Intel® 810E2 Chipset Platform

Design Guide

For Use With Universal Socket 370

August 2002

Document Number: 298303-002

Contents

Figures

Tables

Revision History

Revision	Version	Description	Date
-001	1.0	Initial Release.	Sept 2001
-002	1.0	Replaced Table 7 in Section 5.2, General Topology and Layout Guidelines Replaced Figure 91, Power Delivery Map in Section 8, Power Delivery Added SUSCLK in Section 9.2.13, RTC Checklist Revised Section 9.2.16 Power Checklist Revised Section 3.3.3 3.3V/5VREF Sequencing Replaced Figure 67 in Section 6.20.2.1, General Trace Routing Considerations Added Figure 63, Power-well Isolation Control, in Section 6.19.8, Power Well Isolation Control Revised V5REFSUS in Section 9.2.16, Power Checklist Revised APIC Checklist in Section 9.2.6, Interrupt Interface	Aug 2002

This page is intentionally left blank.

1 Introduction

This design guide organizes Intel's design recommendations for the Intel® 810E2 for use with the universal socket 370 platform. Motherboard design recommendations such as layout and routing guidelines are covered. In addition, this document also addresses system design issues (e.g., thermal requirements for the 810E2 chipset universal platform).

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

Consult the debug recommendations when debugging a platform based on the 810E2 chipset for use with universal socket 370. However, these debug recommendations should be understood before completing board design, to ensure that the debug port, in addition to other debug features, are implemented correctly.

The 810E2 chipset platform supports the following processors:

- Intel® Pentium® III processor based on 0.18 micron technology (CPUID = 068xh).
- Intel® Celeron® processor based on 0.18 micron technology (CPUID = 068xh). This applies to 533A MHz and ≥566 MHz Celeron processors
- Future 0.13 micron socket 370 processors

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

Note: The 810E2 chipset for use with the universal socket 370 is **not** compatible with the Intel® Pentium® II processor (CPUID = 066xh) 370-pin socket.

There are four chipsets in the 810 chipset family:

1. Intel® 810A3 Chipset. Components are 82810A3 GMCH and the 82801AA ICH.
2. Intel® 810E Chipset. Components are 82810E GMCH and the 82801AA ICH.
3. Intel® 810E2 Chipset. Components are 82810E2 GMCH and the 82801BA ICH2.
4. Intel® 810E2 Chipset for use with Universal Socket 370. Components are 82810E2 GMCH for use with the Universal Socket 370 and the 82801BA ICH2.

Each of these four chipsets has a separate design guide. This design guide is #4 above. It is intended to allow the use of existing 82810E2 A3 stepping devices with future 0.13 micron socket 370 processors.

1.1 Terminology

Term	Definition
Aggressor	A network that transmits a coupled signal to another network is called the aggressor network.
AGP	Accelerated Graphics Port
AGTL/AGTL+	Refers to processor bus signals that are implemented using either Assisted Gunning Transceiver Logic (AGTL+) or its lower voltage variant (AGTL), depending on which processor is being used.
Bus Agent	A component or group of components that, when combined, represent a single load on the AGTL+ bus.
Core power rail	A power rail that is only on during *full-power* operation. These power rails are on when the PSON signal is asserted to the ATX power supply. The core power rails that are distributed *directly* from the ATX power supply are: ±5V, ±12V and +3.3V.
Corner	Describes how a component performs when all parameters that could impact performance are adjusted to have the same impact on performance. Examples of these parameters include variations in manufacturing process, operating temperature, and operating voltage. The results in performance of an electronic component that may change as a result of corners include (but are not limited to): clock to output time, output driver edge rate, output drive current, and input drive current. Discussion of the “slow” corner would mean having a component operating at its slowest, weakest drive strength performance. Similar discussion of the “fast” corner would mean having a component operating at its fastest, strongest drive strength performance. Operation or simulation of a component at its slow corner and fast corner is expected to bound the extremes between slowest, weakest performance and fastest, strongest performance.
Crosstalk	The reception on a victim network of a signal imposed by aggressor network(s) through inductive and capacitive coupling between the networks. • Backward Crosstalk–coupling that creates a signal in a victim network that travels in the opposite direction as the aggressor’s signal. • Forward Crosstalk–coupling that creates a signal in a victim network that travels in the same direction as the aggressor’s signal. • Even Mode Crosstalk–coupling from single or multiple aggressors when all the aggressors switch in the same direction that the victim is switching. • Odd Mode Crosstalk–coupling from single or multiple aggressors when all the aggressors switch in the opposite direction that the victim is switching.
Derived power rail	A *derived* power rail is any power rail that is generated from another power rail using an on-board voltage regulator. For example, 3.3VSB is usually derived (on the motherboard) from 5VSB using a voltage regulator.
Dual power rail	A dual power rail is derived from different rails at different times (depending on the power state of the system). Usually, a dual power rail is derived from a *standby supply* during *suspend* operation and derived from a *core supply* during *full-power* operation.

Term	Definition
Flight Time	Flight Time is a term in the timing equation that includes the signal propagation delay, any effects the system has on the T_{CO} of the driver, plus any adjustments to the signal at the receiver needed to guarantee the setup time of the receiver. More precisely, *flight time* is defined to be: The time difference between a signal at the input pin of a receiving agent crossing VREF (adjusted to meet the receiver manufacturer's conditions required for AC timing specifications; i.e., ringback, etc.), and the output pin of the driving agent crossing VREF if the driver was driving the Test Load used to specify the driver's AC timings. The VREF Guardband takes into account sources of noise that may affect the way an AGTL+ signal becomes valid at the receiver. See the definition of the VREF Guardband. • **Maximum and Minimum Flight Time** - Flight time variations can be caused by many different parameters. The more obvious causes include variation of the board dielectric constant, changes in load condition, crosstalk, VTT noise, VREF noise, variation in termination resistance and differences in I/O buffer performance as a function of temperature, voltage and manufacturing process. Some less obvious causes include effects of *Simultaneous Switching Output (SSO)* and packaging effects. — The Maximum Flight Time is the largest flight time a network will experience under all variations of conditions. Maximum flight time is measured at the appropriate VREF Guardband boundary. — The Minimum Flight Time is the smallest flight time a network will experience under all variations of conditions. Minimum flight time is measured at the appropriate VREF Guardband boundary. For more information on flight time and the VREF Guardband, see the *Intel® Pentium® II Processor Developer's Manual*.
Full-power operation	During *full-power* operation, all components on the motherboard remain powered. Note that *full-power* operation includes both the *full-on* operating state (S0) and the processor Stop Grant state (S1).
GMCH	Graphics and Memory Controller Hub. A component of the Intel® 810E2 chipset platform for use with the Universal Socket 370.
GTL+	GTL+ is the bus technology used by the Pentium Pro processor. This is an incident wave switching, open-drain bus with pull-up resistors that provide both the high logic level and termination. It is an enhancement to the GTL (Gunning Transceiver Logic) technology. See the *Intel® Pentium® II Processor Developer's Manual* for more details of GTL+.
Intel® ICH2	Intel® 82801BA I/O Controller Hub component.
ISI	Inter-symbol interference is the effect of a previous signal (or transition) on the interconnect delay. For example, when a signal is transmitted down a line and the reflections due to the transition have not completely dissipated, the following data transition launched onto the bus is affected. ISI is dependent upon frequency, time delay of the line, and the reflection coefficient at the driver and receiver. ISI can impact both timing and signal integrity.
Network	The trace of a Printed Circuit Board (PCB) that completes an electrical connection between two or more components.
Network Length	The distance between agent 0 pins and the agent pins at the far end of the bus.
Overdrive Region	Is the voltage range, at a receiver, located above and below VREF for signal integrity analysis. See the *Intel® Pentium® II Processor Developer's Manual* for more details.

Term	Definition
Overshoot	Maximum voltage allowed for a signal at the processor core pad. See each processor's datasheet for overshoot specification.
Pad	The electrical contact point of a semiconductor die to the package substrate. A pad is only observable in simulation.
Pin	The contact point of a component package to the traces on a substrate such as the motherboard. Signal quality and timings can be measured at the pin.
Power rails	An ATX power supply has 6 power rails: +5V, -5V, +12V, -12V, +3.3V, +5VSB. In addition to these power rails, several other power rails can be created with voltage regulators.
Ringback	The voltage that a signal rings back to after achieving its maximum absolute value. Ringback may be due to reflections, driver oscillations, or other transmission line phenomena.
Settling Limit	Defines the maximum amount of ringing at the receiving pin that a signal must reach before its next transition. See the respective processor's datasheet for settling limit specification.
Setup Window	The time between the beginning of Setup to Clock (T_{SU_MIN}) and the arrival of a valid clock edge. This window may be different for each type of bus agent in the system.
Simultaneous Switching Output (SSO)	Simultaneous Switching Output (SSO) Effects refers to the difference in electrical timing parameters and degradation in signal quality caused by multiple signal outputs simultaneously switching voltage levels (e.g., high-to-low) in the opposite direction from a single signal (e.g., low-to-high) or in the same direction (e.g., high-to-low). These are respectively called odd-mode switching and even-mode switching. This simultaneous switching of multiple outputs creates higher current swings that may cause additional propagation delay (or "push-out"), or a decrease in propagation delay (or "pull-in"). These SSO effects may impact the setup and/or hold times and are not always taken into account by simulations. System timing budgets should include margin for SSO effects.
Standby power rail	A power rail that in on during *suspend* operation (these rails are also on during *full-power* operation). These rails are on at all times (when the power supply is plugged into AC power). The only standby power rail that is distributed *directly* from the ATX power supply is 5VSB (5V Standby). There can be other standby rails that are created with voltage regulators.
Stub	The branch from the bus trunk terminating at the pad of an agent.
Suspend operation	During suspend operation, power is removed from some components on the motherboard. The customer reference board supports three suspend states: processor Stop Grant (S1), Suspend-to-RAM (S3) and Soft-off (S5).
Suspend-to-RAM (STR	In the STR state, the system state is stored in main memory and all unnecessary system logic is turned off. Only main memory and logic required to *wake* the system remain powered.
System Bus	The system bus is the processor bus.
Test Load	Intel uses a 50 Ω test load for specifying its components.
Trunk	The main connection, excluding interconnect branches, from one end agent pad to the other end agent pad.
Undershoot	Minimum voltage observed for a signal to extend below VSS at the device pad.

Term	Definition
Universal Socket 370	Refers to the 810E2 chipset using the “universal” PGA370 socket. In general, these designs support 66/100/133 MHz system bus operation, Intel VRM guidelines for future 0.13 micron processors, and Intel® Celeron processors (CPUID=068xh), Intel Pentium III processor (CPUID=068xh), and future Pentium® III processors in single-microprocessor based designs.
Victim	A network that receives a coupled crosstalk signal from another network is called the victim network.
VREF Guardband	A guardband (DVREF) defined above and below VREF to provide a more realistic model accounting for noise such as crosstalk, VTT noise, and VREF noise.

1.2 Related Documents

Document and Location	Location
Intel® 810E Chipset: 82810E Graphics and Memory Controller Hub (GMCH) Datasheet	http://developer.intel.com/design/chipsets/datashts/290676.htm
Intel® 82801BA I/O Controller Hub 2 (ICH2) and Intel® 82801 BAM I/O Controller Hub 2 Mobile (ICH2-M) Datasheet	http://developer.intel.com/design/chipsets/datashts/290687.htm
Intel® 82802AB/AC Firmware Hub (FWH) Datasheet	http://developer.intel.com/design/chipsets/datashts/290658.htm
Intel® Celeron® Processor Datasheet	http://developer.intel.com/design/celeron/datashts/243658.htm
Intel® Celeron® Processor Specification Update	http://developer.intel.com/design/celeron/specupdt/243748.htm
CK 810E Clock Synthesizer Driver Specification, r 0.9	Note 1
PPGA 370 Power Delivery Guidelines	Note 1
Pentium® III Processor for the SC242 at 450 MHz to 1.13 GHz Datasheet	http://developer.intel.com/design/pentiumiii/datashts/244452.htm
Intel® Pentium® III Processor Specification Update	http://developer.intel.com/design/pentiumiii/specupdt/244453.htm
Intel® Pentium® III Power Distribution Guidelines (AP-907) Application Note	http://developer.intel.com/design/pentiumiii/applnots/245085.htm
PCI Local Bus Specification, Revision 2.2	http://www.pcisig.com/
Universal Serial Bus Specification, Revision 1.0	http://www.usb.org/developers/docs.html
Intel® 82562ET Platform LAN Connect (PLC) Datasheet	Note 1
PCB Design for the Intel® 82562 ET/EM Platform LAN Connect	Note 1
Intel® 82562EH HomePNA 1 Mb/s Physical Layer Interface Brief Datasheet	http://www.intel.com/design/network/index.htm

NOTES:
1. Contact your Intel representative for the current revision of this document.

1.3 System Overview

The Intel 810E2 chipset platform for use with the universal socket 370 contains a Graphics and Memory Controller Hub (GMCH) component and I/O Controller Hub 2 (ICH2) component for desktop platforms.

The GMCH provides the processor interface optimized for future 0.13 micron 370 socket processors, DRAM interface, hub interface, and internal graphics. This product provides flexibility and scalability in graphics and memory subsystem performance.

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 GMCH and the I/O controller hub. The chipset architecture also enables a security and manageability infrastructure through the firmware hub component.

An ACPI-compliant Intel 810E2 chipset universal 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.

The Intel 810E2 chipset contains two core components:

- Intel 82810E2 Graphics and Memory Controller Hub (GMCH)
- Intel 82801BA I/O Controller Hub 2 (ICH2)

The GMCH integrates a 66/100/133 MHz, P6 family system bus controller, integrated 2D/3D graphics accelerator, 100 MHz SDRAM controller and a high-speed hub interface for communication with the I/O Controller Hub (ICH2). The ICH2 integrates an Ultra ATA/100 controller, 2 USB host controllers with a total of 4 ports, LPC interface controller, FWH Flash BIOS interface controller, PCI interface controller, AC '97 digital controller and a hub interface for communication with the GMCH.

1.3.1 System Features

The Intel 810E2 chipset platform contains two components: the Intel 82810E Graphics and Memory Controller Hub (GMCH) and the Intel 82801BA I/O Controller Hub 2 (ICH2). The GMCH integrates a 66/100/133 MHz, P6 family system bus controller, integrated 2D/3D graphics accelerator, 100 MHz SDRAM controller, and a high-speed accelerated hub architecture interface for communication with the ICH2. The ICH2 integrates an UltraATA/100 controller, two Universal Serial Bus (USB) host controllers with a total of four ports, Low Pin Count (LPC) interface controller, Firmware Hub (FWH) interface controller, PCI interface controller, AC-link, integrated LAN controller, and a hub interface for communication with the GMCH.

