guidelines" via the *suspend-to-RAM* (STR) state.

During STR, only the necessary devices are powered. These devices include: main memory, the ICH2 resume well, PCI wake devices (via 3.3 Vaux), AC'97, and optionally USB. (USB can be powered only if sufficient standby power is available.) To ensure that enough power is available during STR, a thorough power budget should be completed. The power requirements should include each device's power requirements, both in *suspend* and in *full-power*. The power requirements should be compared with the power budget supplied by the power supply. Due to the requirements of main memory and the PCI 3.3 Vaux (and possibly other devices in the system), it is necessary to create a *dual* power rail.

The solutions in this Design Guide are only examples. Many power distribution methods achieve the similar results. When deviating from these examples, it is critical to consider the effect of a change.

Figure 79. Power Delivery Map



10.1. Thermal Design Power

Thermal Design power (TDP) is defined as the estimated maximum possible expected power generated in a component by a realistic application. It is based on extrapolations in both hardware and software technology over the life of the product. It does not represent the expected power generated by a power virus.

The TDP of the 82815EP MCH component is 5.1W.

10.1.1. Power Sequencing

This section shows the timings between various signals during different power state transitions.

Figure 80. G3-S0 Transistion

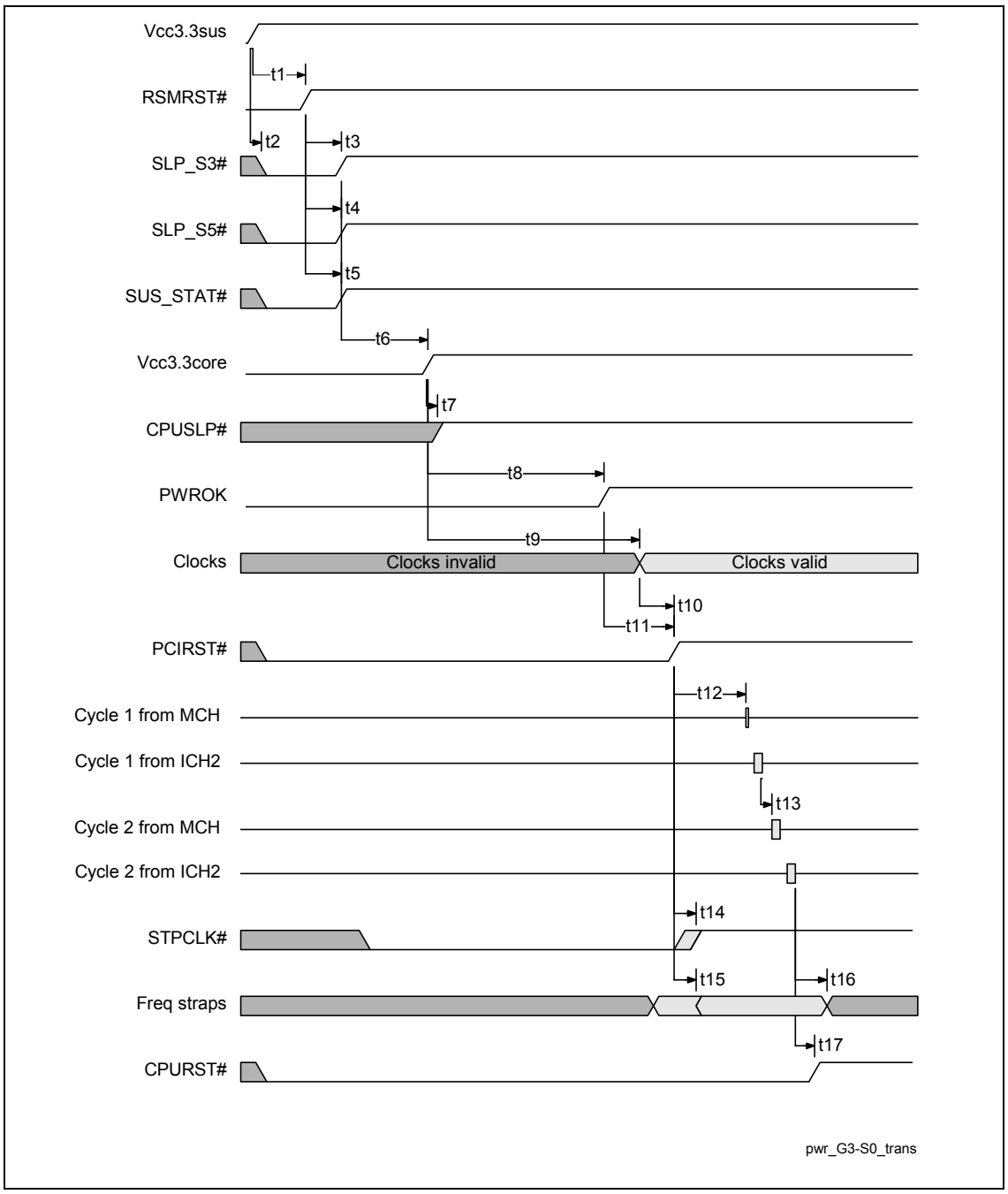


Figure 81. S0-S3-S0 Transition

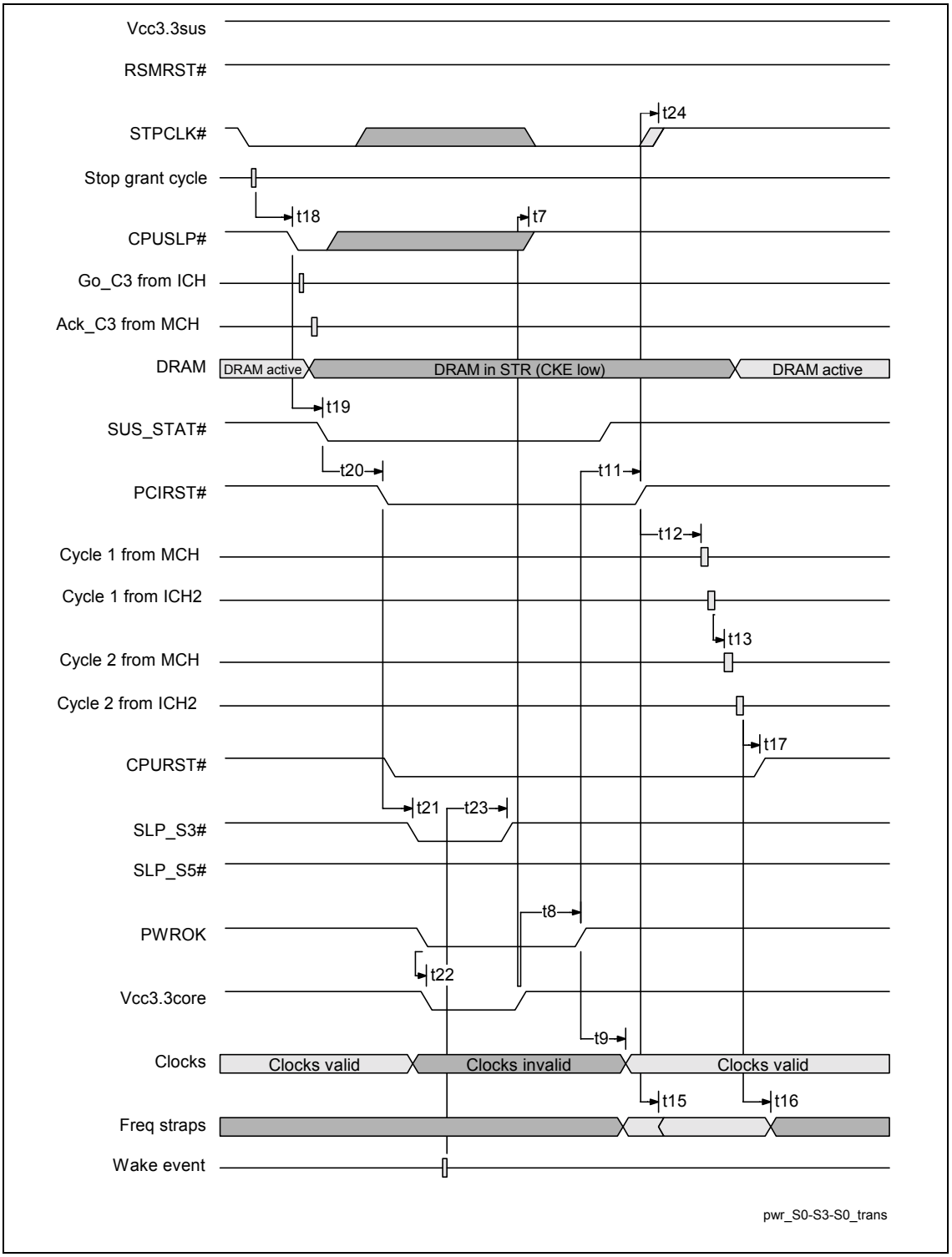


Figure 82. S0-S5-S0 Transition

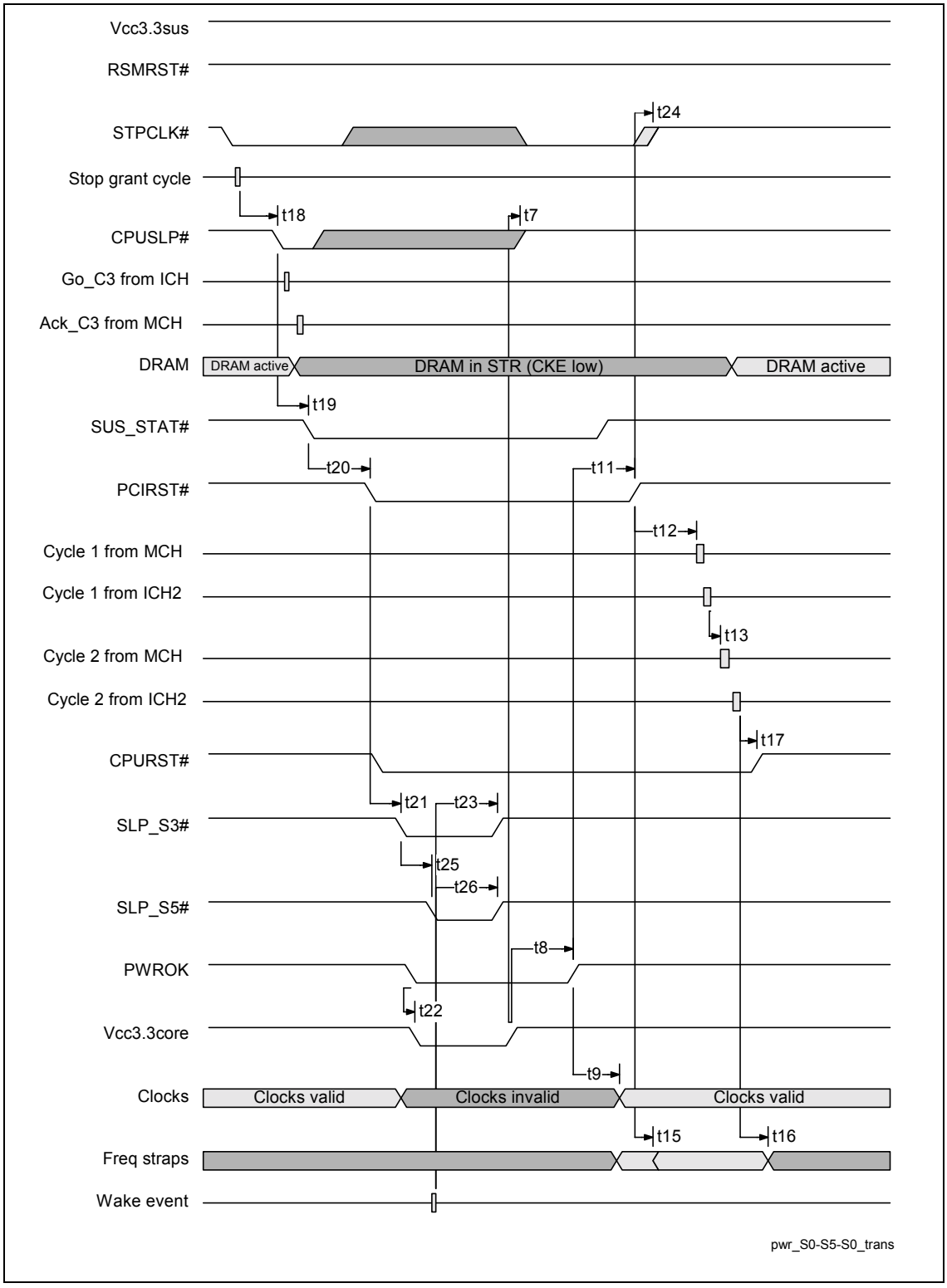


Table 36. Power Sequencing Timing Definitions

Symbol	Parameter	Min.	Max.	Units
t1	VccSUS good to RSMRST# inactive	1	25	ms
t2	VccSUS good to SLP_S3#, SLP_S5#, and PCIRST# active		50	ns
t3	RSMRST# inactive to SLP_S3# inactive	1	4	RTC clocks
t4	RSMRST# inactive to SLP_S5# inactive	1	4	RTC clocks
t5	RSMRST# inactive to SUS_STAT# inactive	1	4	RTC clocks
t6	SLP_S3#, SLP_S5#, SUS_STAT# inactive to Vcc3.3core good	*	*	
t7	Vcc3.3core good to CPUSLP# inactive		50	ns
t8	Vcc3.3core good to PWROK active	*	*	
t9	Vcc3.3core good to clocks valid	*	*	
t10	Clocks valid to PCIRST# inactive	500		µs
t11	PWROK active to PCIRST# inactive	.9	1.1	ms
t12	PCIRST# inactive to cycle 1 from MCH		1	ms
t13	Cycle 1 from Intel® ICH2 to cycle 2 from MCH		60	ns
t14	PCIRST# inactive to STPCLK de-assertion	1	4	PCI clocks
t15	PCIRST# to frequency straps valid	-4	4	PCI clocks
t16	Cycle 2 from ICH2 to frequency straps invalid		180	ns
t17	Cycle 2 from ICH2 to CPURST# inactive		110	ns
t18	Stop Grant Cycle to CPUSLP# active		8	PCI clocks
t19	CPUSLP# active to SUS_STAT# active		1	RTC clock
t20	SUS_STAT# active to PCIRST# active	2	3	RTC clocks
t21	PCIRST# active to SLP_S3# active	1	2	RTC clocks
t22	PWROK inactive to Vcc3.3core not good	20		ns
t23	Wake event to SLP_S3# inactive	2	3	RTC clocks
t24	PCIRST# inactive to STPCLK# inactive	1	4	PCI clocks
t25	SLP_S3# active to SLP_S5# active	1	2	RTC clocks
t26	SLP_S5# inactive to SLP_S3# inactive	2	3	RTC clocks

10.2. Pull-Up and Pull-Down Resistor Values

The pull-up and pull-down values are system dependent. The appropriate value for a system can be determined from an AC/DC analysis of the pull-up voltage used, the current drive capability of the output driver, the input leakage currents of all devices on the signal net, the pull-up voltage tolerance, the pull-up/pull-down resistor tolerance, the input high-voltage/low-voltage specifications, the input timing specifications (RC rise time), etc. Analysis should be performed to determine the minimum/maximum values usable on an individual signal. Engineering judgment should be used to determine the optimal value. This determination can include cost concerns, commonality considerations, manufacturing issues, specifications, and other considerations.

A simplistic DC calculation for a pull-up value is:

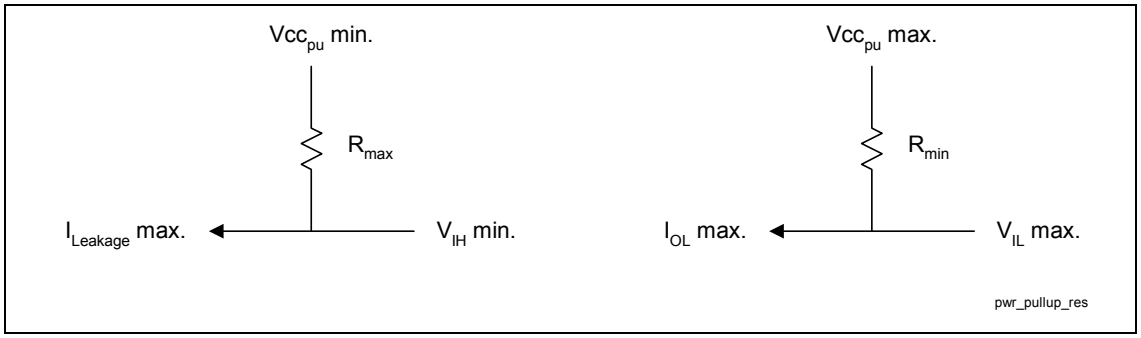
$$R_{MAX} = (Vcc_{PU}\,MIN - V_{IH}\,MIN) / I_{LEAKAGE}\,MAX$$

$$R_{MIN} = (Vcc_{PU}\,MAX - V_{IL}\,MAX) / I_{OL}\,MAX$$

Since $I_{LEAKAGE}$ MAX is normally very small, R_{MAX} may not be meaningful. R_{MAX} also is determined by the maximum allowable rise time. The following calculation allows for t, the maximum allowable rise time, and C, the total load capacitance in the circuit, including the input capacitance of the devices to be driven, the output capacitance of the driver, and the line capacitance. This calculation yields the largest pull-up resistor allowable to meet the rise time t.

$$R_{MAX} = -t / (C * \ln(1-(V_{IH}\,MIN / Vcc_{PU}\,MIN)))$$

Figure 83. Pull-up Resistor Example



10.3. ATX Power Supply PWRGOOD Requirements

The PWROK signal must be glitch free for proper power management operation. The ICH2 sets the PWROK_FLR bit (ICH2 GEN_PMCON_2, General PM Configuration 2 Register, PM-dev31: function 0, bit 0, at offset A2h). If this bit is set upon resume from S3 power-down, the system will reboot and control of the system will not be given to the program running when entering the S3 state. System designers should insure that PWROK signal designs are glitch free.

10.4. Power Management Signals

- A power button is required by the ACPI specification.
- PWRBTN# is connected to the front panel on/off power button. The ICH2 integrates 16-ms debouncing logic on this pin.
- AC power loss circuitry has been integrated into the ICH2 to detect power failure.
- It is recommended that the PS_POK signal from the power supply connector be routed through a Schmitt trigger to square-off and maintain its signal integrity. It should not be connected directly to logic on the board.
- PS_POK logic from the power supply connector can be powered from the core voltage supply.
- RSMRST# logic should be powered by a standby supply, while making sure that the input to the ICH2 is at the 3-V level. The RSMST# signal requires a minimum time delay of 1 ms from the rising edge of the standby power supply voltage. A Schmitt trigger circuit is recommended to drive the RSMRST# signal. To provide the required rise time, the 1-ms delay should be placed before the Schmitt trigger circuit. The reference design implements a 20-ms delay at the input of the Schmitt trigger to ensure that the Schmitt trigger inverters have sufficiently powered up before switching the input. Also ensure that voltage on RSMRST# does not exceed VCC(RTC).
- It is recommended that 3.3-V logic be used to drive RSMRST# to alleviate rise time problems when using a resistor divider from VCC5.
- The PWROK signal to the chipset is a 3-V signal.
- The core well power valid to PWROK asserted at the chipset is a minimum of 1 ms.
- PWROK to the chipset must be deasserted after RSMRST#.
- PWRGOOD signal to processor is driven with an open-collector buffer pulled up to 2.5 V, using a 330-Ω resistor.
- RI# can be connected to the serial port if this feature is used. To implement ring indicate as a wake event, the RS232 transceiver driving the RI# signal must be powered when the ICH2 suspend well is powered. This can be achieved with a serial port transceiver powered from the standby well that implements a shutdown feature.
- SLP_S3# from the ICH2 must be inverted and then connected to PSON of the power supply connector to control the state of the core well during sleep states.
- For an ATX power supply, when PSON is Low, the core wells are turned on. When PSON is high, the core wells from the power supply are turned off.

10.4.1. Power Button Implementation

The following items should be considered when implementing a power management model for a desktop system. The power states are as follows:

S1 – Stop Grant – (processor context not lost)

S3 - STR (Suspend to RAM)

S4 - STD (Suspend to Disk)

S5 - Soft-off

- Wake: Pressing the power button wakes the computer from S1-S5.
- Sleep: Pressing the power button signals software/firmware in the following manner:
- If SCI is enabled, the power button will generate an SCI to the OS.
  - The OS will implement the power button policy to allow orderly shutdowns.
  - Do not override this with additional hardware.
- If SCI is not enabled:
  - Enable the power button to generate an SMI and go directly to soft-off or a supported sleep state.
  - Poll the power button status bit during POST while SMIs are not loaded and go directly to soft-off if it gets set.
  - Always install an SMI handler for the power button that operates until ACPI is enabled.
- Emergency Override: Pressing the power button for 4 seconds goes directly to S5.
  - This is only to be used in EMERGENCIES when system is not responding.
  - This will cause the user data to be lost in most cases.
- Do not promote pressing the power button for 4 seconds as the normal mechanism to power the machine off. This violates ACPI.
- To be compliant with the latest PC9x specification, machines must appear to the user to be off when in the S1-S4 sleeping states. This includes:
  - All lights, except a power state light, must be off.
  - The system must be inaudible: silent or stopped fan, drives off.
- Note: Contact Microsoft for the latest information concerning PC9x and Microsoft Logo programs.