1.3.2 Component Features

1.3.2.1 Intel® 82810E2 GMCH Features

- Processor/System Bus Support
  - Optimized for future Pentium III processors that use 0.13 micron technology at 133 MHz system bus frequency
  - Supports 32-bit AGTL or AGTL+ bus addressing
  - Supports uniprocessor systems
  - Utilizes AGTL and AGTL+ bus driver technology (gated AGTL/AGTL+ receivers for reduced power)
- Integrated DRAM controller
  - 32 MB to 512 MB using 16Mb/64Mb/128 Mb technology
  - 100 MHz interface
  - 64-bit data interface
  - Standard Synchronous DRAM (SDRAM) support
  - Supports only 3.3V DIMM DRAM configurations
  - No registered DIMM support
  - Support for asymmetrical DRAM addressing only
  - Support for x8, x16, and X32 DRAM device widths
- Integrated Graphics Controller
  - Full 2D hardware acceleration
  - Texture-mapped 3D with point sampled, bilinear, trilinear, and anisotropic filtering
  - Hardware motion compensation assist for software MPEG/DVD decode
  - Digital Video Out interface for support of digital displays
  - Integrated 230 MHz DAC
- Optional Local Graphics Memory Controller (Display Cache)
  - 32-bit data interface
  - 133 MHz memory clock
  - Support for 1MX16 (4MB ONLY)
- Packaging/Power
  - 421 BGA
  - 1.8V core and 3.3V CMOS I/O

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

The ICH2 allows the I/O subsystem to access the rest of the system, as follows:

- Upstream accelerated hub architecture interface for access to the GMCH
- PCI 2.2 interface (6 PCI Request/Grant pairs)
- 2 channel Ultra ATA/100 Bus Master IDE controller
- USB controller (Expanded capabilities for 4 ports)
- I/O APIC
- 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
  - 1.8V (± 3% within margins of 1.795 V to 1.9 V) core and 3.3V standby

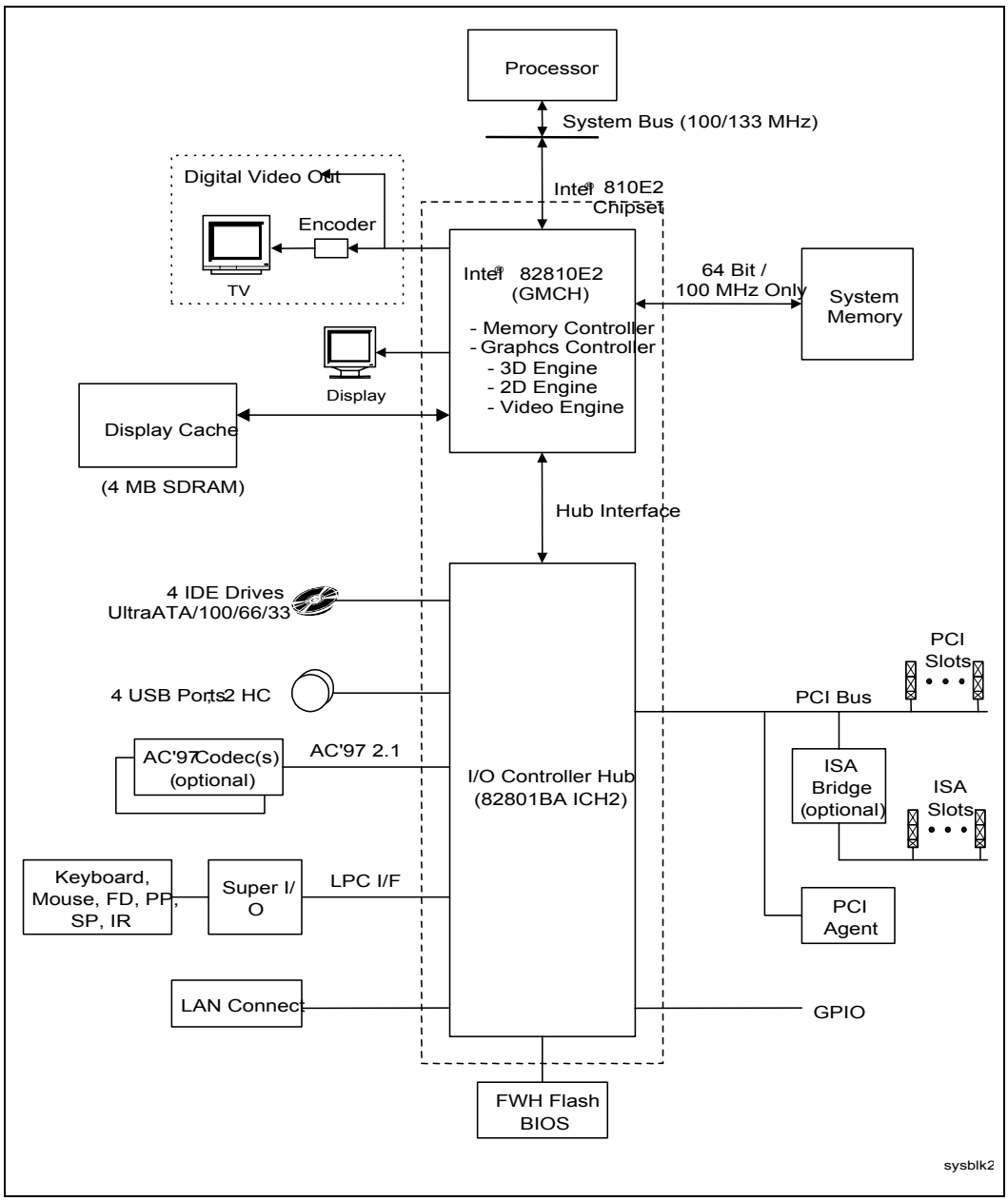
1.3.2.3 Firmware Hub (FWH)

The hardware features of the firmware hub include:

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

1.3.3 System Configurations

Figure 1. Intel® 810E2 Chipset System



1.4 Platform Initiatives

1.4.1 Hub Interface

As I/O speeds increase, the demand placed on the PCI bus by the I/O bridge has become 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 810E2 chipset's *hub interface architecture* ensures that the I/O subsystem (both PCI and the integrated I/O features such as IDE, AC'97, USB, etc.), receives adequate bandwidth. By placing the I/O bridge on the hub interface (instead of PCI), the hub architecture ensures that both the I/O functions integrated into the ICH2 and the PCI peripherals obtain the bandwidth necessary for peak performance.

1.4.2 Integrated LAN Controller

The 810E2 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.

1.4.3 Ultra ATA/100 Support

The 810E2 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 100 MB/sec, in addition to Ultra ATA/66 and Ultra ATA/33 modes.

1.4.4 Expanded USB Support

The 810E2 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.

1.4.5 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.

1.4.6 Interrupt Controller

The interrupt capabilities of the 810E2 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.

1.4.7 Firmware Hub (FWH) Flash BIOS

The 810ET2 chipset platform supports firmware hub BIOS memory sizes up to 8 MB for increased system flexibility.

1.4.8 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 810E2 chipset platform’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 810E2 chipset’s integrated AC-link reduces cost and eases migration from ISA.

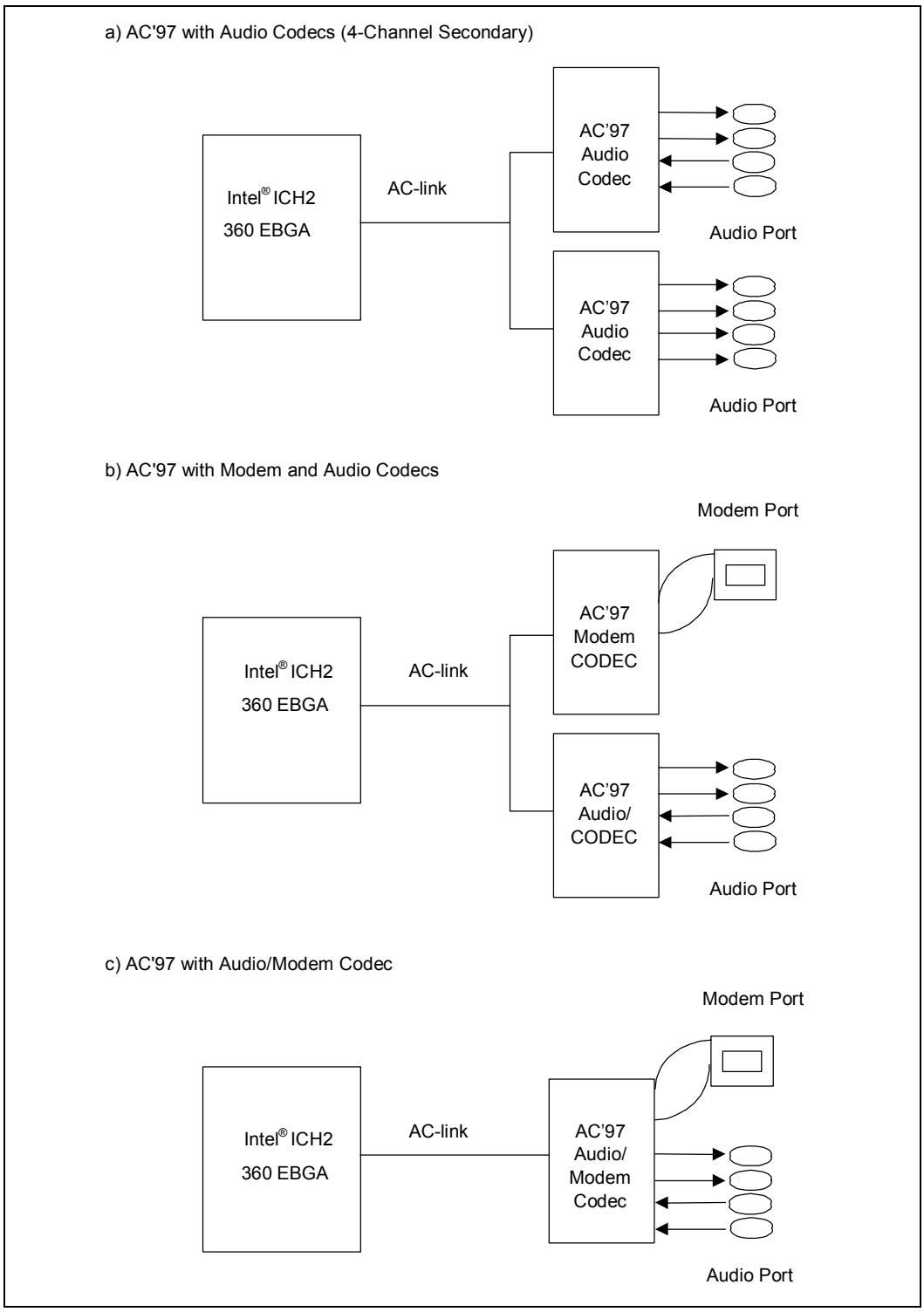
By using an audio codec, the AC-link allows for cost-effective, high-quality, integrated audio on the 810E2 chipset platform. In addition, an AC ’97 soft modem can be implemented with the use of a modem codec. Several system options exist when implementing AC ’97. The 810ET2 chipset’s integrated digital 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. The digital link is expanded to support two audio codecs or a combination of an audio and modem codec.

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 810E2 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’97 digital link.

Figure 2. AC'97 With Audio and Modem Codec Connections



1.4.9 Low Pin Count (LPC) Interface

In the 810E2 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.

This page is intentionally left blank.

2 General Design Considerations

This design guide documents motherboard layout and routing guidelines for 810E2 universal socket 370 platform-based systems. This design guide 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. As bus speeds increase, it is imperative that the guidelines documented are followed precisely. 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 crosstalk and settling time.

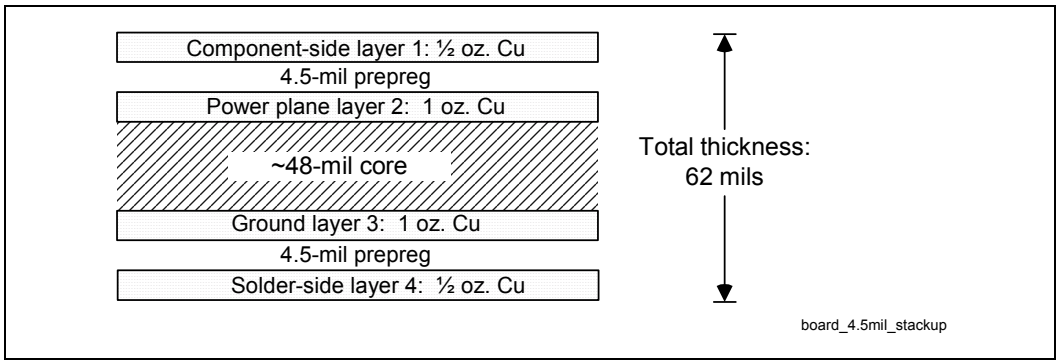
Coupling between two traces is a function of the coupled length, the distance separating the traces, the signal edge rate, and the degree of mutual capacitance and inductance. To minimize the effects of trace-to-trace coupling, follow the routing guidelines documented in this section.

Additionally, these routing guidelines are created using a PCB *stack-up* similar to that illustrated in Figure 3.

2.1 Nominal Board Stack-Up

The 810E2 chipset universal platform requires a board stack-up yielding a target impedance of 60 Ω ± 10% with a 5 mil nominal trace width. Figure 3 presents an example stack-up that achieves this. It is a 4-layer printed circuit board (PCB) construction using 53%-resin FR4 material.

Figure 3. Board Construction Example for 60 Ω Nominal Stack-Up



This page is intentionally left blank.

3 Component Layouts

Figure 4 illustrates the relative signal quadrant locations on the GMCH ballout. It does not represent the actual ballout. Refer to the *Intel® 810E Chipset: 82810E Graphics and Memory Controller Hub (GMCH)* Datasheet for the actual ballout.

Figure 4. Intel® 82810E2 GMCH 421-BGA Quadrant Layout (Top View)

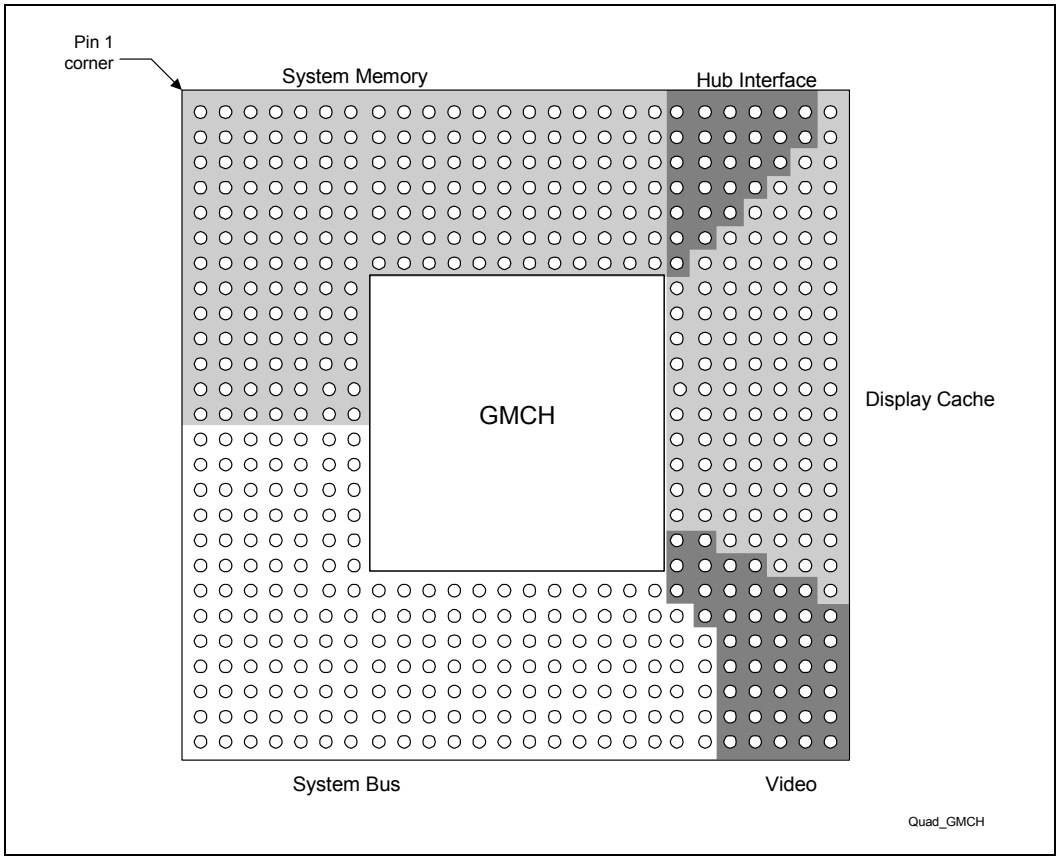


Figure 5 illustrates the relative signal quadrant locations on the ICH2 ballout. It does not represent the actual ballout. Refer to the *Intel® 82801BA I/O Controller Hub (ICH2) and Intel® 82801BAM I/O Controller Hub (ICH2-M) Datasheet* for the actual ballout.