10.4.2. 1.8V/3.3V Power Sequencing

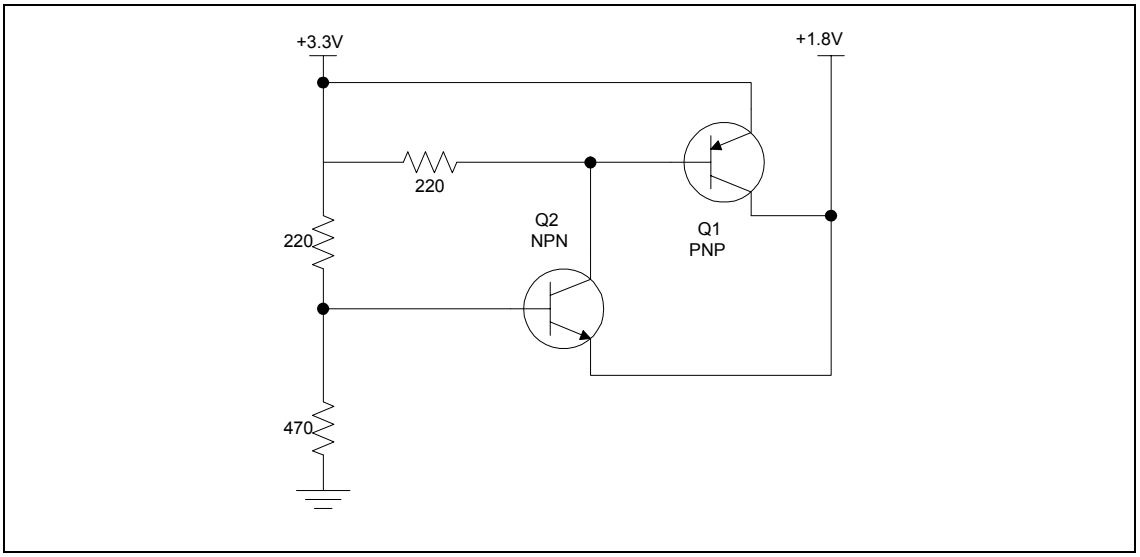
The ICH2 has two pairs of associated 1.8V and 3.3V supplies. These are {Vcc1_8, Vcc3_3} and {VccSus1_8, VccSus3_3}. These pairs are assumed to power up and power down together. **The difference between the two associated supplies must never be greater than 2.0V.** The 1.8V supply may come up before the 3.3V supply without violating this rule (though this is generally not practical in a desktop environment, since the 1.8V supply is typically derived from the 3.3V supply by means of a linear regulator).

One serious consequence of violation of this "2V Rule" is electrical overstress of oxide layers, resulting in component damage.

The majority of the ICH2 I/O buffers are driven by the 3.3V supplies, but are controlled by logic that is powered by the 1.8V supplies. Thus, another consequence of faulty power sequencing arises if the 3.3V supply comes up first. In this case the I/O buffers will be in an undefined state until the 1.8V logic is powered up. Some signals that are defined as "Input-only" actually have output buffers that are normally disabled, and the ICH2 may unexpectedly drive these signals if the 3.3V supply is active while the 1.8V supply is not.

The figure below shows an example power-on sequencing circuit that ensures the “2V Rule” is obeyed. This circuit uses a NPN (Q2) and PNP (Q1) transistor to ensure the 1.8V supply tracks the 3.3V supply. The NPN transistor controls the current through PNP from the 3.3V supply into the 1.8V power plane by varying the voltage at the base of the PNP transistor. By connecting the emitter of the NPN transistor to the 1.8V plane, current will not flow from the 3.3V supply into 1.8V plane when the 1.8V plane reaches 1.8V.

Figure 84. Example 1.8V/3.3V Power Sequencing Circuit



When analyzing systems that may be "marginally compliant" to the 2V Rule, please pay close attention to the behavior of the ICH2's RSMRST# and PWROK signals, since these signals control internal isolation logic between the various power planes:

- RSMRST# controls isolation between the RTC well and the Resume wells.
- PWROK controls isolation between the Resume wells and Main wells

If one of these signals goes high while one of its associated power planes is active and the other is not, a leakage path will exist between the active and inactive power wells. This could result in high, possibly damaging, internal currents.

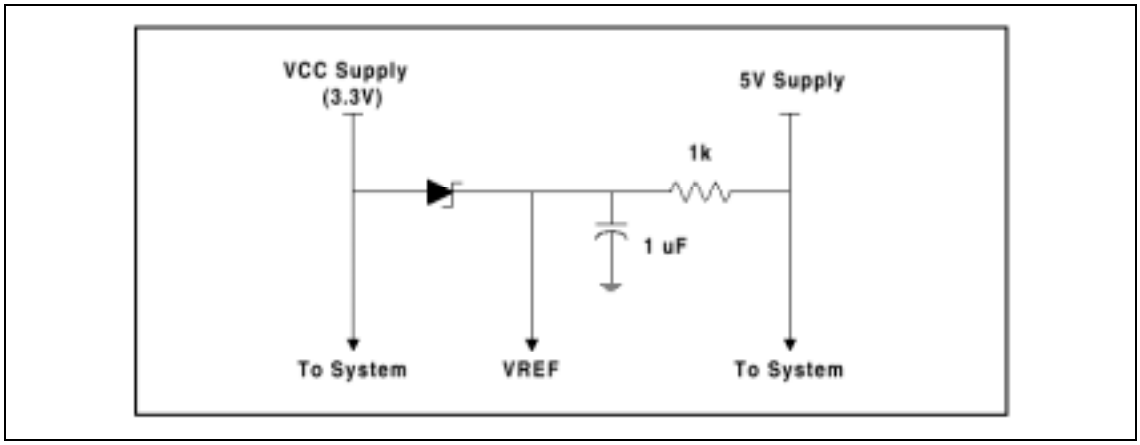
10.4.3. 3.3V/V5REF Sequencing Requirement

V5REF is the reference voltage for 5 V tolerance on inputs to the ICH2. V5REF must be powered up before Vcc3_3, or after Vcc3_3 within 0.7 V. Also, V5REF must power down after Vcc3_3, or before Vcc3_3 within 0 .7 V. The rule must be followed in order to ensure the safety of the ICH2. If the rule is violated, internal diodes will attempt to draw power sufficient to damage the diodes from the Vcc3_3 rail. Figure 77.a. shows a sample implementation of how to satisfy the V5REF/3.3V sequencing rule.

This rule also applies to the stand-by rails. However, in most platforms the VccSus3_3 rail is derived from the VccSus5 and therefore, the VccSus3_3 rail will always come up after the VccSus5 rail. As a result, V5REF_Sus will always be powered up before VccSus3_3. In platforms that do not derive the VccSus3_3 rail from the VccSus5 rail, this rule must be comprehended in the platform design.

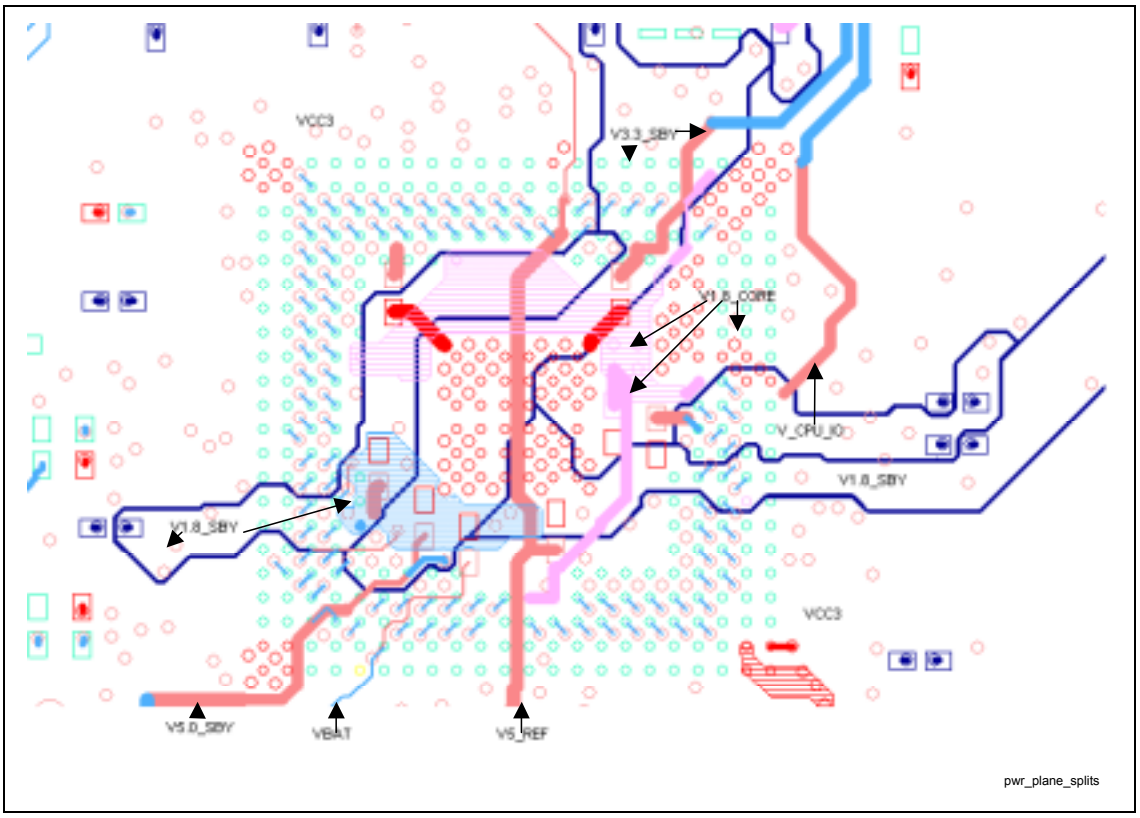
As an additional consideration, during suspend the only signals that are 5V tolerant are USB pins (both over-current and data lines). If USB is not implemented in the system then V5REF_SUS can be connected to the VccSus3_3 rail. Otherwise when USB is supported, V5REF_SUS must be connected to 5V_AUX, which remains powered during S5

Figure 85. Example 3.3V/V5REF Sequencing Circuitry



10.5. Power Plane Splits

Figure 86. Power Plane Split Example



10.6. Thermal Design Power

The thermal design power is the estimated maximum possible expected power generated in a component by a realistic application. It is based on extrapolations in both hardware and software technology over the life of the product. It does not represent the expected power generated by a power virus.

The thermal design power for the ICH2 is 1.5 W ±15%.

10.7. Glue Chip 3 (Intel® ICH2 Glue Chip)

To reduce the component count and BOM cost of the ICH2 platform, Intel has developed an ASIC component that integrates miscellaneous platform logic into a single chip. The Glue Chip 3 is designed to integrate some or all of the following functions into a single device. By integrating much of the required glue logic into a single device, overall board cost can be reduced.

Features

- PWROK signal generation
- Control circuitry for Suspend To RAM
- Power Supply power up circuitry
- RSMRST# generation
- Backfeed cutoff circuit for suspend to RAM
- 5V reference generation
- Flash FLUSH# / INIT# circuit
- HD single color LED driver
- IDE reset signal generation/PCIRST# buffers
- Voltage translation for Audio MIDI signal
- Audio-disable circuit
- Voltage translation for DDC to monitor
- Tri-state buffers for test

More information regarding this component is available from the following vendors.

Vendor	Contact	Contact Information
Fujitsu Microelectronics	Customer Response Center	3545 North 1st Street, M/S 104 San Jose, CA 95134-1804 *phone:* 1-800-866-8600 *fax:* 1-408-922-9179 *email:* fmicrc@fmi.fujitsu.com
Mitel Semiconductor	Greg Kizik Regional Business Manager	1735 Technology Drive Suite 240 San Jose, CA 95110 *phone:* 408-451-4723 *fax*: 408-451-4710 *e-mail:* greg_kizik@mitel.com http://www.mitelsemi.com

11. System Design Checklist

11.1. Design Review Checklist

11.1.1. Introduction

This checklist highlights design considerations that should be reviewed prior to manufacturing a motherboard that implements an 815EP chipset. This is not a complete list and does guarantee that a design will function properly. For items other than those in the following text, refer to the latest revision of the Design Guide for more-detailed instructions regarding motherboard design.

11.1.2. Design Checklist Summary

The following set of tables provides design considerations for the various portions of a design. Each table describes one of those portions and is titled accordingly. Contact your Intel Field Representative in the event of questions or issues regarding the interpretation of the information in these tables.

11.2. Processor Checklist

11.2.1. GTL Checklist

Checklist Items	Recommendations
A[35:3]# [1]	Connect A[31:3]# to MCH. Leave A[35:32]# as No Connect (not supported by chipset).
ADS#, BNR#, BPRI#, DBSY#, DEFER#, DRDY#, HA[31:3]#, HD[63:0]#, HIT#, HITM#, LOCK#, REQ[4:0]#, RS[2:0]#, TRDY#	Terminate to Vtt1.5 through 56 Ω resistor. Connect to MCH.
BREQ[0]# (BR0#)	56 Ω pull-down resistor to ground
RESET#, RESET2#	Terminate to Vtt1.5 through 86 Ω resistor, decoupled through 22 Ω resistor in series with 10 pF capacitor to ground. Connect to MCH. Also terminated to Vtt1.5 through 86 Ω resistor.

11.2.2. CMOS Checklist

Checklist Items	Recommendations
IERR#	150 Ω pull-up resistor to VCC_{CMOS} if tied to custom logic, or leave as No Connect (not used by chipset)

Checklist Items	Recommendations
PREQ#	200–330 Ω pull-up resistor to VCC_{CMOS} / Connect to ITP or else leave as No Connect.
THERMTRIP#	150 Ω pull-up resistor to VCC_{CMOS}, and connect to power off logic or leave as No Connect
A20M#, CPUSLP#, IGNNE#, INIT#, INTR, NMI, SLP#, SMI#, STPCLK#	Connect to Intel® ICH2. External pull-ups are not needed.
FERR#	Requires external weak pull-up to VCC_{CMOS}.
PWRGOOD	330 Ω pull-up to VCC2_5 / Connect to POWERGOOD logic.
V_{TT}	Route V_{TT} to all components on the host bus.

11.2.3. TAP Checklist for 370-Pin Socket Processors

Checklist Items	Recommendations
TCK, TMS	1 kΩ pull-up resistor to VCC_{CMOS} / 47-Ω series resistor to ITP
TDI	200–330 Ω pull-up resistor to VCC_{CMOS} / Connect to ITP.
TDO	150 Ω pull-up resistor to VCC_{CMOS} / Connect to ITP.
TRST#	680 Ω pull-down resistor to ground / Connect to ITP.
PRDY#	150 Ω pull-up resistor to Vtt / 240 Ω series resistor to ITP.

11.2.4. Miscellaneous Checklist for 370-Pin Socket Processors

Checklist Items	Recommendations
BCLK	Connect to clock generator. / 22–33-Ω series resistor (though OEM needs to simulate based on driver characteristics). To reduce pin-to-pin skew, tie host clock outputs together at the clock driver then route to the MCH and processor.
BSEL0	Case 1 (66/100/133-MHz support): 1-kΩ pull-up resistor to 2.5 V. Connect to CK815E SEL0 input. Connect to MCH LMD29 pin via 10-kΩ series resistor. Case 2 (100/133-MHz support): 1-kΩ pull-up resistor to 2.5 V. Connect to PWRGOOD logic such that a logic Low on BSEL0 negates PWRGOOD.
BSEL1	1-kΩ pull-up resistor to 2.5 V. Connect to CK815E REF pin via 10-kΩ series resistor. Connect to MCH LMD13 pin via 10-kΩ series resistor.
CLKREF	Connect to divider on VCC_2.5 or VCC_3.3 to create 1.25-V reference with a 4.7-µF decoupling capacitor. Resistor divider must be created from 1% tolerance resistors. Do not use VTT as source voltage for this reference!
CPUPRES#	Tie to ground. Leave as No Connect or connect to PWRGOOD logic to gate system from powering on if no processor is present. If used, 1-kΩ to 10-kΩ pull-up resistor to any voltage.
EDGCTRL/VRSEL	For Intel® Pentium® III processors, pulled high to VCC_{CORE} with a 51 Ω resistor.
PICCLK	Connect to clock generator. 22–33-Ω series resistor (though OEM needs to simulate based on driver characteristics)

Checklist Items	Recommendations
PLL1, PLL2	Low-pass filter on VCC_{CORE} provided on motherboard. Typically a 4.7-µH inductor in series with VCC_{CORE} is connected to PLL1, and then through a series 33-µF capacitor to PLL2.
RTTCTRL[5] (S35), SLEWCTRL (E27)	110-Ω ± 1% pull-down resistor to ground
THERMDN, THERMDP	No Connect if not used. Otherwise, connect to thermal sensor using vendor guidelines.
VCC_1.5	Connected to same voltage source as VTT. Must have some high- and low-frequency decoupling.
VCC_2.5	No connect for Pentium III processors
VCC_CMOS	Used as pull-up voltage source for CMOS signals between processor and chipset and for TAP signals between processor and ITP. Must have some decoupling (HF/LF) present.
VCC_{CORE}	10 ea. (min.) 4.7-µF in 1206 package all placed within the PGA370 socket cavity. 8 ea. (min.) 1 µF in 0612 package placed in the PGA370 socket cavity.
VCORE_DET (E21)	This pin should be left as a no-connect.
VID[3:0]	Connect to on-board VR or VRM. For on-board VR, 10-kΩ pull-up resistor to power solution-compatible voltage is required (usually pulled up to input voltage of the VR). Some of these solutions have internal pull-ups. Optional override (jumpers, ASIC, etc.) could be used. May also connect to system monitoring device.
VID[4]	Connect regulator controller pin to ground (not on processor).
V_{REF} [7:0]	Connect to V_{REF} voltage divider made up of 75-Ω and 150-Ω 1% resistors connected to Vtt. Decoupling Guidelines: 4 ea. (min.) 0.1 µF in 0603 package placed within 500 mils of V_{REF} pins
Vtt	Connect AH20, AK16, AL13, AL21, AN11, AN15, and G35 to 1.5-V regulator. Provide high- and low-frequency decoupling. Decoupling Guidelines: 19 ea (min.) 0.1 µF in 0603 package placed within 200 mils of AGTL+ termination resistor packs (r-paks). Use one capacitor for every two (r-paks). 4 ea (min.) 0.47 µF in 0612 package
NO CONNECTS	The following pins must be left as no-connects: AK30, AM2, F10, L33, N33, N35, N37, Q33, Q35, Q37, R2, W35, X2, Y1
AA33, AA35, AN21, E23, S33, S37, U35, U37	A platform using an Intel® 815EP chipset is not compatible with an Intel® Celeron® processor (PPGA). These pins must be connected directly to Vtt.
G37	No Connect

11.3. MCH Checklist

11.3.1. AGP Interface 1X Mode Checklist

Checklist Items	Recommendations
RBF#, WBF#, PIPE#, GREQ#, GGNT#, GPAR, GFRAME#, GIRDY#, GTRDY#, GSTOP#, GDEVSEL#, GPERR#, GSERR# , ADSTB0, ADSTB1, SBSTB	Pull up to VDDQ through 8.2 kΩ
ADSTB0#, ADSTB1#, SBSTB#	Pull down to ground through 8.2 kΩ
PME#	Connect to PCI connector 0 device Ah. / Connect to PCI connector 1 device Bh. / Connect to Intel® 82559 LAN (if implemented).
TYPEDET#	Connect to AGP voltage regulator circuitry / AGP reference circuitry.
PIRQ#A, PIRQ#B	Pull up to 5 V through 2.7 kΩ. / Follow ref. schematics (other device connections).

11.3.2. Hub Interface Checklist

Checklist Items	Recommendations
HUBREF	Connect to HUBREF generation circuitry.
HL_COMP	Pull up to VCC1.8 through 40 Ω (both MCH and Intel® ICH2 side).

11.4. Intel® ICH2 Checklist

11.4.1. PCI Interface

Checklist Items	Recommendations
All	All inputs to the Intel® ICH2 must not be left floating. Many GPIO signals are fixed inputs that must be pulled up to different sources. See GPIO section for recommendations.
PERR#, SERR# PLOCK#, STOP# DEVSEL#, TRDY# IRDY#, FRAME# REQ#[0:4], GPIO[0:1], THRM#	These signals require a pull-up resistor. Recommend an 8.2 kΩ pull-up resistor to VCC3.3 or a 2.7 kΩ pull-up resistor to VCC5. See PCI 2.2 Component Specification for pull-up recommendations for VCC3.3 and VCC5.
PCIRST#	The PCIRST# signal should be buffered to form the IDERST# signal. 33 Ω series resistor to IDE connectors.
PCIGNT#	No external pull-up resistors are required on PCI GNT signals. However, if external pull-up resistors are implemented they must be pulled up to VCC3.3.
PME#	No extra pull-up resistors These signals have integrated pull-up resistors of 9 kΩ ± 3 kΩ.
SERIRQ	External weak (8.2 kΩ) pull-up resistor to VCC3.3 is recommended.
GNT[A]# /GPIO[16], GNT[B]/ GNT[5]#/ GPIO[17]	No extra pull-up needed. These signals have integrated pull-ups of 24 kΩ. GNT[A] has an added strap function of “top block swap”. The signal is sampled on the rising edge of PWROK. Default value is high or disabled due to pull-up. A Jumper to a pull-down resistor can be added to manually enable the function.