Figure 5. Intel® ICH2 360 EBGA Quadrant Layout (Top View)

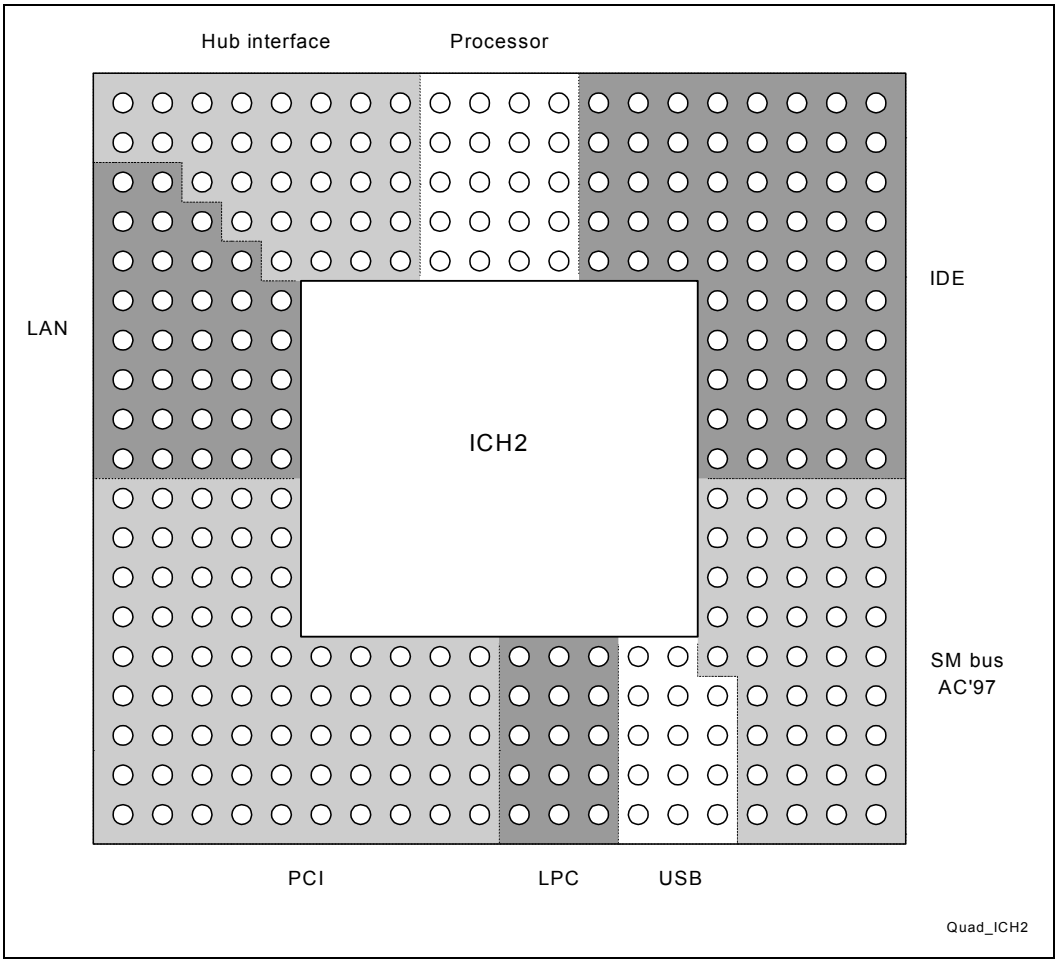
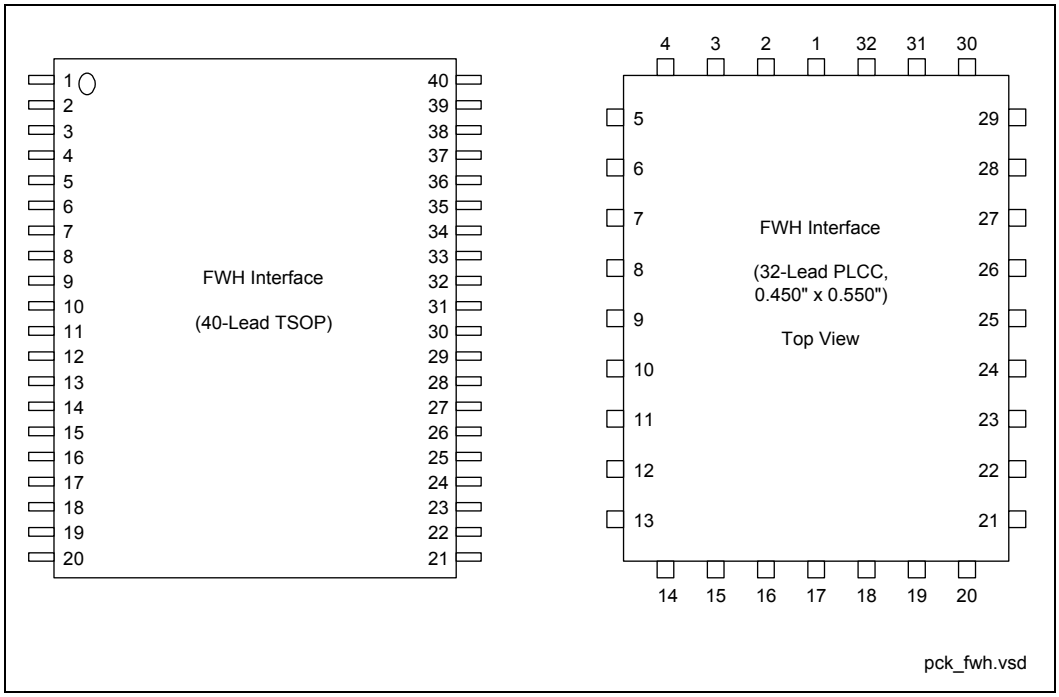


Figure 6. Firmware Hub (FWH) Packages



This page is intentionally left blank.

4 Universal Motherboard Design

4.1 Universal Motherboard Definition Details

The universal socket 370 platform supports Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) as well as future 0.13 micron socket 370 processors. The Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) have different requirements for functioning properly in a platform than the future 0.13 micron socket 370 processors. It is necessary to understand these differences and how they affect the design of the platform. Refer to Table 1 through Table 3 for a high-level description of the differences that require additional circuitry on the motherboard. Specific details on implementing this circuitry are discussed further in this chapter. For a detailed description of the pin differences between the Pentium III processor (CPUID=068xh) / Celeron processor (CPUID=068xh), and future 0.13 micron socket 370 processor pins, refer to Section 5.4.

Table 1. Processor Considerations for Universal Motherboard Design

Signal Name or Pin Number	Function In Intel® Pentium® III Processor (CPUID=068xh) and Intel® Celeron® Processor (CPUID=068xh)	Function In Future 0.13 Micron Socket 370 Processors	Implementation for Universal Socket 370 Design
AF36	VSS	No connect	Addition of circuitry that generates a processor identification signal used to configure board-level operation.
AG1	VSS	VTT	Addition of FET switch to ground or VTT, controlled by processor identification signal. **Note:** FET must have no more than 100 milliohms resistance between source and drain.
AJ3	**NC**	**NC**	**PGA 370 socket pin AJ3 is a "NC" (no connect) for either processor.**
AK22	GTL_REF	VCMOS_REF	Addition of resistor-divider network to provide 1.0V, which will satisfy voltage tolerance requirements of the Intel® Pentium III processor (CPUID=068xh) and Intel® Celeron® processor (CPUID=068xh) as well as future 0.13 micron socket 370 processors.
PICCLK	Requires 2.5V	Requires 2.0V	Addition of FET switch to provide proper voltage, controlled by processor identification signal.

Signal Name or Pin Number	Function In Intel® Pentium® III Processor (CPUID=068xh) and Intel® Celeron® Processor (CPUID=068xh)	Function In Future 0.13 Micron Socket 370 Processors	Implementation for Universal Socket 370 Design
PWRGOOD	Requires 2.5V	Requires 1.8V	Addition of resistor-divider network to provide 2.1V, which will satisfy voltage tolerance requirements of the Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) as well as future 0.13 micron socket 370 processors.
VTT	Requires 1.5V	Requires 1.25V	Modification to VTT generation circuit to switch between 1.5V or 1.25V, controlled by processor identification signal.
VTTPWRGD	Not used	Input signal to future 0.13 micron socket 370 processors to indicate that VID signals are stable	Addition of VTTPWRGD generation circuit.

Table 2. Intel® ICH2 Considerations for Universal Motherboard Design

Signal	Issue	Implementation For Universal Motherboard Design
PWROK	GMCH and CK810 must not sample BSEL[1:0] until VTTPWRGD asserted. Intel® ICH2 must not initialize before CK810 clocks stabilize	Addition of circuitry to have VTTPWRGD gate PWROK from power supply to ICH2. The ICH2 will hold GMCH in reset until VTTPWRGD asserted plus 20 ms time delay to allow CK810 clocks to stabilize

Table 3. Clock Synthesizer Considerations for Universal Motherboard Design

Signal	Issue	Implementation For Universal Motherboard Design
VDD	CK810 does not support VTTPWRGD	Addition of a FET switch, which supplies power to VDD only when VTTPWRGD is asserted. Note: FET must have no more than 100 milliohms resistance between source and drain.

4.2 Processor Design Requirements

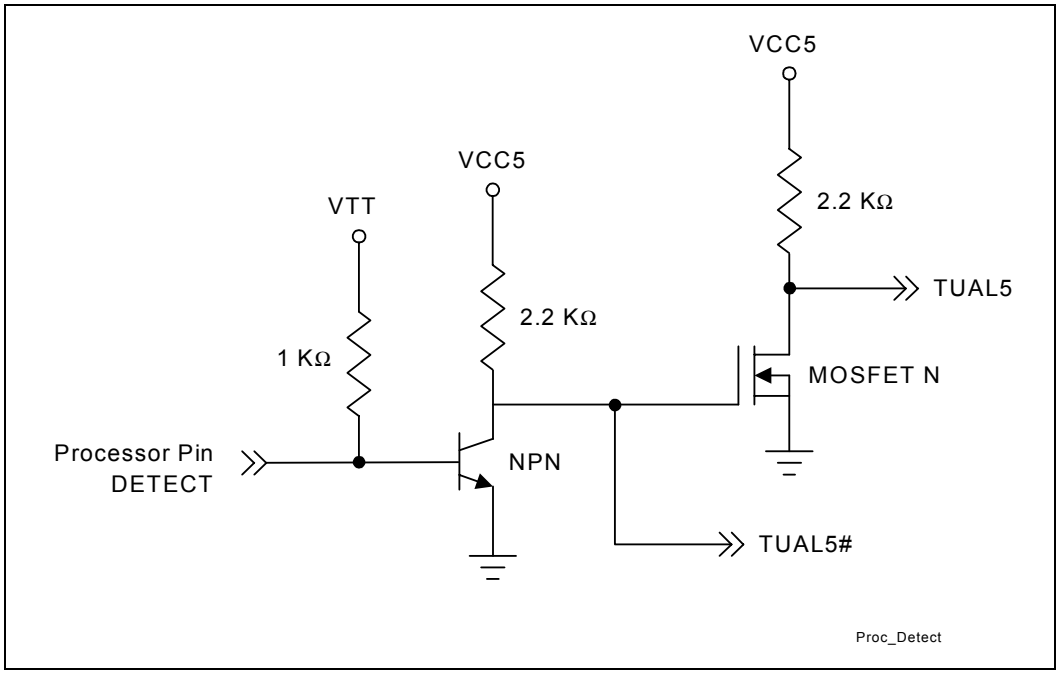
4.2.1 Use of Universal Motherboard Design with the Intel® 82810E2 GMCH

The universal socket 370 design is intended for use with the 810E2 chipset platform for use with the universal socket 370. A universal socket 370 design populated with an A3 stepping of the GMCH is compatible with future 0.13 micron socket 370 processors.

4.2.2 Identifying the Processor at the Socket

For the platform to configure for the requirements of the processor in the socket, it must first identify whether the processor is a Pentium III processor (CPUID=068xh) / Celeron processor (CPUID=068xh), or a future 0.13 micron socket 370 processors. Pin AF36 is a ground pin on a Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh); pin AF36 is an unconnected pin on future 0.13 micron Socket 370 processors. Referring to Figure 7, the platform uses a detect circuit connected to this processor pin. If a future 0.13 micron Socket 370 processor is present in the socket, the TUAL5 reference schematic signal will be pulled to the 5V rail and the TUAL5# reference schematic signal will be pulled to ground. Otherwise, for a Pentium III processor (CPUID=068xh) or Celeron processor (CPUID=068xh), the TUAL5 reference schematic signal will be pulled to ground and the TUAL5# will be pulled to the 5V rail.

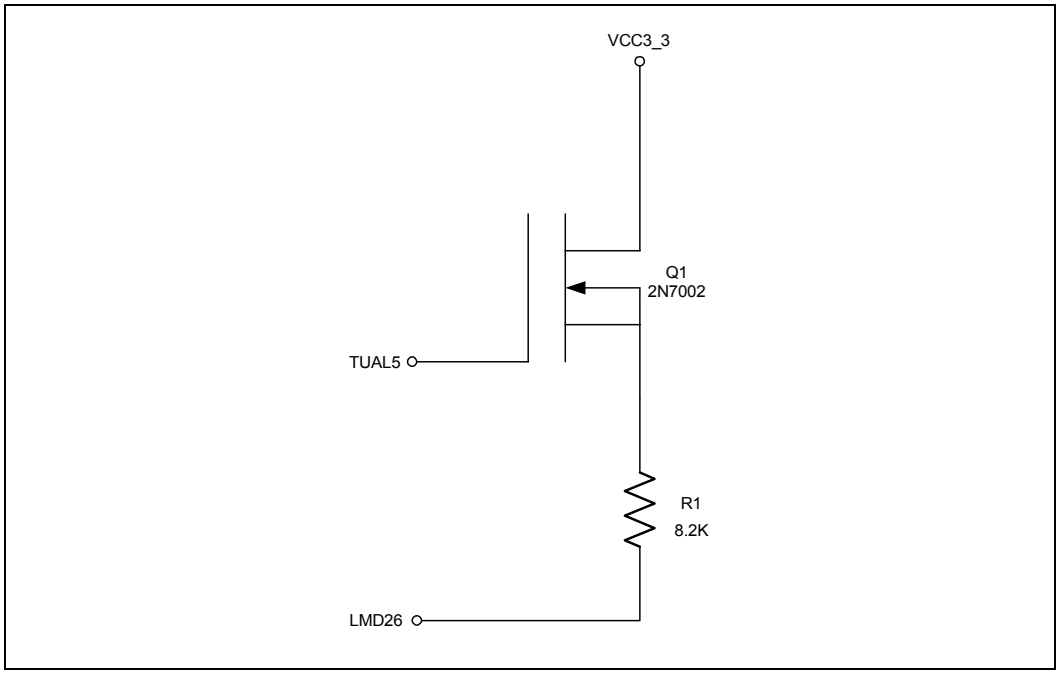
Figure 7. Processor Detect Mechanism at Socket/TUAL5 Generation Circuit



4.2.3 Setting Processor Auto-Detect

A processor auto-detect circuit for the functionality is shown as Figure 7, Section 4.2.2. A circuit that can use the processor auto-detect information to set LMD26 appropriately is shown as Figure 8 below.

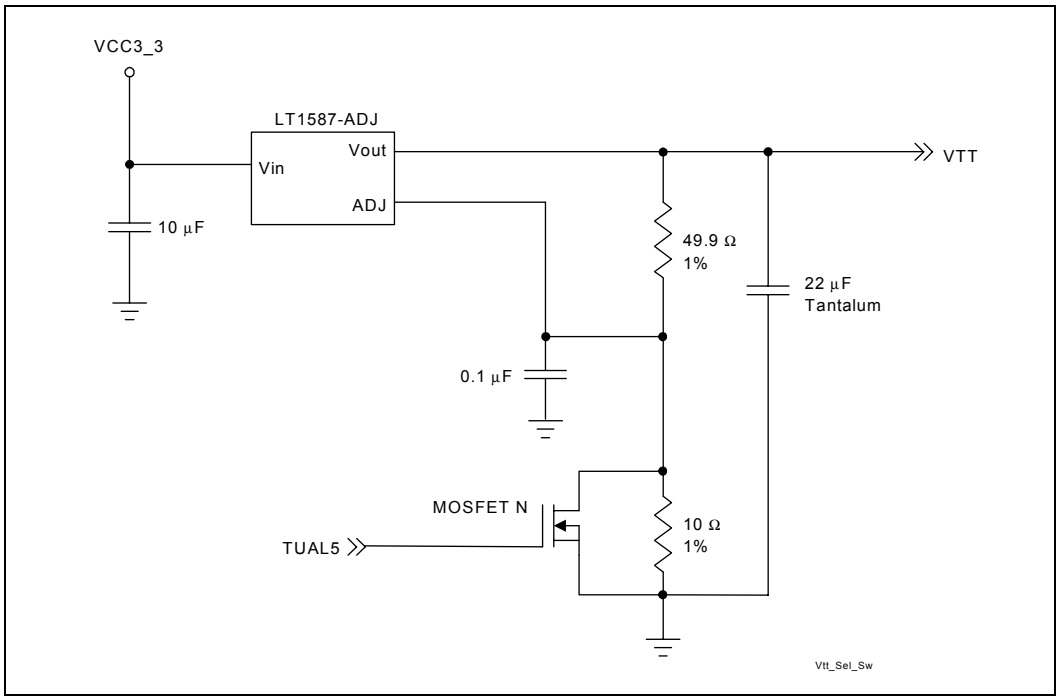
Figure 8. Processor Auto-Detect Circuit for LMD 26



4.2.4 Setting the Appropriate Processor VTT Level

Because the Pentium III processor (CPUID=068xh) / Celeron processor (CPUID=068xh), and future 0.13 micron socket 370 processors require different VTT levels, the platform must be able to provide the appropriate voltage level after determining which processor is in the socket. Referring to Figure 9, the TUAL5 reference schematic signal serves to control the FET, and by doing so determines whether the voltage regulator supplies 1.25V or 1.5V to VTT for AGTL or AGTL+, respectively.

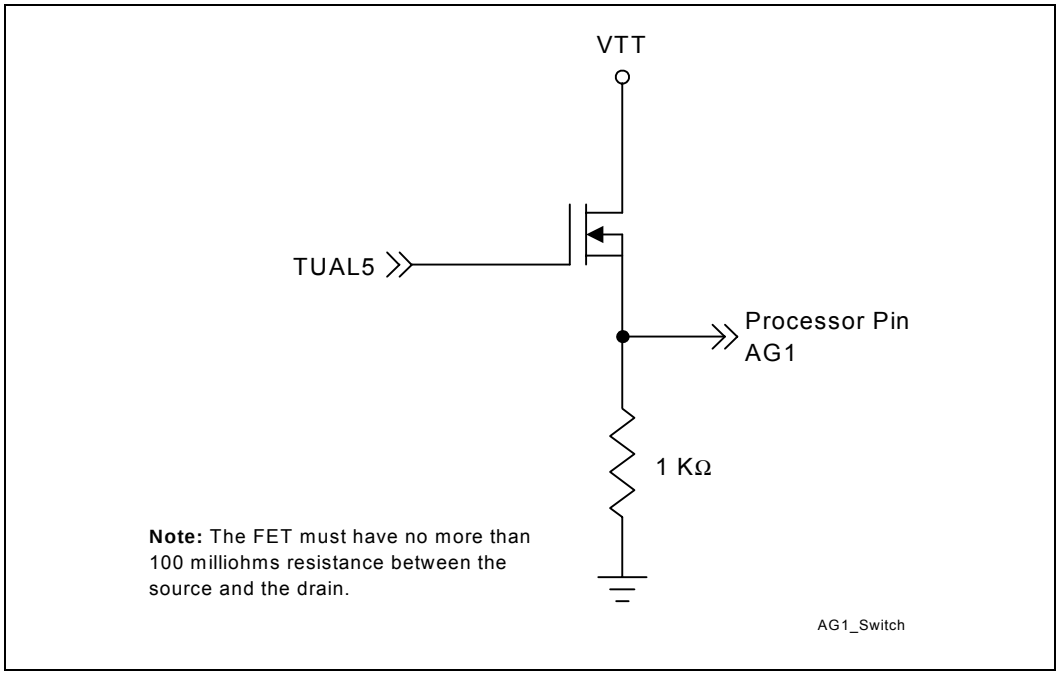
Figure 9. VTT Selection Switch



4.2.5 VTT Processor Pin AG1

Processor pin AG1 requires additional attention since it is a ground pin on a Pentium III processor (CPUID=068xh) / Celeron processor (CPUID=068xh) and a VTT pin on a future 0.13 micron socket 370 processor. A separate switch controlled by the TUAL5 reference schematic signal determines whether pin AG1 is pulled to ground or VTT. Refer to Figure 10 for an example implementation.

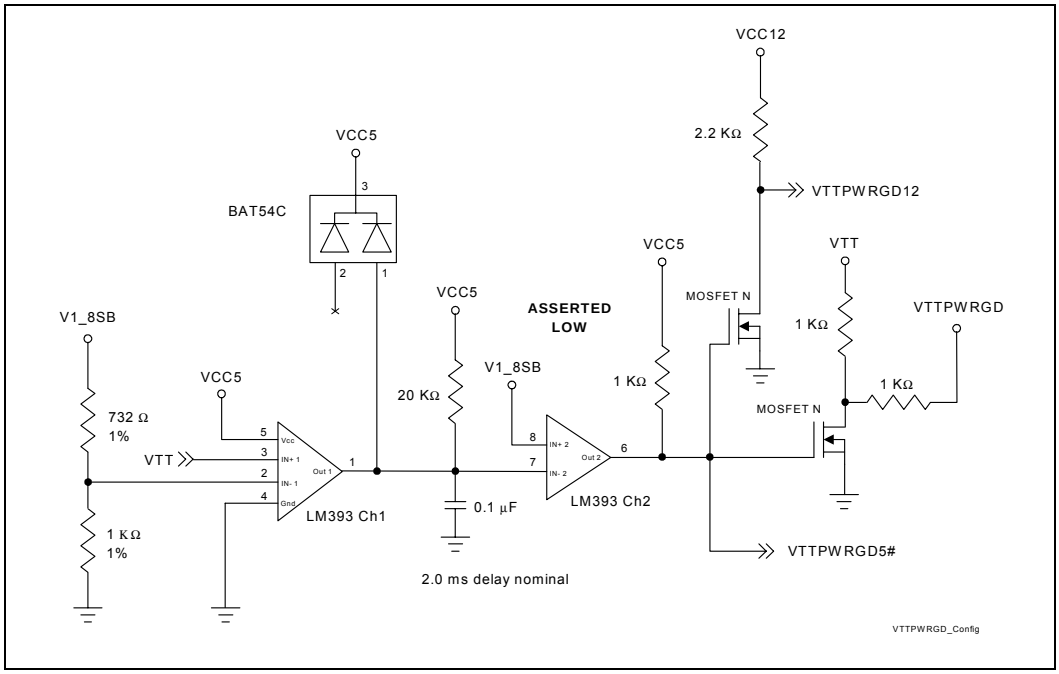
Figure 10. Switching Pin AG1



4.2.6 Configuring Non-VTT Processor Pins

When asserted, the VTTPWGRD signal must be level-shifted to 12V to properly drive the gating circuitry of the CK810. Furthermore, while the VTTPWRGD signal is connected to the VTTPWRGD pin on a future 0.13 micron socket 370 processor, on a Pentium III processor (CPUID=068xh) or Celeron processor (CPUID=068xh) that same pin is a ground. To provide proper functionality, a 1.0 kΩ resistor must be placed in series between the circuitry that generates the signal VTTPWRGD and the processor pin VTTPWRGD. Refer to Figure 11 for an example implementation. Voltage regulators that generate the standard VTTPWRGD signal are available.

Figure 11. VTTPWRGD Configuration Circuit

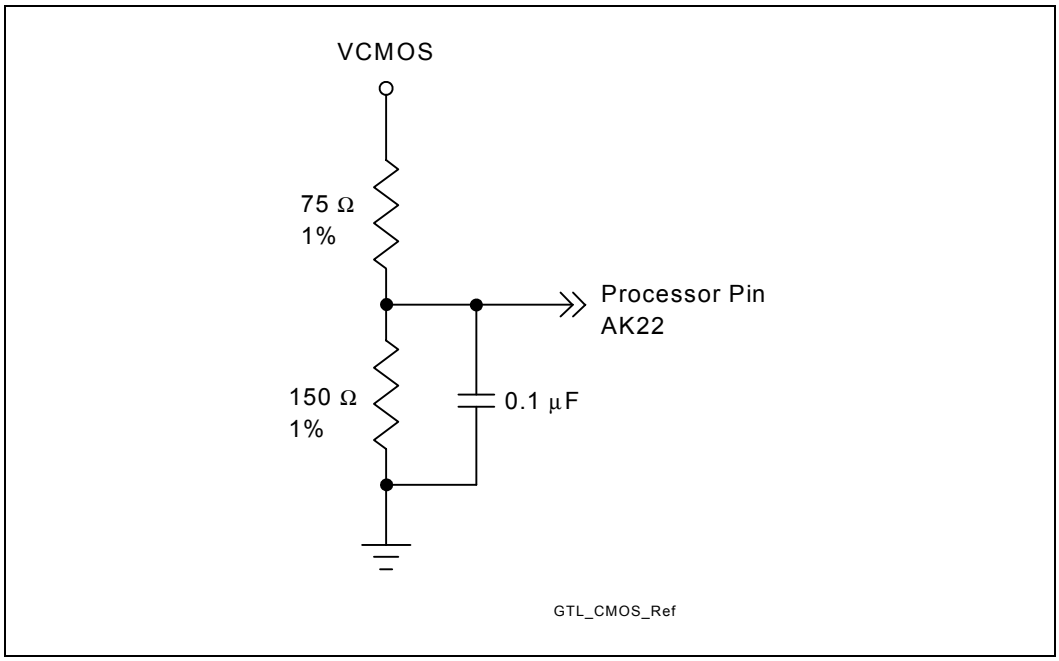


NOTE: The diode is included so that repeated pressing of the reset or power button does not cause the capacitor to build up enough charge to circumvent the 20 ms delay.

4.2.7 VCMOS Reference

In previous platforms supporting the Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh), VCMOS was generated by the same power plane as VTT. The future 0.13 micron socket 370 processors do not generate VCMOS, and the universal platform is required to generate this separately on the motherboard. Processor pin AK22, which is a GTL_REF pin on a Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh), has been changed to a VCMOS_REF pin on future 0.13 micron socket 370 processors. Referring to Figure 12, a network of resistors and a capacitor must be added so that this pin operates appropriately for whichever processor is in the socket).

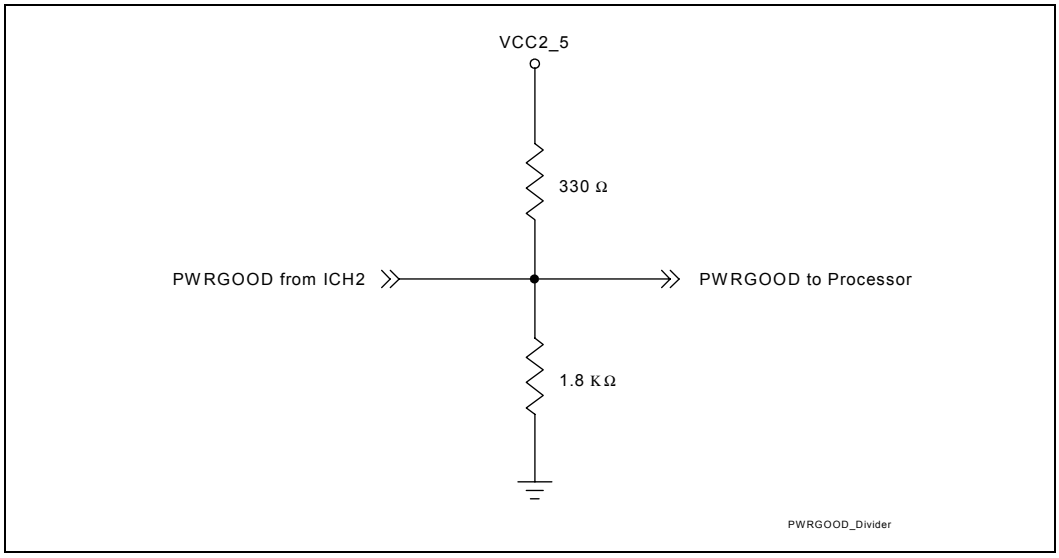
Figure 12. GTL_REF/VCMOS_REF Voltage Divider Network



4.2.8 Processor Signal PWRGOOD

The processor signal PWRGOOD is specified at different voltage levels depending on whether it is a Pentium III processor (CPUID=068xh) / Celeron processor (CPUID=068xh), or whether it is a future 0.13 micron socket 370 processor. As there is an overlap between the ranges of accepted voltage levels for these two processor groups, a resistor divider network that provides 2.1V will satisfy the requirements of all supported processors. See Figure 13 for an example implementation.

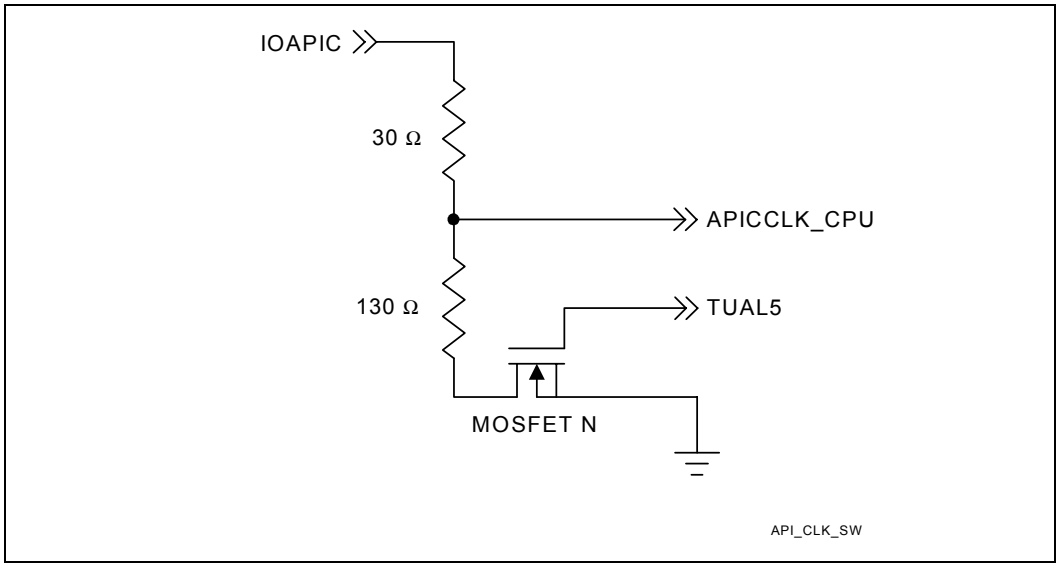
Figure 13. Resistor Divider Network for Processor PWRGOOD



4.2.9 APIC Clock Voltage Switching Requirements

The processor's APIC clock is also specified at different voltage levels depending on whether it is for the Pentium III processor (CPUID=068xh) / Celeron processor (CPUID=068xh) or whether it is for a future 0.13 micron socket 370 processor. There is no overlap in the range of accepted voltage levels for the two processor groups, so a voltage switch is required to ensure proper operation. Figure 14 shows an example implementation.