11.4.2. Hub Interface

Checklist Items	Recommendations
HL[11]	No pull-up resistor required. Use a no-stuff or a test point to put the ICH2 into NAND chain mode testing
HL_COMP	Tie the COMP pin to a 40Ω 1% or 2% (or 39 Ω - 1%) pull-up resistor (to VCC1.8) via a 10-mil wide, very short (~0.5 inch) trace. ZCOMP No longer supported.

11.4.3. LAN Interface

Checklist Items	Recommendations	Comments
1	Trace Spacing: 5 mils wide, 10 mil spacing	
2	LAN Max Trace Length Intel® ICH2 to CNR: L = 3” to 9” (0.5” to 3” on card)	To meet timing requirements
3	Stubs due to R-pak CNR/LOM stuffing option should not be present.	To minimize inductance

Checklist Items	Recommendations	Comments
4	Maximum Trace Lengths: ICH2 to Intel® 82562EH: L = 4 inches; Intel® 82562EM: L = 4.5 inches to 8.5 inches.	To meet timing requirements
5	Max mismatch between the length of a clock trace and the length of any data trace is 0.5 inches (clock must be longest trace)	To meet timing and signal quality requirements
6	Maintain constant symmetry and spacing between the traces within a differential pair out of the LAN phy.	To meet timing and signal quality requirements
7	Keep the total length of each differential pair under 4 inches.	Issues found with traces longer than 4 inches: IEEE phy conformance failures, excessive EMI and or degraded receive BER.
8	Do not route the transmit differential traces closer than 100 mils to the receive differential traces.	To minimize crosstalk
9	Distance between differential traces and any other signal line is 100 mils. (300 mils recommended)	To minimize crosstalk
10	Route 5 mils on 7 mils for differential pairs (out of LAN phy)	To meet timing and signal quality requirements
11	Differential trace impedance should be controlled to be ~100 Ω.	To meet timing and signal quality requirements
12	For high-speed signals, the number of corners and vias should be kept to a minimum. If a 90° bend is required, it is recommended to use two 45° bends.	To meet timing and signal quality requirements
13	Traces should be routed away from board edges by a distance greater than the trace height above the ground plane.	This allows the field around the trace to couple more easily to the ground plane rather than to adjacent wires or boards.
14	Do not route traces and vias under crystals or oscillators.	This will prevent coupling to or from the clock
15	Trace width to height ratio above the ground plane should be between 1:1 and 3:1.	To control trace EMI radiation
16	Traces between decoupling and I/O filter capacitors should be as short and wide as practical.	Long and thin lines are more inductive and would reduce the intended effect of decoupling capacitors
17	Vias to decoupling capacitors should be sufficiently large in diameter.	To decrease series inductance
18	Avoid routing high-speed LAN* or Phoneline traces near other high-frequency signals associated with a video controller, cache controller, CPU, or other similar devices.	To minimize crosstalk
19	Isolate I/O signals from high-speed signals.	To minimize crosstalk
20	Place the 82562ET/EM part more than 1.5 inches away from any board edge.	This minimizes the potential for EMI radiation problems
21	Place at least one bulk capacitor (4.7 µF or greater) on each side of the 82562ET/EM.	Research and development has shown that this is a robust design requirement

Checklist Items	Recommendations	Comments
22	Place decoupling caps (0.1 µF) as close to the 82562ET/EM as possible.	
23 LAN_CLK	Connect to LAN_CLK on Platform LAN Connect Device.	
24 LAN_RXD[2:0]	Connect to LAN_RXD on Platform LAN Connect Device. ICH2 contains integrated 9 kΩ pull-up resistors on interface.	
25 LAN_TXD[2:0] LAN_RSTSYNC	Connect to LAN_TXD on Platform LAN Connect Device.	

NOTES:
1. LAN connect interface can be left NC if not used. Input buffers internally terminated.
2. In the event of EMI problems during emissions testing (FCC Classifications) you may need to place a decoupling cap (~470 pF) on each of the 4 LED pins. Reduces emissions attributed to LAN subsystem.

11.4.4. EEPROM Interface

Checklist Items	Recommendations
EE_DOUT	Prototype Boards should include a placeholder for a pulldown resistor on this signal line, but do not populate the resistor. Connect to EE_DIN of EEPROM or CNR Connector. Connected to EEPROM data input signal (input from EEPROM perspective and output from Intel® ICH2 perspective).
EE_DIN	No extra circuitry required. Connect to EE_DOUT of EEPROM or CNR Connector. ICH2 contains an integrated pull-up resistor for this signal. Connected to EEPROM data output signal (output from EEPROM perspective and input from ICH2 perspective).

11.4.5. FWH/LPC Interface

Checklist Items	Recommendations
FWH[3:0]/ LAD[3:0] LDRQ[1:0]	No extra pull-ups required. Intel® ICH2 Integrates 24 kΩ pull-up resistors on these signal lines.

11.4.6. Interrupt Interface

Checklist Items	Recommendations
PIRQ#[D:A]	These signals require a pull-up resistor. The recommendation is a 2.7 kΩ pull-up resistor to VCC5 or 8.2 kΩ to VCC3.3. In Non-APIC Mode the PIRQx# signals can be routed to interrupts 3, 4, 5, 6, 7, 9, 10, 11, 12, 14 or 15 as described in the Interrupt Steering section. Each PIRQx# line has a separate Route Control Register. In APIC mode, these signals are connected to the internal I/O APIC in the following fashion: PIRQ[A]# is connected to IRQ16, PIRQ[B]# to IRQ17, PIRQ[C]# to IRQ18, and PIRQ[D]# to IRQ19. This frees the ISA interrupts.
PIRQ#[G:F]/ GPIO[4:3]	These signals require a pull-up resistor. Recommend a 2.7 kΩ pull-up resistor to VCC5 or 8.2 kΩ to VCC3.3. In Non-APIC Mode the PIRQx# signals can be routed to interrupts 3, 4, 5, 6, 7, 9, 10, 11, 12, 14 or 15 as described in the Interrupt Steering section. Each PIRQx# line has a separate Route Control Register. In APIC mode, these signals are connected to the internal I/O APIC in the following fashion: PIRQ[E]# is connected to IRQ20, PIRQ[F]# to IRQ21, PIRQ[G]# to IRQ22, and PIRQ[H]# to IRQ23. This frees the ISA interrupts.
PIRQ#[H] PIRQ#[E]	These signals require a pull-up resistor. Recommend a 2.7 kΩ pull-up resistor to VCC5 or 8.2 kΩ to VCC3.3. Since PIRQ[H]# and PIRQ[E]# are used internally for LAN and USB controllers, they cannot be used as GPIO(s) pin.
APIC	**If the APIC is used:** 150 Ω pull-up resistors on APICD[0:1] Connect APICCLK to CK133 with a 20–33 Ω series termination resistor. **If the APIC is not used on UP systems:** The APICCLK can either be tied to GND or connected to CK133, but not left floating. Pull APICD[0:1] to GND through 10 kΩ pull-down resistors. Use pull-downs for each APIC signal. Do not share resistor to pull signals up.

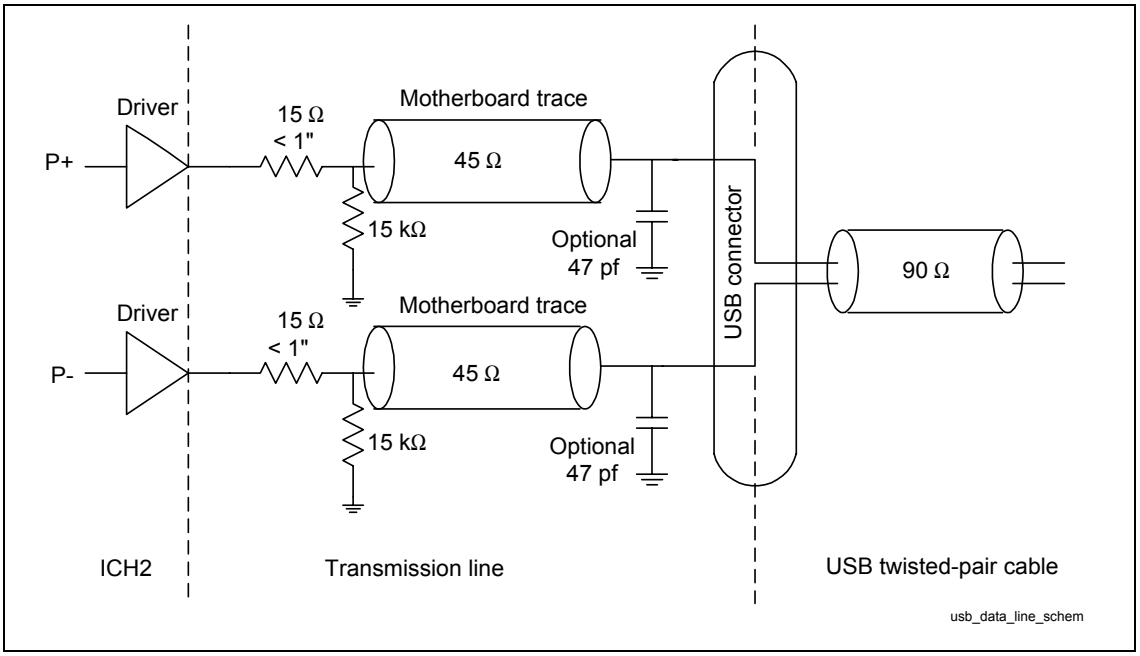
11.4.7. GPIO Checklist

Checklist Items	Recommendations
All	Ensure ALL unconnected signals are OUTPUTS ONLY!
GPIO[7:0]	These pins are in the Main Power Well. Pull-ups must use the VCC3.3 plane. Unused **core** well inputs must either be pulled up to VCC3.3 or pulled down. Inputs must not be allowed to float. These signals are 5V tolerant. GPIO[1:0] can be used as REQ[A:B]#. GPIO[1] can also used as PCI REQ[5]#.
[13:11], GPIO[8]	These pins are in the Resume Power Well. Pull-ups must use the VCCSUS3.3 plane. These are the only GPI signals in the resume well with associated status bits in the GPE1_STS register. Unused **resume** well inputs must be pulled up to VCCSUS3.3. These signals are not 5V tolerant. These are the only GPIs that can be used as ACPI compliant wake events.
GPIO[23:16]	Fixed as output only. Can be left NC. In Main Power Well. GPIO22 is open drain.
GPIO[24,25,27,28]	These I/O pins can be NC. These pins are in the resume power well.

11.4.8. USB

Checklist Items	Recommendations
USBP[3:0]P USBP[3:0]N	See Figure 1 for circuitry needed on each differential Pair.
VCC USB (Cable power)	It should be powered from the 5V core instead of the 5V standby, unless adequate standby power is available.
Voltage drop considerations	The resistive component of the fuses, ferrite beads and traces must be considered when choosing components, and power and GND trace widths. Minimize the resistance between the Vcc5 power supply and the USB ports to prevent voltage drop. Sufficient bypass capacitance should be located near the USB receptacles to minimize the voltage drop that occurs during the hot plugging a new device. For more information, see the USB specification.
Fuse	A fuse larger than 1A can be chosen to minimize the voltage drop.

Figure 87. USB Data Line Schematic



11.4.9. Power Management

Checklist Items	Recommendations
THRM#	Connect to temperature Sensor. Pull-up if not used.
SLP_S3# SLP_S5#	No pull-up/down resistors needed. Signals driven by Intel® ICH2.
PWROK	This signal should be connected to power monitoring logic, and should go high no sooner than 10 ms after both Vcc3_3 and Vcc1_8 have reached their nominal voltages.
PWRBTN#	No extra pull-up resistors. These signals have integrated pull-ups of 9 kΩ ±3 kΩ.
RI#	RI# does not have an internal pull-up. Recommend an 8.2 kΩ pull-up resistor to Resume well. If this signal is enabled as a wake event, it is important to keep this signal powered during the power loss event. If this signal goes low (active), when power returns the RI_STS bit will be set and the system will interpret that as a wake event.
RSMRST#	Connect to power monitoring logic, and should go high no sooner than 10 ms after both VccSUS3_3 and VccSus1_8 have reached their nominal voltages. Requires weak pull-down. Also requires well isolation control as directed in Section 8.16.8

11.4.10. Processor Signals

Checklist Items	Recommendations
A20M#, CPUSLP#, IGNNE#, INIT#, INTR, NMI, SMI#, STPCLK#	Internal circuitry has been added to the ICH2, external pull-up resistors are not needed.
FERR#	Requires Weak external pull-up resistor to VCC_{CMOS}.
RCIN# A20GATE	Pull-up signals to VCC3.3 through a 10 kΩ resistor.
CPUPWRGD	Connect to the processor's CPUPWRGD input. Requires weak external pull-up resistor.

11.4.11. System Management

Checklist Items	Recommendations
SMBDATA SMBCLK	Requires external pull-up resistors. See SMBus Architecture and Design Consideration section to determine the appropriate power well to use to tie the pullup resistors. (Core well, suspend well, or a combination.) Value of pull-up resistors determined by line load. Typical value used is 8.2 kΩ.
SMBALERT#/ GPIO[11]	See GPIO section if SMBALERT# not implemented
SMLINK[1:0]	Requires external pull-up resistors. See SMBus Architecture and Design Consideration section to determine the appropriate power well to use to tie the pullup resistors. (Core well, suspend well, or a combination.) Value of pull-up resistors determined by line load. Typical value used is 8.2 kΩ.
INTRUDER#	Pull signal to VCCRTC (VBAT), if not needed.

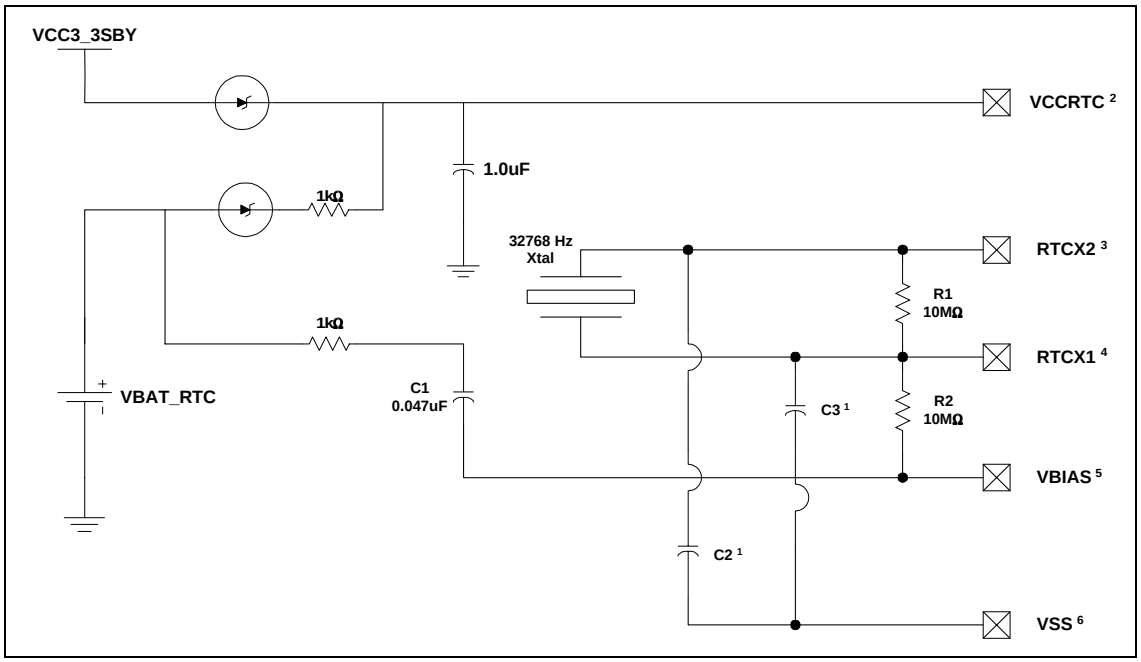
11.4.12. ISA Bridge Checklist

Checklist Items	Recommendations
ICH2 GPO[21] / MISA NOGO input	Connect Intel® ICH2 GPO[21] to MISA NOGO input. If GPO[21] is not available on the ICH2, any other GPO that defaults High in the system can be used. GPO[21] is the only ICH2 GPO that defaults high.
ICH2 AD22 / MISA IDSEL input	Connect ICH2 AD22 to the MISA IDSEL input.

11.4.13. RTC

Checklist Items	Recommendations
VBIAS	The VBIAS pin of the Intel® ICH2 is connected to a .047 µF cap. See Figure 88, below.
RTCX1 RTCX2	Connect a 32.768 kHz Crystal Oscillator across these pins with a 10 MΩ resistor and use 18 pF decoupling caps (assuming crystal with CLOAD=12.5 pF) at each signal. The ICH2 implements a new internal oscillator circuit as compared with the PIIX4 to reduce power consumption. The external circuitry shown in Figure 88, below will be required to maintain the accuracy of the RTC. The circuitry is required since the new RTC oscillator is sensitive to step voltage changes in VCCRTC and VBIAS. A negative step on power supply of more than 100 mV will temporarily shut off the oscillator for hundreds of milliseconds. RTCX1 may optionally be driven by an external oscillator instead of a crystal. These signals are 1.8V only, and must not be driven by a 3.3V source.
RTCRST#	Ensure 10 ms-20 ms RC delay (8.2 K & 2.2 µF) See Figure, *RTCRST External Circuit for the ICH2 RTC*.
SUSCLK	To assist in RTC circuit debug, route SUSCLK to a test point if SUSCLK is unused.

Figure 88. Intel® ICH2 Oscillator Circuitry



NOTES: Capacitors C2 and C3 are crystal dependent.

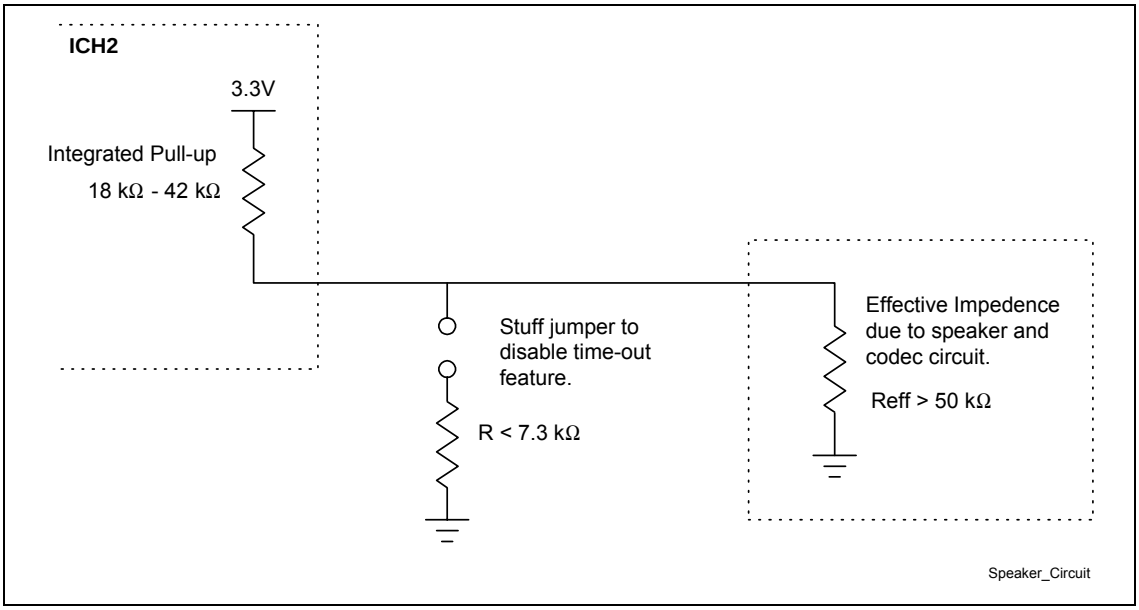
11.4.14. AC'97

Checklist Items	Recommendations
AC_BITCLK	No extra pull-down resistors required. When nothing is connected to the link, BIOS must set a shut off bit for the internal keeper resistors to be enabled. At that point, you do not need pull-ups/pull-downs on any of the link signals.
AC_SYNC	No extra pull-down resistors required. Some implementations add termination for signal integrity. Platform specific.
AC_SDOUT	Requires a jumper to 8.2 kΩ pull-up resistor. Should not be stuffed for default operation. This pin has a weak internal pull-down. To properly detect a safe_mode condition a strong pull-up will be required to over-ride this internal pull-down.
AC_SDIN[1], AC_SDIN[0]	Requires pads for weak 10 kΩ pull-downs. Stuff resistor for unused AC_SDIN signal or AC_SDIN signal going to the CNR connector. AC_SDIN[1:0] are inputs to an internal OR gate. If a pin is left floating, the output of the OR gate will be erroneous. If there is no codec on the system board, then both AC_SDIN[1:0] should be pull-down externally with resisters to ground.
CDC_DN_ENAB#	If the primary codec is down on the motherboard, this signal must be low to indicate the motherboard codec is active and controlling the AC '97 interface.
	Z_0 AC '97 = 60 Ω ± 15%
	5-mil trace width, 5-mil spacing between traces
	Max Trace Length Intel® ICH2/Codec/CNR = 14 inches

11.4.15. Miscellaneous Signals

Checklist Items	Recommendations
SPKR	No extra pull-up resistors. Has integrated pull-up of between 18 kΩ and 42 kΩ. The integrated pull-up is only enabled at boot/reset for strapping functions; at all other times, the pull-up is disabled. A low effective impedance may cause the TCO Timer Reboot function to be erroneously disabled. Effective Impedance due Speaker and Codec circuitry must be greater than 50 kΩ or a means to isolate the resistive load from the signal while PWROK is low be found. see following figure.
TP[0]	Requires external pull-up resistor to VCCSUS3.3
FS[0]	Rout to a testpoint. Intel® ICH2 contains an integrated pull-up for this signal. Testpoint used for manufacturing appears in XOR tree.