Figure 14 Voltage Switch for Processor APIC Clock

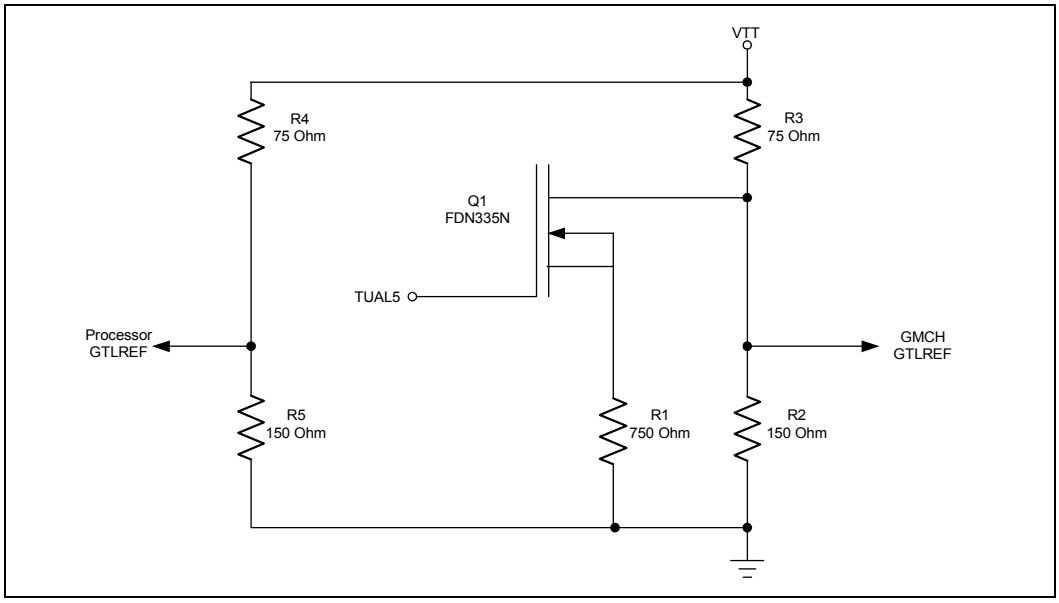


NOTE: The 30 Ω resistor represents the series resistor typically used in connecting the APIC clock to the processor.

4.2.10 GTLREF Topology and Layout

In a platform supporting the future 0.13 micron socket 370 processors, the voltage requirements for GTLREF are different for the processor and the chipset. This difference requires that separate resistor sites be added to the layout to split the GTLREF sources. In a universal motherboard design, a Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) will be unaffected by the difference in GTLREF. The recommended GTLREF circuit topology is shown in Figure 15. To auto-set the appropriate GTLREF voltage depending on which processor is used in the platform.

Figure 15. GTLREF Circuit Topology



GTLREF Layout and Routing Guidelines

- Place all resistor sites for GTLREF generation close to the GMCH.
- Route GTLREF with as wide a trace as possible.
- Use one 0.1 µF decoupling capacitor for every two GTLREF pins at the processor (four capacitors total). Place as close as possible (within 500 mils) to the Socket 370 GTLREF pins.
- Use one 0.1 µF decoupling capacitor for each of the two GTLREF pins at the GMCH (two capacitors total). Place as close as possible to the GMCH GTLREF balls.

4.3 Power Sequencing on Wake Events

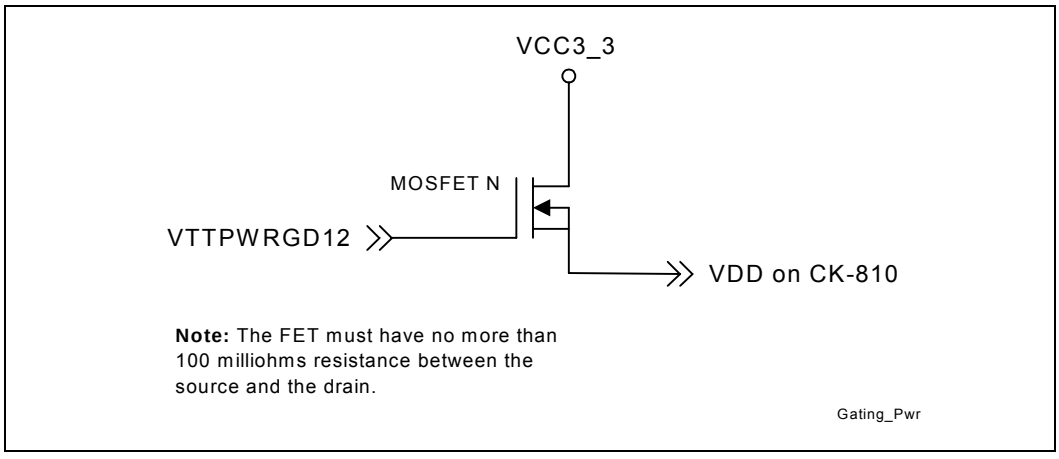
In addition to the mechanism for identifying the processor in the socket, special handling of wake events is required for 810E2 chipset universal platforms that support functionality of future 0.13 micron socket 370 processors. When a wake event is triggered, the GMCH and the CK810 must not sample BSEL[1:0] until the signal VTTPWRGD is asserted. This is handled by setting up the following sequence of events:

1. Power is not connected to the CK810-compliant clock driver until VTTPWRGD12 is asserted.
2. Clocks to the ICH2 stabilize before the power supply asserts PWROK to the ICH2. There is no guarantee this will occur as the implementation for the previous step relies on the 12 V supply. Thus it is necessary to gate PWROK to the ICH2 from the power supply while the CK810 is given sufficient time for the clocks to become stable. The amount of time required is a minimum 20 ms.
3. ICH2 takes the GMCH out of reset.
4. GMCH samples BSEL[1:0]. (CK-815 will have sampled BSEL[1:0] much earlier.)

4.3.1 Gating of CK810 to VTTPWRGD

System designers must ensure that the VTTPWRGD signal is asserted before the CK810-compliant clock driver receives power. This is handled by having the 3.3 V rail of the clock driver gated by the VTTPWRGD12 reference schematic signal. Unlike previous 810E chipset designs, the 3.3 V standby rail is not used to power the clock as the VTTPWRGD12 reference schematic signal will cut power to the clock when going into any sleep state. Refer to Figure 16 for an example implementation.

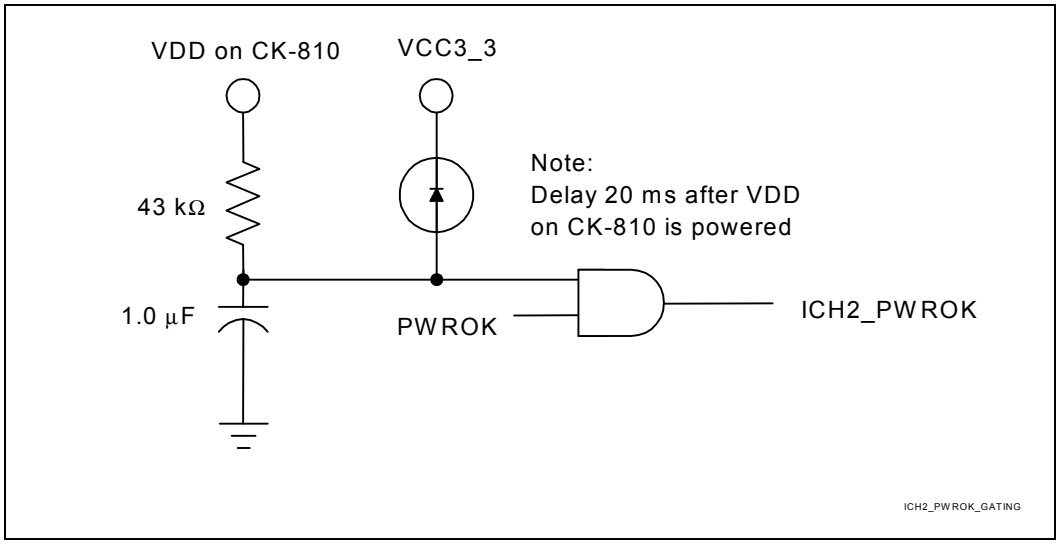
Figure 16. Gating Power to CK810



4.3.2 Gating of PWROK to Intel® ICH2

With power being gated to the CK810 by the signal VTTPWRGD12, it is important that the clocks to the ICH2 are stable before the power supply asserts PWROK to the ICH2. As the clocking power gating circuitry relies on the 12 V supply, there is no guarantee that these conditions will be met. This is why an estimated minimum time delay of 20ms must be added after power is connected to the CK810 to give the clock driver sufficient time to stabilize. This time delay will gate the power supply's assertion of PWROK to the ICH2. After the time delay, the power supply can safely assert PWROK to the ICH2, with the ICH2 subsequently taking the GMCH out of reset. Refer to Figure 17 for an example implementation.

Figure 17 PWROK Gating Circuit for Intel® ICH2



NOTE: The diode is included so that repeated pressing of the reset or power button does not cause the capacitor to build up enough charge to circumvent the 20ms delay.

This page is intentionally left blank.

5 System Bus Design Guidelines

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 higher core and system bus speeds than previous-generation IA-32 processors while maintaining hardware and software compatibility with earlier Pentium III processors. The new Flip Chip-Pin Grid Array 2 (FC-PGA2) package technology enables compatibility with previous Flip Chip-Pin Grid Array (FC-PGA) packages using the PGA370 socket.

This section presents the considerations for designs capable of using the 810E2 chipset universal platform with the full range of Pentium III processors using the PGA370 socket.

5.1 System Bus Routing Guidelines

The following layout guide supports designs using Pentium III processor (CPUID=068xh) / Celeron processor (CPUID=068xh), and future 0.13 micron socket 370 processors with the 810E2 chipset platform. The solution covers system bus speeds of 66/100/133 MHz for the Pentium III processor (CPUID=068xh) / Celeron processor (CPUID=068xh), and future 0.13 micron socket 370 processors. All processors must also be configured to 56 Ω on-die termination.

5.1.1 Initial Timing Analysis

Table 4 lists the AGTL/AGTL+ component timings of the processors and 810E2 chipset universal platform's GMCH defined at the pins. **These timings are for reference only. Obtain each processor's specifications from its respective processor datasheet and the chipset values from the appropriate 810E2 chipset datasheet.**

Table 4. Intel® Pentium® III Processor AGTL/AGTL+ Parameters for Example Calculations

IC Parameters	Intel® Pentium® III Processor at 133 MHz System Bus	Intel® 82810E2 GMCH	Notes
Clock to Output maximum (T_{CO_MAX})	3.25 ns (for 66/100/133 MHz system bus speeds)	4.1 ns	1, 2
Clock to Output minimum (T_{CO_MIN})	0.40 ns (for 66/100/133 MHz system bus)	1.05 ns	1, 2
Setup time (T_{SU_MIN})	1.20 ns for BREQ Lines 0.95 ns for all other AGTL/AGTL+ Lines @ 133 MHz 1.20 ns for all other AGTL/AGTL+ Lines @ 66/100 MHz	2.65 ns	1, 2,3
Hold time (T_{HOLD})	1.0 ns (for 66/100/133 MHz system bus speeds)	0.10 ns	1

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 GMCH sees a minimum edge rate equal to 0.3 V/ns.

Table 5 contains an example AGTL+ initial maximum flight time, and Table 6 contains an example minimum flight time calculation for a 133 MHz, uniprocessor system using the Pentium III processor and the 810E2 chipset universal platform'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.**

Table 5 and Table 6 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 CK810 Clock Synthesizer/Driver Specification.)
- CLKJITTER = 0.250 ns

See the respective processor's datasheet, the appropriate *810E2* chipset universal documentation, and the *CK810 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 5. 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	GMCH	7.50	3.25	2.65	0.20	0.25	0.40	1.1
GMCH	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 6. Example T_{FLT_MIN} Calculations (Frequency Independent)

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

NOTE: All times in nanoseconds

The flight times in Table 5 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 which can cause variation in the signals

Additional effects exist that **may not necessarily** be covered by the multi-bit adjustment factor and should be budgeted as appropriate to the baseboard design. These effects are included as M_{ADJ} in the example calculations in Table 5. Examples include:

- The effective board propagation constant (SEFF), which is a function of:
  - Dielectric constant (εr) 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.

5.2 General Topology and Layout Guidelines

Figure 18. Topology for 370-Pin Socket Designs with Single-Ended Termination (SET)

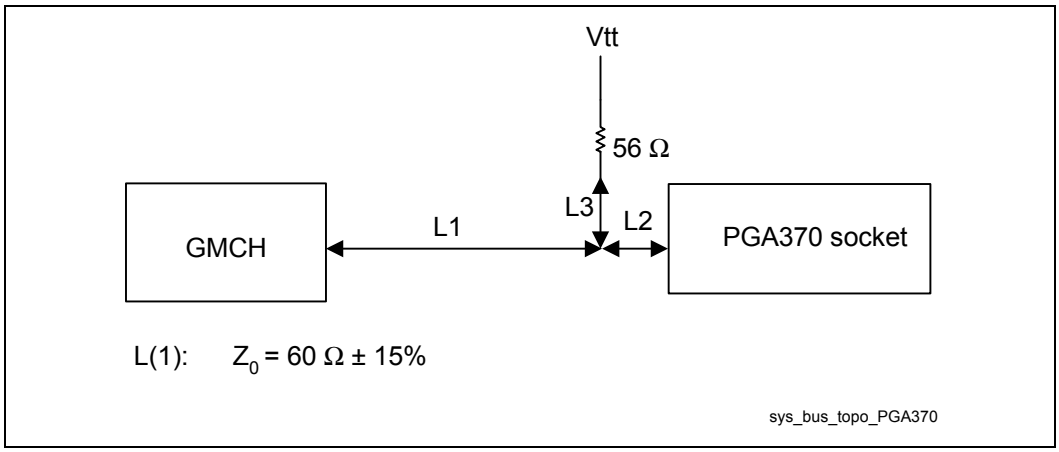


Table 7. Trace Guidelines for Figure 18

Segment	Description	Min Length (inches)	Max length (inches)
L1 + L2	Intel® 82810E GMCH 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

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

- Use an intragroup AGTL/AGTL+ spacing: line width: dielectric thickness ratio of at least 2:1:1 for microstrip geometry. If εr = 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.

Table 8 contains the trace width : space ratios assumed for this topology. Three types of crosstalk are considered in this guideline: Intragroup AGTL/AGTL+, Intergroup AGTL/AGTL+, and AGTL/AGTL+ to non-AGTL/AGTL+. Intragroup AGTL/AGTL+ crosstalk involves interference between AGTL/AGTL+ signals within the same group. Intergroup AGTL/AGTL+ crosstalk involves interference from AGTL/AGTL+ signals in a particular group to AGTL/AGTL+ signals in a different group. An example of AGTL/AGTL+ to non-AGTL/AGTL+ crosstalk is when CMOS and AGTL/AGTL+ signals interfere with each other. The AGTL/AGTL+ signals consist of the following groups: data signals, control signals, clock signals, and address signals.

Table 8. Trace Width: Space Guidelines

Crosstalk Type	Trace Width: Space Ratios[1, 2]
Intragroup AGTL/AGTL+ signals (same group AGTL/AGTL+)	5:10 or 6:12
Intergroup AGTL/AGTL+ signals (different group AGTL/AGTL+)	5:15 or 6:18
AGTL/AGTL+ to System Memory Signals	5:30 or 6:36
AGTL/AGTL+ to non-AGTL/AGTL+	5:25 or 6:24

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

5.2.1 Motherboard Layout Rules for AGTL/AGTL+ Signals

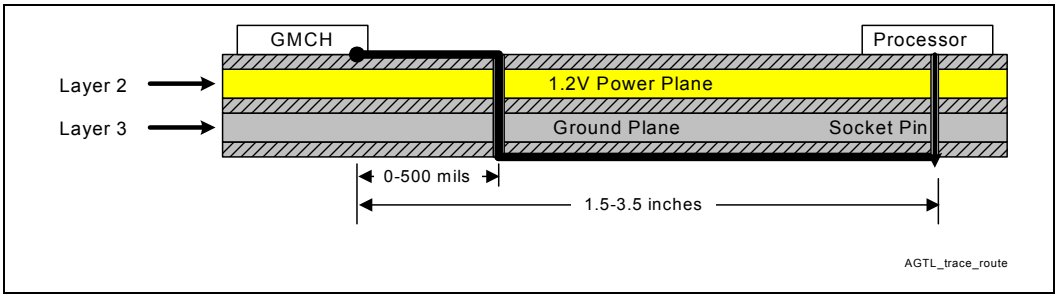
Ground Reference

It is strongly recommended that AGTL/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. If a signal has to go through routing layers, the recommendations are:

Note: Following these layout rules is critical for AGTL/AGTL+ signal integrity, particularly for 0.18 micron and smaller process technology.