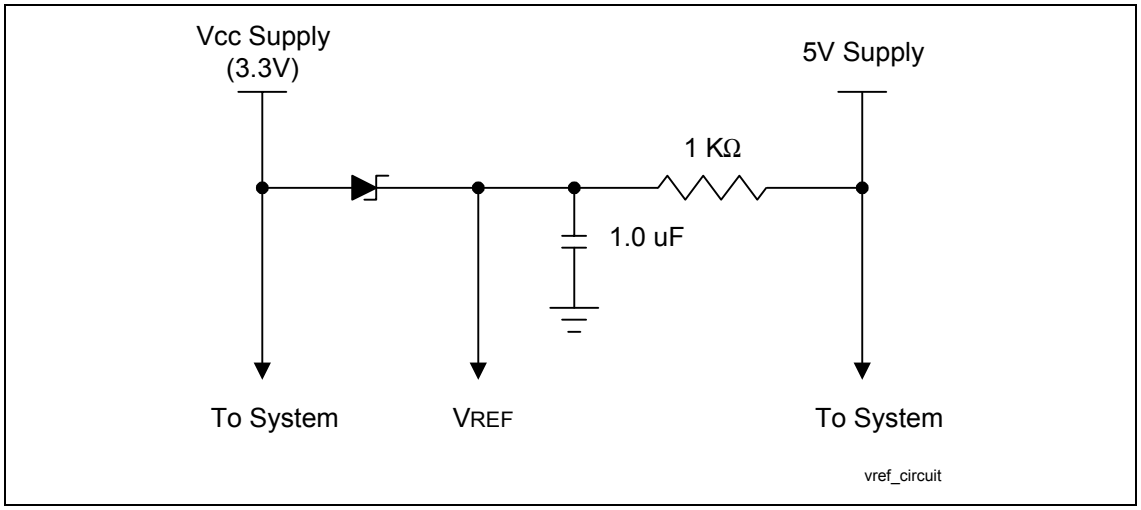
Figure 89. SPKR Circuitry



11.4.16. Power

Checklist Items	Recommendations
V_CPU_IO[1:0]	The power pins should be connected to the proper power plane for the processor 's CMOS Compatibility Signals. Use one 0.1 µF decoupling cap.
VccRTC	No clear CMOS jumper on VccRTC. Use a jumper on RTCRST# or a GPI, or use a safemode strapping for Clear CMOS
Vcc3.3	Requires six 0.1 µF decoupling caps
VccSus3.3	Requires one 0.1 µF decoupling cap.
Vcc1.8	Requires two 0.1 µF decoupling caps.
VccSus1.8	Requires one 0.1 µF decoupling cap.
V5_REF SUS	Requires one 0.1 µF decoupling cap. V5REF_SUS affects 5 V-tolerance for all USB pins and can be connected to VccSUS3_3 if Intel® ICH2 USB is not supported in the platform. If USB is supported, V5REF_SUS must be connected to 5V_AUX, which remains powered during S5.
V5_REF	V5REF is the reference voltage for 5V tolerant inputs in the ICH2. Tie to pins VREF[2:1]. V5REF must power up before or simultaneous to Vcc3_3. It must power down after or simultaneous to Vcc3_3. Refer to the figure below for an example circuit schematic that may be used to ensure the proper V5REF sequencing.

Figure 90. V5REF Circuitry



11.4.17. IDE Checklist

Checklist Items	Recommendations
PDD[15:0], SDD[15:0]	No extra series termination resistors or other pull-ups/pull-downs are required. These signals have integrated series resistors. **NOTE:** Simulation data indicates that the integrated series termination resistors can range from 31 Ω to 43 Ω. PDD7/SDD7 does not require a 10 kΩ pull-down resistor. Refer to ATA ATAPI-4 specification.
PDIOW#, PDIOR#, PDDACK#, PDA[2:0], PDCS1#, PDCS3#, SDIOW#, SDIOR#, SDDACK#, SDA[2:0], SDCS1#, SDCS3#	No extra series termination resistors. Pads for series resistors can be implemented should the system designer have signal integrity concerns. These signals have integrated series resistors. **NOTE:** Simulation data indicates that the integrated series termination resistors can range from 31 Ω to 43 Ω.
PDREQ SDREQ	No extra series termination resistors. No pull-down resistors needed. These signals have integrated series resistors in the Intel® ICH2. These signals have integrated pull-down resistors in the ICH2.
PIORDY SIORDY	No extra series termination resistors. These signals have integrated series resistors in the ICH2. Pull-up to VCC3.3 via a 4.7 kΩ resistor.
IRQ14, IRQ15	Recommend 8.2 kΩ—10 kΩ pull-up resistors to VCC3.3. No extra series termination resistors.
IDERST#	The PCIRST# signal should be buffered to form the IDERST# signal. A 33 Ω series termination resistor is recommended on this signal.
Cable Detect:	**Host Side/Device Side Detection:** Connect IDE pin PDIAG/CBLID to an ICH2 GPIO pin. Connect a 10 kΩ resistor to GND on the signal line. The 10 kΩ resistor to GND prevents GPI from floating if no devices are present on either IDE interface. Allows use of 3.3V and 5V tolerant GPIOs. **Device Side Detection:** Connect a 0.047 µF capacitor from IDE pin PDIAG/CBLID to GND. No ICH2 connection. **NOTE:** All ATA66/ATA100 drives will have the capability to detect cables

NOTES: The maximum trace length from the ICH2 to the ATA connector is 8 inches.

Figure 91. Host/Device Side Detection Circuitry

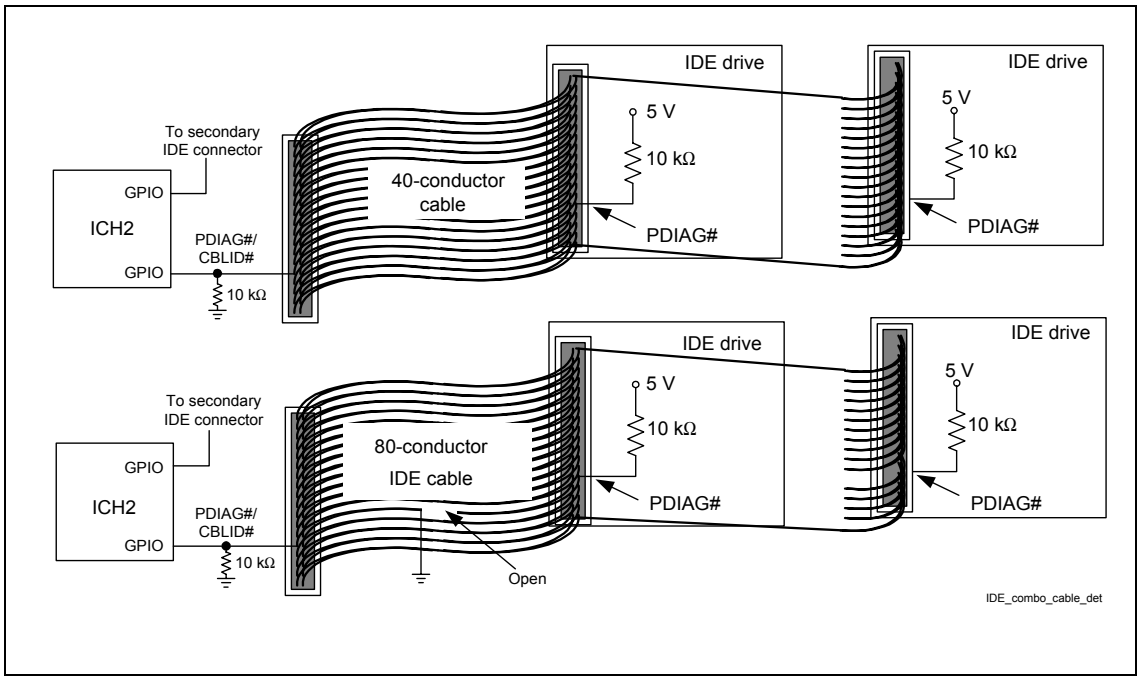
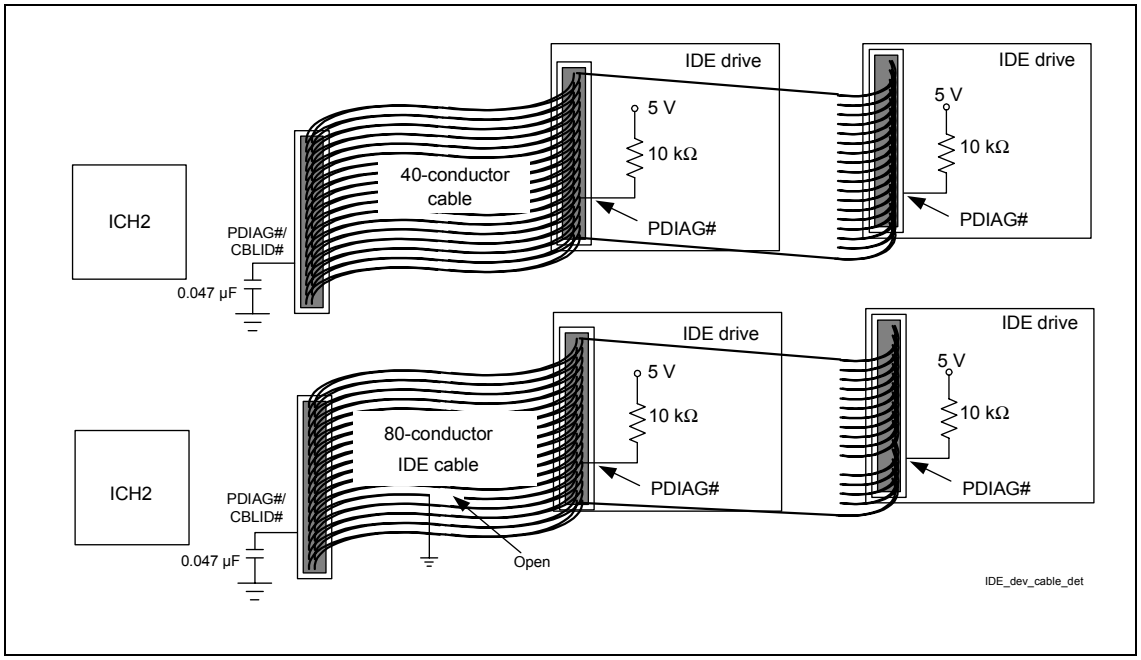


Figure 92. Device Side Only Cable Detection



11.5. LPC Checklist

Checklist Items	Recommendations
RCIN#	Pull up through 8.2-kΩ resistor to Vcc3_3.