For signals going from a ground reference to a power reference, add capacitors between ground and power near the vias to provide an AC return path. One capacitor should be used for every three signal lines that change reference layers. Capacitor requirements are as follows: C=100 nF, ESR=80 mΩ, ESL=0.6 nH. Refer to Figure 19 for an example of switching reference layers.

Figure 19. AGTL/AGTL+ Trace Routing



NOTE: For signals going from one ground reference to another, separate ground reference, add vias between the two ground planes to provide a better return path.

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/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.

Minimizing Crosstalk

The following general rules minimize the impact of crosstalk in a high-speed AGTL/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/AGTL+ is a low-signal-swing technology, it is important to isolate AGTL/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/AGTL+ signals must be well isolated from system memory signals. AGTL/AGTL+ signal trace edges must be at least 30 mils from system memory trace edges within 100 mils of the ball of the 82810E2 GMCH.
- Select a board stack-up that minimizes the coupling between adjacent signals. Minimize the nominal characteristic impedance within the AGTL/AGTL+ specification. Minimizing the height of the trace from its reference plane, which minimizes crosstalk, can do this.
- Route AGTL/AGTL+ address, data, and control signals in separate groups to minimize crosstalk between groups. Keep at least 15 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.

5.2.1.1 Motherboard Layout Rules for Non-AGTL/AGTL+ (CMOS) Signals

Table 9. Routing Guidelines for Non-AGTL/AGTL+ Signals

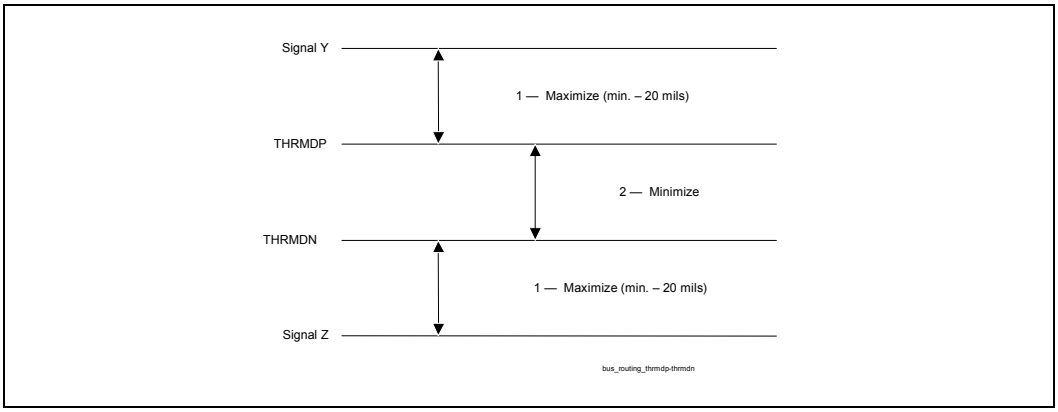
Signal	Trace Width	Spacing to Other Traces	Trace Length
A20M#	5 mils	10 mils	1” to 9”
FERR#	5 mils	10 mils	1” to 9”
FLUSH#	5 mils	10 mils	1” to 9”
IERR#	5 mils	10 mils	1” to 9”
IGNNE#	5 mils	10 mils	1” to 9”
INIT#	5 mils	10 mils	1” to 9”
LINT[0] (INTR)	5 mils	10 mils	1” to 9”
LINT[1] (NMI)	5 mils	10 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.

5.2.1.2 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 20. Routing for THRMDP and THRMDN



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

5.2.1.3 Additional Routing and Placement Considerations

- Distribute VTT with a wide trace. A 0.050 inch 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 when a Pentium III processor (CPUID=068xh) or Celeron processor (CPUID=068xh) processor are present in the socket. If a future 0.13 micron socket 370 processor is being used, the VTT voltage should then be 1.25 V ± 3% for static conditions, and 1.25 V ± 9% for worst-case transient conditions.
- Place resistor divider pairs for VREF generation at the GMCH component. VREF also is delivered to the processor.

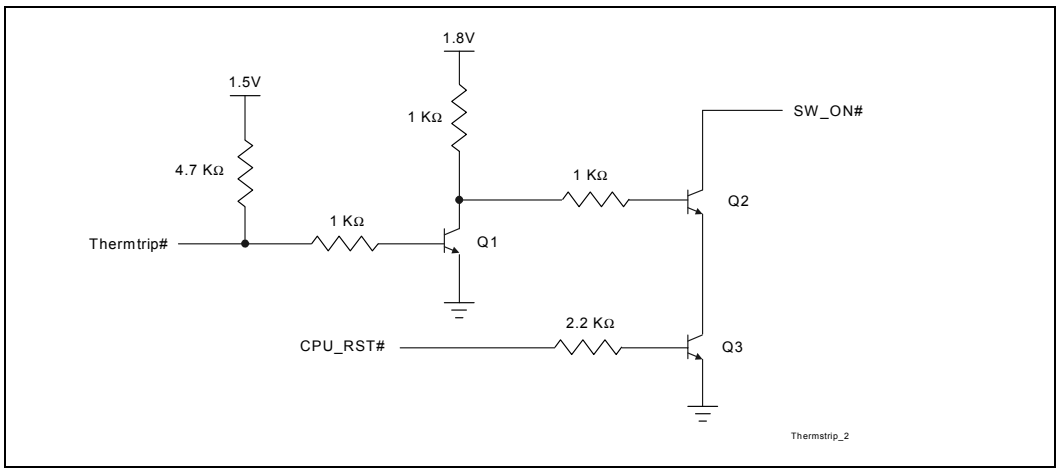
5.3 Electrical Differences for Universal PGA370 Designs

There are several electrical changes between previous PGA370 designs and the *universal PGA370* design, as follows:

- Changes to the PGA370 socket pin definitions.
- Addition of VTTPWRGD signal to ensure stable VID selection for future 0.13 micron socket 370 processors.
- Addition of THERMTRIP circuit to allow processor to detect catastrophic overheat.
- Addition of VID[25mV] signal to support future 0.13 micron socket 370 processors.
- Processor VTT level is switchable to 1.25V or 1.5V, depending on which processor is present in the socket.
- In designs using future 0.13 micron socket 370 processors, the processor does not generate VCMOS_REF.

5.3.1 THERMTRIP Circuit

Figure 21. Example Implementation of THERMTRIP Circuit



NOTES:

1. The pull-up voltage on the collector of Q1 is required to be 1.8V derived from a 3.3V source.
2. THERMTRIP is not valid until after CPU_RST# is deasserted. This is handled by gating the assertion of THERMTRIP with CPU_RST#. Using the CPU_RST# in this manner has minimal impact to the signal quality.
3. THERMTRIP must not go higher than VCC_{CMOS} levels. The pull-up on THERMTRIP is now connected to 1.5V.
4. CPU_RST# must gate SW_ON# from ground. This prevents glitching on SW_ON# during power-up and powerdown.
5. The resistance to the base of the transistor gating CPU_RST# must be at least 2.2 kΩ for proper Vih levels on CPU_RST#.

5.3.1.1 THERMTRIP Timing

When the THERMTRIP signal is asserted, both the VCC and VTT supplies to the processor must be turned off to prevent thermal runaway of the processor. The time required from THERMTRIP asserted to VCC rail at ½ nominal is 500ms and THERMTRIP asserted to VTT rail at ½ nominal is 500ms. System designers must ensure that the decoupling scheme used on these rails does not violate the THERMTRIP timing specifications.

5.3.1.2 Workaround for THERMTRIP on 0.13 Micron Processors with CPUID=6B1h

A platform supporting the future 0.13 micron socket 370 processors must implement a workaround required for the future 0.13 micron socket 370 processor with a CPUID = 06B1h. The internal control register bit responsible for operation of the THERMTRIP circuit functionality may power up in an uninitialized state. As a result, THERMTRIP# may be incorrectly asserted during deassertion of RESET# at nominal operating temperatures. When THERMTRIP# is asserted as a result of this situation, the processor may shut down internally and stop execution. In addition, when the THERMTRIP# pin is asserted, the processor may incorrectly continue to execute, leading to intermittent system power-on boot failures. The occurrence and repeatability of failures is system dependent; however, all systems and processors are susceptible to failure.

To prevent the risk of power-on boot failures, a platform workaround is required. The system must provide a rising edge on the TCK signal during the power-on sequence that meets all of the following requirements:

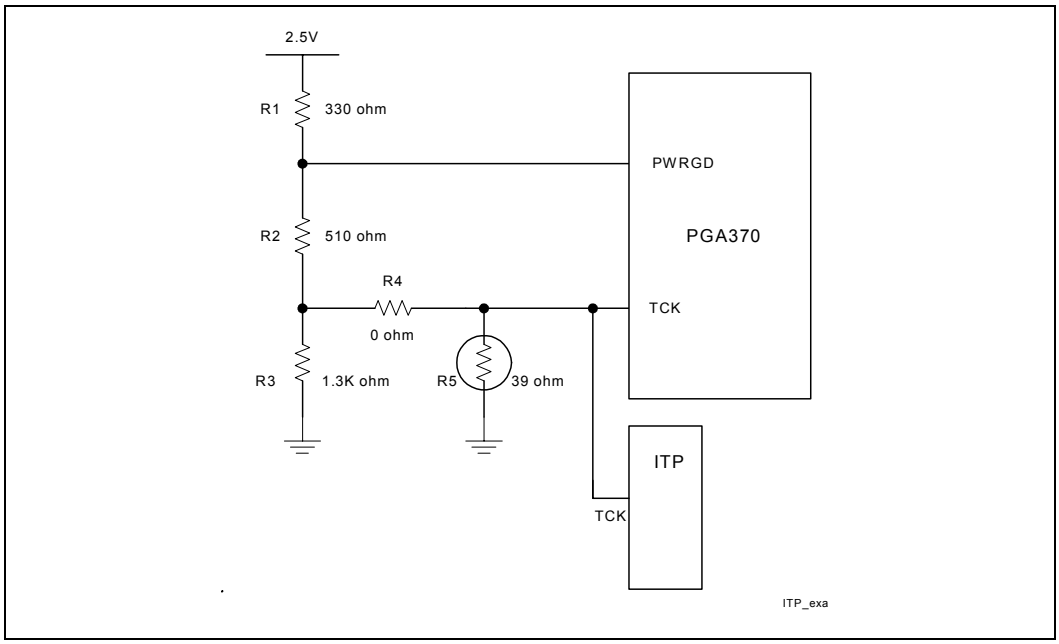
- •Rising edge occurs after VCC_{CORE} is valid and stable
- •Rising edge occurs before or at the deassertion of RESET#
- •Rising edge occurs after all VREF input signals are at valid voltage levels
- •TCK input meets the Vih min (1.3V) and max (1.65V) specification requirements

Specific workaround implementations may be platform specific. The following examples have been tested as acceptable workaround implementations.

The example workaround circuits (see Figure 22) attached require circuit modification for ITP tools to function correctly. These modifications must remove the workaround circuitry from the platform and may cause systems to fail to boot. Review the accompanying notes with each workaround for ITP modification details. If the system fails to boot when using ITP, issuing the ITP 'Reset Target' command on failing systems will reset the system and provide a sufficient rising edge on the TCK pin to ensure proper system boot.

In addition, the example workaround circuits shown do not support production motherboard test methodologies that require the use of the processor JTAG/TAP port. Alternative workaround solutions must be found if such test capability is required.

Figure 22. Example Circuit Showing ITP Workaround



NOTES:
1. For Production Boards: Depopulate R5
2. To use ITP: Install R5, Depopulate R4

5.4 PGA370 Socket Definition Details

Table 10 compares the pin names and functions of the Intel processors supported in the 810E2 chipset universal platform.

Table 10. Processor Pin Definition Comparison

Pin #	Intel® Celeron® Processor (CPUID = 068xh) Pin Name	Intel® Pentium® III Processor (CPUID=068xh) Pin Name	Intel® Pentium® III Processor that uses 0.13 Micron Technology Pin Name	Function
AA33	Reserved	V_{TT}	V_{TT}	• AGTL/AGTL+ termination voltage
AA35	Reserved	V_{TT}	V_{TT}	• AGTL/AGTL+ termination voltage
AB36	VCC_{CMOS}	VCC_{CMOS}	V_{TT}	• CMOS voltage level for Intel® Pentium® III processor (CPUID=068xh) and Intel® Celeron® processor (CPUID=068xh) • AGTL termination voltage for future 0.13 micron socket 370 processors
AD36	VCC1.5	VCC1.5	V_{TT}	• VCC1.5 for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • V_{TT} for future 0.13 micron socket 370 processors
AF36	VSS	VSS	DETECT	• Ground for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • No connect for future 0.13 micron socket 370 processors
AG1[1]	VSS	VSS	V_{TT}	• Ground for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • V_{TT} for future 0.13 micron socket 370 processors
AH4	Reserved	RESET#	RESET#	• Processor reset for the Pentium III processor (068xh) and future 0.13 micron socket 370 processors
AH20	Reserved	V_{TT}	V_{TT}	• AGTL/AGTL+ termination voltage

Pin #	Intel® Celeron® Processor (CPUID = 068xh) Pin Name	Intel® Pentium® III Processor (CPUID=068xh) Pin Name	Intel® Pentium® III Processor that uses 0.13 Micron Technology Pin Name	Function
AJ3[1]	VSS	VSS	RESET2#	• Ground for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • RESET for future 0.13 micron socket 370 processors
AK4	VSS	VSS	VTTPWRGD	• Ground for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • VID control signal on future 0.13 micron socket 370 processors
AK16	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage
AK22	GTL_REF	GTL_REF	VCMOS_REF	• GTL reference voltage for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • CMOS reference voltage future 0.13 micron socket 370 processors
AK36	VSS	VSS	VID[25mV]	• Ground for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • 25mV step VID select bit for future 0.13 micron socket 370 processors
AL13	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage
AL21	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage
AN3	GND	GND	DYN_OE	• Ground for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • Dynamic output enable for future 0.13 micron socket 370 processors
AN11	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage
AN15	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage
AN21	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage

Pin #	Intel® Celeron® Processor (CPUID = 068xh) Pin Name	Intel® Pentium® III Processor (CPUID=068xh) Pin Name	Intel® Pentium® III Processor that uses 0.13 Micron Technology Pin Name	Function
E23	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage
G35	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage
G37	Reserved	Reserved	VTT	• Reserved for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • AGTL termination voltage for future 0.13 micron socket 370 processors
N37	NC	NC	NCHCTRL	• No connect for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • NCHCTRL for future 0.13 micron socket 370 processors
S33	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage
S37	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage
U35	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage
U37	Reserved	VTT	VTT	• AGTL/AGTL+ termination voltage
W3	Reserved	A34#	A34#	• Additional AGTL/AGTL+ address
X4[1]	RESET#	RESET2#	VSS	• Processor reset Pentium III processor (CPUID=068xh) and Celeron processor • Ground for future 0.13 micron socket 370 processors
X6	Reserved	A32#	A32#	• Additional AGTL/AGTL+ address
X34	VCC_{CORE}	VCC_{CORE}	VTT	• Reserved for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • AGTL termination voltage for future 0.13 micron socket 370 processors

Pin #	Intel® Celeron® Processor (CPUID = 068xh) Pin Name	Intel® Pentium® III Processor (CPUID=068xh) Pin Name	Intel® Pentium® III Processor that uses 0.13 Micron Technology Pin Name	Function
Y1	Reserved	Reserved	Reserved	• Reserved for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • No connect for future 0.13 micron socket 370 processors
Y33	Reserved	CLKREF	CLKREF	• 1.25 V PLL reference
Z36	VCC2.5	VCC2.5	Reserved	• VCC2.5 for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) • No connect for future 0.13 micron socket 370 processors

NOTES:
1. Refer to Chapter 4.

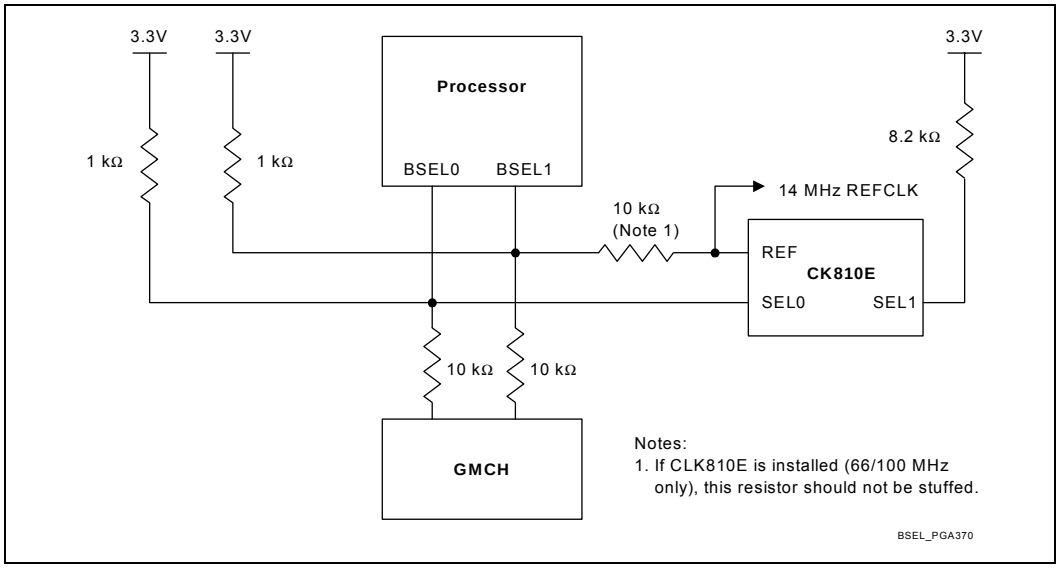
5.5 BSEL[1:0] Implementation Differences

Future 0.13 micron socket 370 processors will select the 133 MHz system bus frequency setting from the clock synthesizer. A Pentium III processor (CPUID=068xh) utilizes the BSEL1 pin to select either the 100 MHz or 133 MHz system bus frequency setting from the clock synthesizer. An Celeron processor (CPUID=068xh) uses both BSEL pins to select 66 MHz system bus frequency from the clock synthesizer. Processors in a FC-PGA or a FC-PGA2 are 3.3 V tolerant for these signals, as are the clock and chipset.

CK810 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. Figure 23 shows the new BSEL[1:0] circuit design for *universal PGA370* designs. Note that BSEL[1:0] now are pulled up using 1 kΩ resistors. Also refer to Figure 24 for more details.

Note: In a design supporting future 0.13 micron socket 370 processors, the BSEL[1:0] lines are not valid until VTTPWRGD is asserted. Refer to Section 4.2.10 for full details.

Figure 23. BSEL[1:0] Circuit Implementation for PGA370 Designs



5.6 CLKREF Circuit Implementation

The CLKREF input, utilized by the Pentium III processor (CPUID=068xh) / Celeron processor (CPUID=068xh) and future 0.13 micron socket 370 processors, require a 1.25V source. It can be generated from a voltage divider on the VCC2.5 or VCC3.3 sources utilizing 1% tolerant resistors. A 4.7 µF decoupling capacitor should be included on this input. See Figure 24 and Table 11 for example CLKREF circuits.

Note: Do not use VTT as the source for this reference!

Figure 24. Examples for CLKREF Divider Circuit

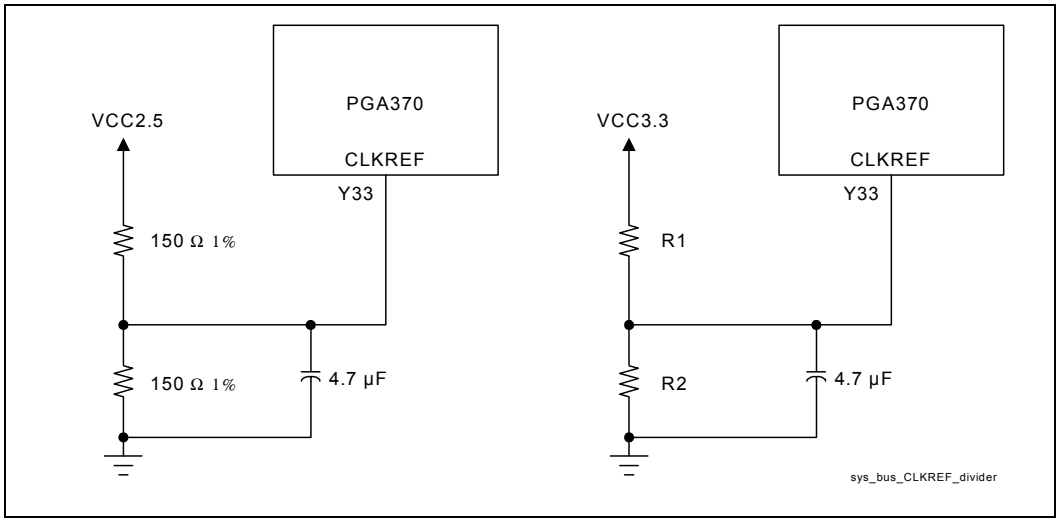


Table 11. Resistor Values for CLKREF Divider (3.3 V Source)

R1 (Ω), 1%	R2 (Ω), 1%	CLKREF Voltage (V)
182	110	1.243
301	182	1.243
374	221	1.226
499	301	1.242

5.7 Undershoot/Overshoot Requirements

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

The Pentium III processor (CPUID=068xh) / Celeron processor (CPUID=068xh) and future 0.13 micron socket 370 processors have 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 or the minimum absolute undershoot voltage limit. Exceeding either of these limits will damage the 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 processor's electrical, mechanical and thermal specification for more details on the processor overshoot/undershoot specifications.

5.8 Processor Reset Requirements

Universal PGA370 designs must route the AGTL/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 the following pins at the PGA370 socket:

- **AH4 (RESET#).** The reset signal is connected to this pin for the Pentium III processor (CPUID=068xh), Celeron processor (CPUID=068xh), and future 0.13 micron socket 370 processors
- **X4 (Reset2# or GND, depending on processor).** The X4 pin is RESET2# for Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh). X4 is GND for future 0.13 micron socket 370 processors. An additional 1kΩ resistor is connected in series with pin X4 to the reset circuitry since pin X4 is a ground pin in future 0.13 micron socket 370 processors.

Note: The AGTL/AGTL+ reset signal must always terminate 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 (CPUID=068xh). Pin X4 should then be connected to ground.

The routing rules for the AGTL/AGTL+ reset signal are shown in Figure 25.

Figure 25. RESET#/RESET2# Routing Guidelines

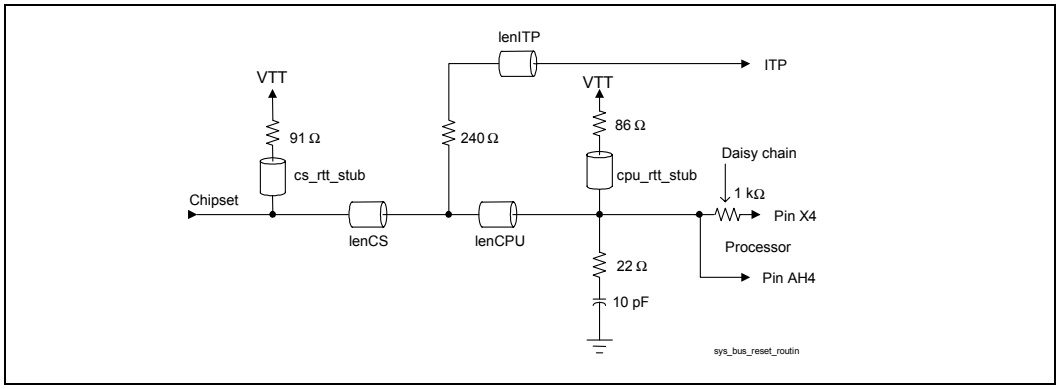


Table 12. RESET#/RESET2# Routing Guidelines (see Figure 25)

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

5.9 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.

5.9.1 Topology

The general desired topology for these PLLs is shown in Figure 27. Not shown are the parasitic routing and local decoupling capacitors. Excluded from the external circuitry are parasitics associated with each component.

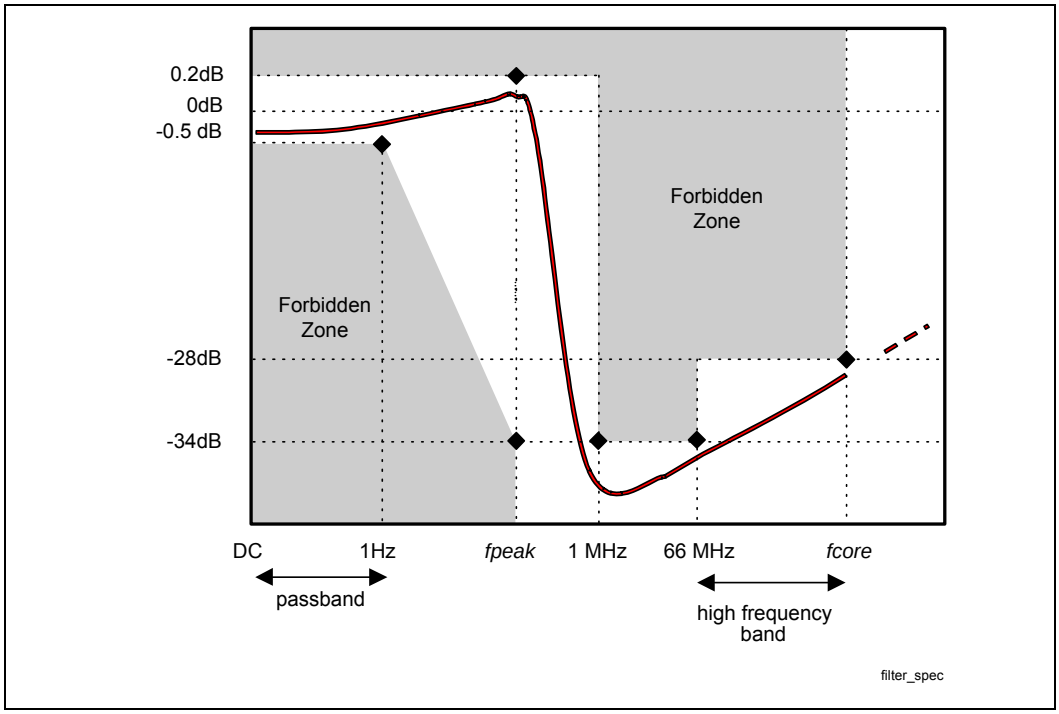
5.9.2 Filter Specification

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 Figure 26.

Figure 26. 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.

5.9.3 Recommendation for Intel® Platforms

The following tables contain examples of components that meet Intel's recommendations, when configured in the topology of Figure 27.

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 Recommendation – 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. See Figure 28.

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 chosen 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. These routes do not count towards the minimum damping R requirement.
- 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.

Figure 27. Example PLL Filter Using a Discrete Resistor

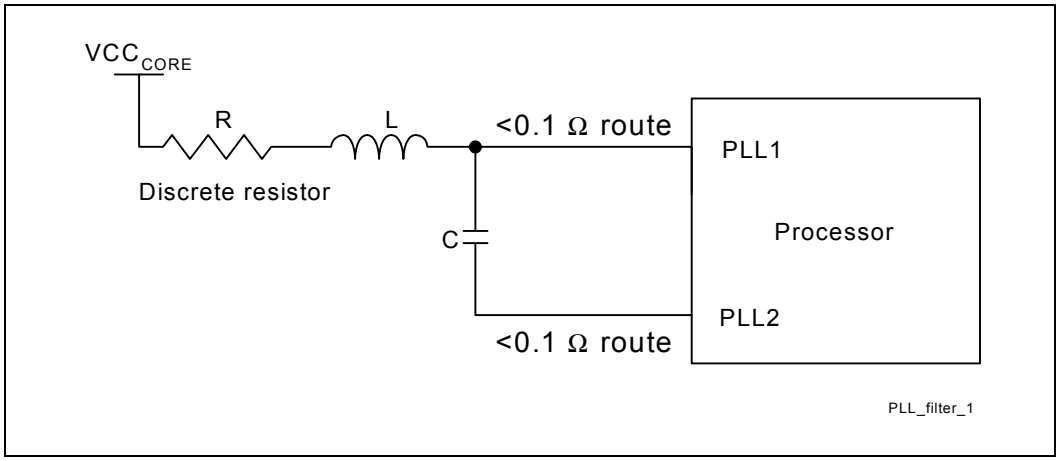
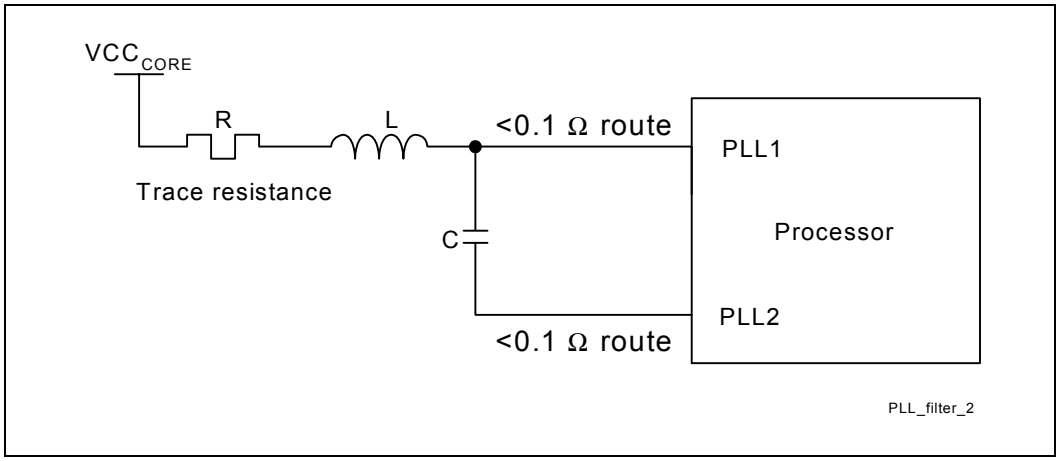


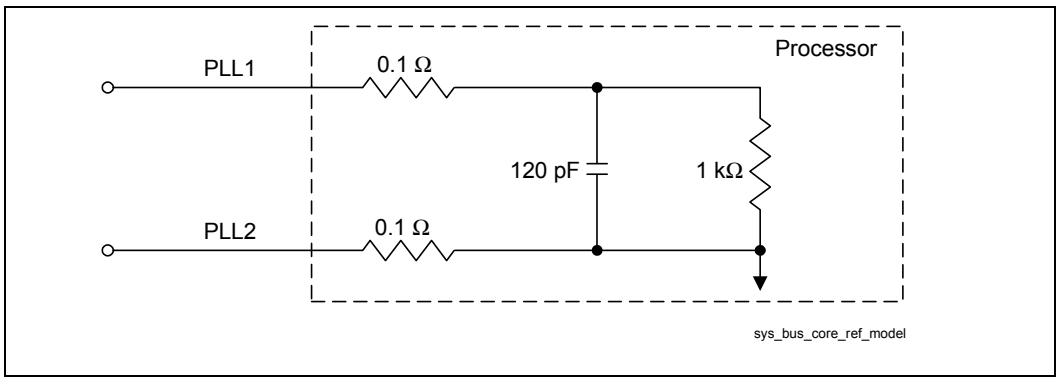
Figure 28. Example PLL Filter Using a Buried Resistor



5.9.4 Custom Solutions

As long as designers satisfy filter performance and requirements as specified and outlined in Section 5.9.2, other solutions are acceptable. Custom solutions should be simulated against a standard reference core model, which is shown in Figure 29.