Checklist Items	Recommendations
LPC_PME#	Pull up through 8.2-kΩ resistor to Vcc3_3. Do not connect LPC PME# to PCI PME#. If the design requires the Super I/O to support wake from any suspend state, connect Super I/O LPC_PME# to a resume well GPI on the Intel® ICH2.
LPC_SMI#	Pull up through 8.2-kΩ resistor to Vcc3_3. This signal can be connected to any ICH2 GPI. The GPI_ROUTE register provides the ability to generate an SMI# from a GPI assertion.
TACH1, TACH2	Pull up through 4.7-kΩ resistor to Vcc3_3. Jumper for decoupling option (decouple with 0.1-µF cap)
J1BUTTON1, JPBUTTON2, J2BUTTON1, J2BUTTON2	Pull up through 1-kΩ resistor to Vcc5. Decouple through 47-pF cap to GND.
LDRQ#1	Pull up through 4.7-kΩ resistor to Vcc3SBY.
A20GATE	Pull up through 8.2-kΩ resistor to Vcc3_3.
MCLK, MDAT	Pull up through 4.7-kΩ resistor to PS2V5.
L_MCLK, L_MDAT	Decoupled using 470-pFcap to ground.
RI#1_C, CTS0_C, RXD#1_C, RXD0_C, RI0_C, DCD#1_C, DSR#1_C, DSR0_C, DTR#1_C, DTR0_C, DCD0_C, RTS#1_C, RTS0_C, CTS#1_C, TXD#1_C, TXD0_C	Decoupled using 100-pF cap to GND.
L_SMBD	Pass through 150-Ω resistor to Intel® 82559.
SERIRQ	Pull up through 8.2 kΩ to Vcc3_3.
SLCT#, PE, BUSY, ACK#, ERROR#	Pull up through 2.2-kΩ resistor to Vcc5_DB25_DR. Decouple through 180-pF cap to GND.
LFRAME#	No required pull-up resistor
LDRQ#0	No required pull-up resistor
STROBE#, ALF#, SLCTIN#, PAR_INIT#	Signal passes through a 33-Ω resistor and is pulled up through a 2.2-kΩ resistor to Vcc5_DB25_CR. Decoupled using a 180-pF cap to GND.
PWM1, PWM2	Pull up to 4.7 kΩ to Vcc3_3 and connected to jumper for decouple with 0.1-µF cap to GND.
INDEX#, TRK#0, RDATA#, DSKCHG#, WRTPRT#	Pull up through 1-kΩ resistor to Vcc5.
PDR0, PDR1, PDR2, PDR3, PDR4, PDR5, PDR6, PDR7	Passes through 33-Ω resistor. Pull up through 2.2 kΩ to Vcc5_DB5_CRD and couple through 180-pF cap to GND.
SYSOPT	Pull down with 4.7-kΩ resistor to GND or IO address of 0x02E.

11.6. System Checklist

Checklist Items	Recommendations
KEYLOCK#	Pull up through 10-kΩ resistor to Vcc3_3.
PBTN_IN	Connects to PBSwitch and PBin.
PWRLED	Pull up through a 220-Ω resistor to Vcc5.
R_IRTX	Signal IRTX after it is pulled down through 4.7-kΩ resistor to GND and passes through 82-Ω resistor.
IRRX	Pull up to 100-kΩ resistor to Vcc3_3. When signal is input for SI/O decouple through 470-pF cap to GND
IRTX	Pull down through 4.7 kΩ to GND. Signal passes through 82-Ω resistor. When signal is input to SI/O decouple through 470-pF cap to GND
FP_PD	Decouple through a 470-pF cap. To GND. Pull up 470 Ω to Vcc5.
PWM1, PWM2	Pull up through a 4.7-kΩ resistor to Vcc3_3.

11.7. FWH Checklist

Checklist Items	Recommendations
No floating inputs	Unused FGPI pins must be tied to a valid logic level.
WPROT, TBLK_LCK	Pull up through a 4.7-kΩ resistor to Vcc3_3.
R_VPP	Pulled up to Vcc3_3 and decoupled with two 0.1-µF caps to GND.
FGPI0_PD, FGPI1_PD, FGPI2_PD, FPGI3_PD, FPGI4_PD, IC_PD	Pull down through a 8.2-kΩ resistor to GND.
FWH_ID1, FWH_ID2, FWH_ID3	Pull down to GND.
INIT#	FWH INIT# must be connected to processor INIT#.
RST#	FWH RST# must be connected to PCIRST#.
ID[3:0]	For a system with only one FWH device, tie ID[3:0] to ground.

11.8. Clock Synthesizer Checklist

Checklist Items	Recommendations
REFCLK	Connects to R-RefCLK, USB_CLK, SIO_CLK14, and ICHCLK14.
ICH_3V66/3V66_0	Passes through 33-Ω resistor. When signal is input for ICH, it is pulled down through a 18-pF cap to GND.
DCLK/DCLK_WR	Passes through 33-Ω resistor. When signal is input for MCH, it is pulled down through a 22-pF cap to GND.
CPUHCLK/CPU_0_1	Passes through 33-Ω resistor. When signal is input for 370PGA, decouple through a 18-pF cap to GND.
R_REFCLK	REFCLK passed through 10-kΩ resistor. When signal is input for 370PGA, pull up through 1-kΩ resistor to Vcc3_3 and pass through 10-kΩ resistor.
USB_CLK, ICH_CLK14	REFCLK passed through 10-Ω resistor.
XTAL_IN, XTAL_OUT	Passes through 14.318-MHz oscillator. Pulled down through 18-pF cap to GND.
SEL1_PU	Pulled up via MEMV3 circuitry through 8.2-kΩ resistor.
FREQSEL	Connected to clock frequency selection circuitry through 10-kΩ resistor. (See CRB schematic, page 4.)
L_VCC2_5	Connects to VDD2_5[0…1] through ferrite bead to Vcc2_5.
MCHHCLK/CPU_1, ITPCLK/CPU_2, PCI_0/PCLK_OICH, PCI_1/PCLK_1, PCI_2/PCLK_2, PCI_3/PCLK_3, PCI_4/PCLK_4, PCI_5/PCLK_5, PCI_6/PCLK_6, APICCLK_CPU/APIC_0, APICCLK)ICH/APIC_1, USBCLK/USB_0, MCH_3V66/3V66_1, AGPCLK_CONN	Passes through 33-Ω resistor.
MEMCLK0/DRAM_0, MEMCLK1/DRAM_1, MEMCLK2/DRAM_2, MEMCLK3/DRAM_3, MEMCLK4/DRAM_4, MEMCLK5/DRAM_5, MEMCLK6/DRAM_6, MEMCLK7/DRAM_7	Pass through 10-Ω resistor.
SCLK	Pass through 22-Ω resistor.

11.9. ITP Probe Checklist

Checklist Items	Recommendations
R_TCK, TCK R_TMS, TMS	Connect to 370-Pin socket through 47-Ω resistor and pull up to VCMOS.
ITPRDY#, R_ITPRDY#	Connect to 370-Pin socket through 243-Ω resistor.
TDI	Pull up through 330-Ω resistor to VCMOS.
TDO	Pull up through 150-Ω resistor to VCMOS.
PLL1	See Design Guide.
PLL2	See Design Guide.

11.10. System Memory Checklist

Checklist Items	Recommendations
SM_CSA#[0:3, SM_CSB#[3:0, SMAA[11:8,3:0], SM_MD[0:63], SM_CKE[0:3], S_DQM[0:7]	Connect from MCH to DIMM0, DIMM1.
SM_MAA[7:4], SM_MAB[7:4]#	Connect from MCH to DIMM0, DIMM1 through 10-Ω resistors.
SM_CAS#	Connected to R_REFCLK through 10-kΩ resistor.
SM_RAS#	Jumpered to GND through 10-kΩ resistor.
SM_WE#	Connected to R_BSEL0# through 10-kΩ resistor.
CKE[5…0] (For 3-DIMM implementation)	When implementing a 3-DIMM configuration, all six CKE signals on the MCH are used. (0,1 for DIMM0; 2, 3 for DIMM1; 4,5 for DIMM2)
REGE	Connect to GND (since the Intel® 815EP chipset does not support registered DIMMs).
WP(Pin 81 on the DIMMS)	Add a 4.7-kΩ pull-up resistor to 3.3 V. This recommendation write-protects the DIMM's EEPROM.
SRCOMP	Needs a 40-Ω resistor pulled up to 3.3 V.

11.11. Power Delivery Checklist

Checklist Items	Recommendations
All voltage regulator components meet maximum current requirements.	Consider all loads on a regulator, including other regulators.
All regulator components meet thermal requirements.	Ensure the voltage regulator components and dissipate the required amount of heat.
VCC1_8	VCC1_8 power sources must supply 1.85 V and be between (1.795 V to 1.905 V).
If devices are powered directly from a dual rail (i.e., not behind a power regulator), then the RDSon of the FETs used to create the dual rail must be analyzed to ensure there is not too much voltage drop across the FET.	“Dual” voltage rails may not be at the expected voltage.
Dropout voltage	The minimum dropout for all voltage regulators must be considered. Take into account that the voltage on a dual rail may not be the expected voltage.
Voltage tolerance requirements are met.	See the individual component specifications for each voltage tolerance.
Total power consumption in S3 must be less than the rated standby supply current.	Adequate power must be supplied by power supply.

This page is intentionally left blank.

12. Third-Party Vendor Information

This design guide has been compiled to give an overview of important design considerations while providing sources for additional information. This chapter includes information regarding various third-party vendors who provide products to support the 815EP chipset. The list of vendors can be used as a starting point for the designer. Intel does not endorse any one vendor, nor guarantee the availability or functionality of outside components. Contact the manufacturer for specific information regarding performance, availability, pricing and compatibility.

Super I/O (Vendors Contact Phone)

- SMSC — Dave Jenoff (909) 244-4937
- Natiobnal Semiconductor — Robert Reneau (408) 721-2981
- ITE — Don Gardenhire (512)388-7880
- Winbond — James Chen (02) 27190505 - Taipei office

Clock Generation (Vendors Contact Phone)

- Cypress Semiconductor — John Wunner 206-821-9202 x325
- ICS — Raju Shah 408-925-9493
- IMI — Elie Ayache 408-263-6300, x235
- PERICOM — Ken Buntaran 408-435-1000

Memory Vendors

http://developer.intel.com/design/motherbd/se/se_mem.htm

Voltage Regulator Vendors (Vendors Contact Phone)

- Linear Tech Corp. — Stuart Washino 408-432-6326
- Celestica — Dariusz Basarab 416-448-5841
- Corsair Microsystems — John Beekley 888-222-4346
- Delta Electronics — Colin Weng 886-2-6988, x233(Taiwan)
- N. America: Delta Products Corp. — Maurice Lee 510-770-0660, x111