Figure 29. Core Reference Model



NOTES:
1. 0.1 Ω resistors represent package routing.
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.

5.10 Voltage Regulation Guidelines

A *universal PGA370* design will need the voltage regulation module (VRM) or on-board voltage regulator (VR) to be compliant with Intel *VRM 8.5 DC-DC Converter Design Guidelines*.

5.11 Decoupling Guidelines for Universal PGA370 Designs

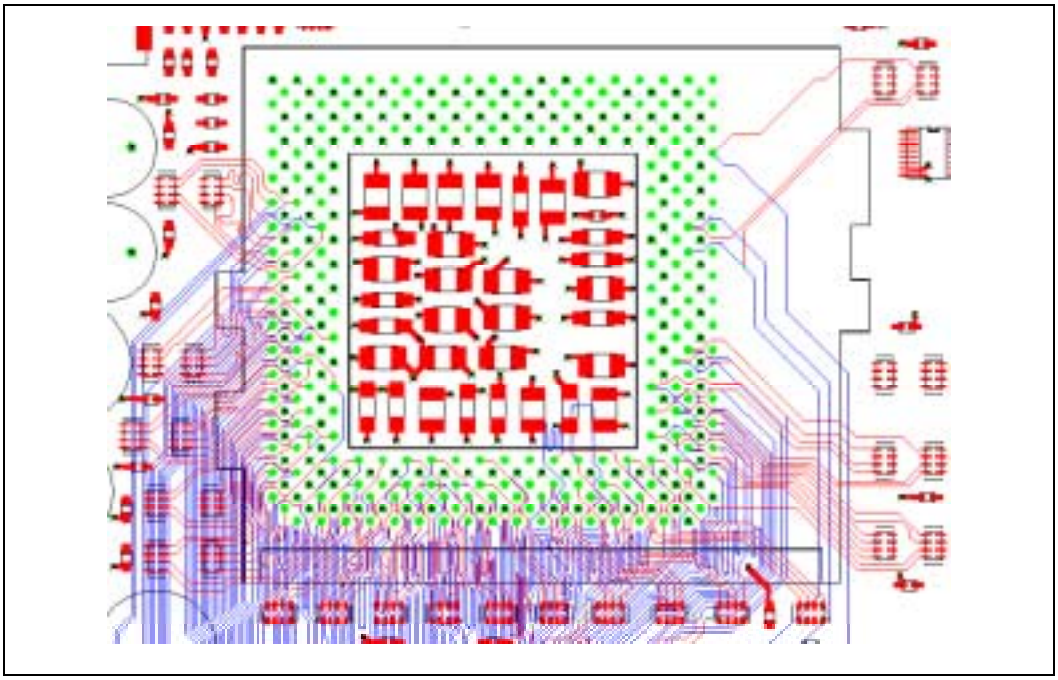
These preliminary decoupling guidelines for *universal PGA370* designs are estimated to meet the specifications of *VRM 8.5 DC-DC Converter Design Guidelines*.

5.11.1 VCC_{CORE} Decoupling Design

- Sixteen 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 Figure 30.

Figure 30. Capacitor Placement on the Motherboard



5.11.2 VTT Decoupling Design

For Itt = 2.3 A (max.)

- 20 0.1 µF capacitors in 0603 packages placed as closed as possible to the processor VTT pins. The capacitors are shown on the exterior of Figure 30.

5.11.3 V_{REF} Decoupling Design

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

5.12 Thermal Considerations

5.12.1 Heatsink Volumetric Keep-Out Regions

Current heatsink recommendations are only valid for supported Celeron and Pentium III processor frequencies.

Figure 31 shows the system component keep-out volume above the socket connector required for the reference design thermal solution for high frequency 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. The heatsink must be compatible with the Integrated Heat Spreader (IHS) used by higher frequency Pentium III processors.

Figure 32 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 high frequency processors. While the keep-out requirements should provide adequate space for the reference design thermal solution, systems integrators should check with their vendors to ensure their specific thermal solutions fit within their specific system designs. 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 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 31. Heatsink Volumetric Keep-Out Regions

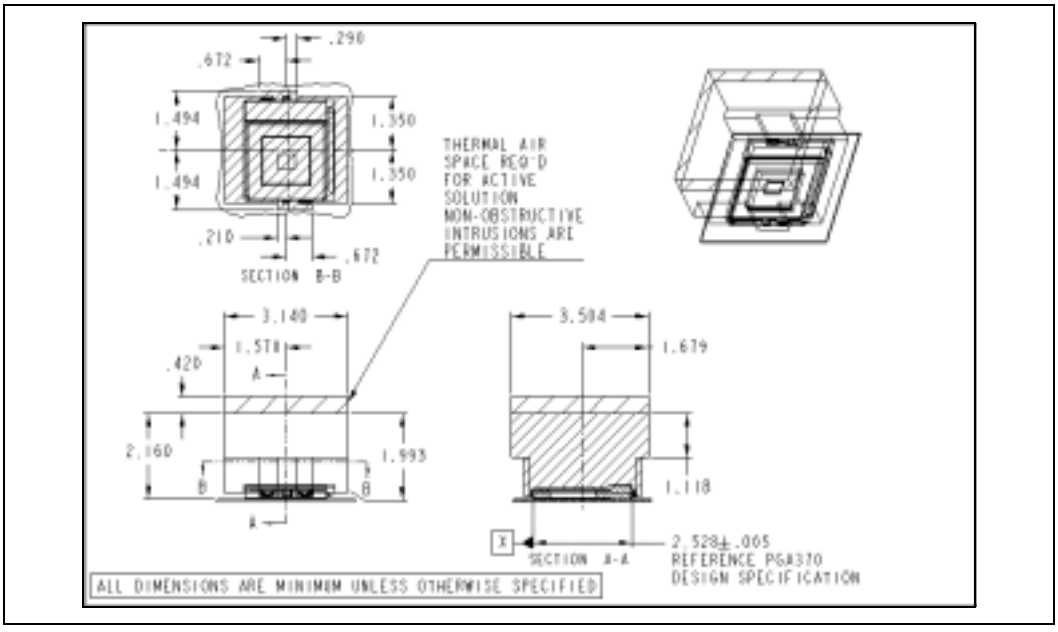
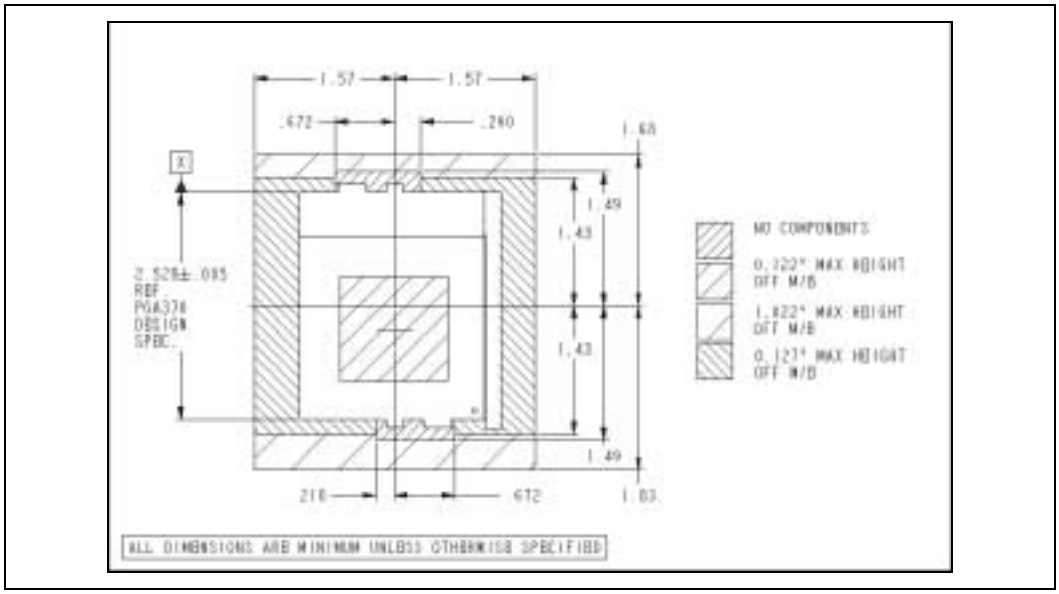


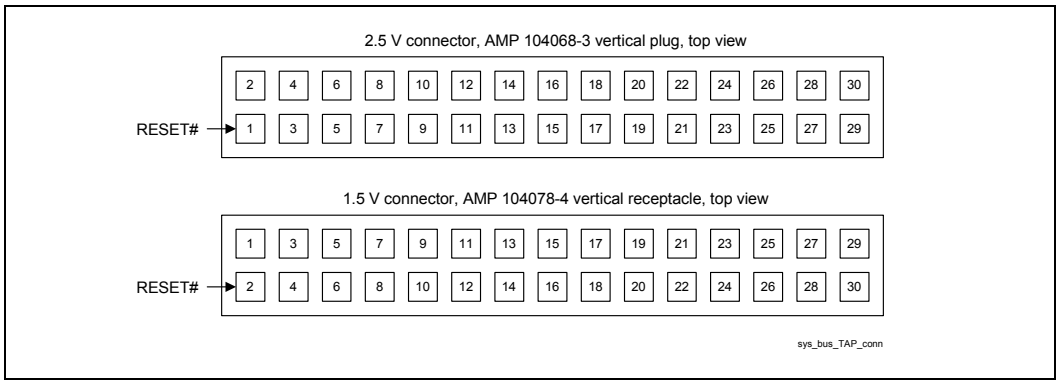
Figure 32. Motherboard Component Keep-Out Regions



5.13 Debug Port Changes

Due to the lower voltage technology employed with newer processors, 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. Pentium III processor (CPUID=068xh) / Celeron processor (CPUID=068xh) and future 0.13 micron socket 370 processors utilize 1.5 V logic levels on the TAP. As a result, the type of debug port connecter used in *universal PGA370* designs is dependent on the processor that is currently in the socket. The 1.5 V connector is a 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 (see Figure 33). Also required, along with the new connector, is an In-Target Probe* (ITP) that is capable of communicating with the TAP at the appropriate logic levels.

Figure 33. TAP Connector Comparison



Caution: Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) 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 any of these specified processors.

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

6 Layout and Routing Guidelines

This chapter describes motherboard layout and routing guidelines for Intel 810ET2 chipset systems, except for the processor layout guidelines. For the PGA370 processor-specific layout guidelines (Refer to Chapter 4). This chapter does not discuss the functional aspects of any bus or the layout guidelines for an add-in device.

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

6.1 General Recommendations

The trace impedance typically noted (i.e., 60 Ω ±15%) is the “nominal” trace impedance for a 5 mil wide trace (i.e., 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. Using wider spaces between the traces can minimize this trace-to-trace coupling. In addition, these wider spaces reduce settling time.

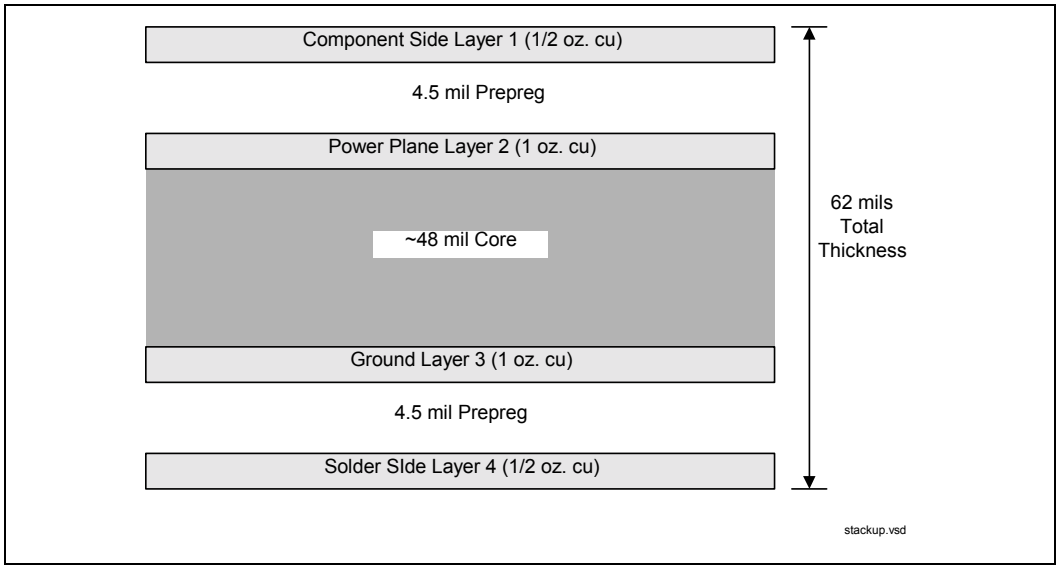
Coupling between two traces is a function of the coupled length, the distance separating the traces, the signal edge rate, and the degree of mutual capacitance and inductance. To minimize the effects of trace-to-trace coupling, the routing guidelines documented in this chapter should be followed.

Additionally, these routing guidelines are created using the *stack-up* (refer to Figure 34). If this stack-up is not used, simulations should be completed.

6.2 Nominal Board Stack-Up

The Intel 810E2 chipset for use with universal socket 370 platform requires a board stack-up yielding a target impedance of 60 Ω ±15% with a 5 mil nominal trace width. Figure 34 presents an example stack-up to achieve this. It is a 4-layer fabrication construction using 53% resin, FR4 material.

Figure 34. Nominal Board Stack-Up



6.3 System Memory Layout Guidelines

6.3.1 System Memory Solution Space

Figure 35. System Memory Topologies

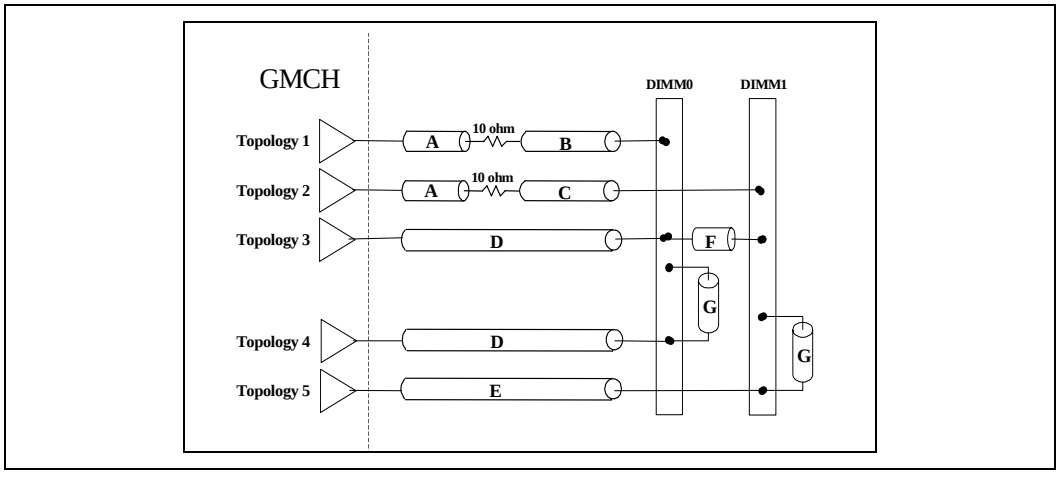


Table 16. System Memory Routing

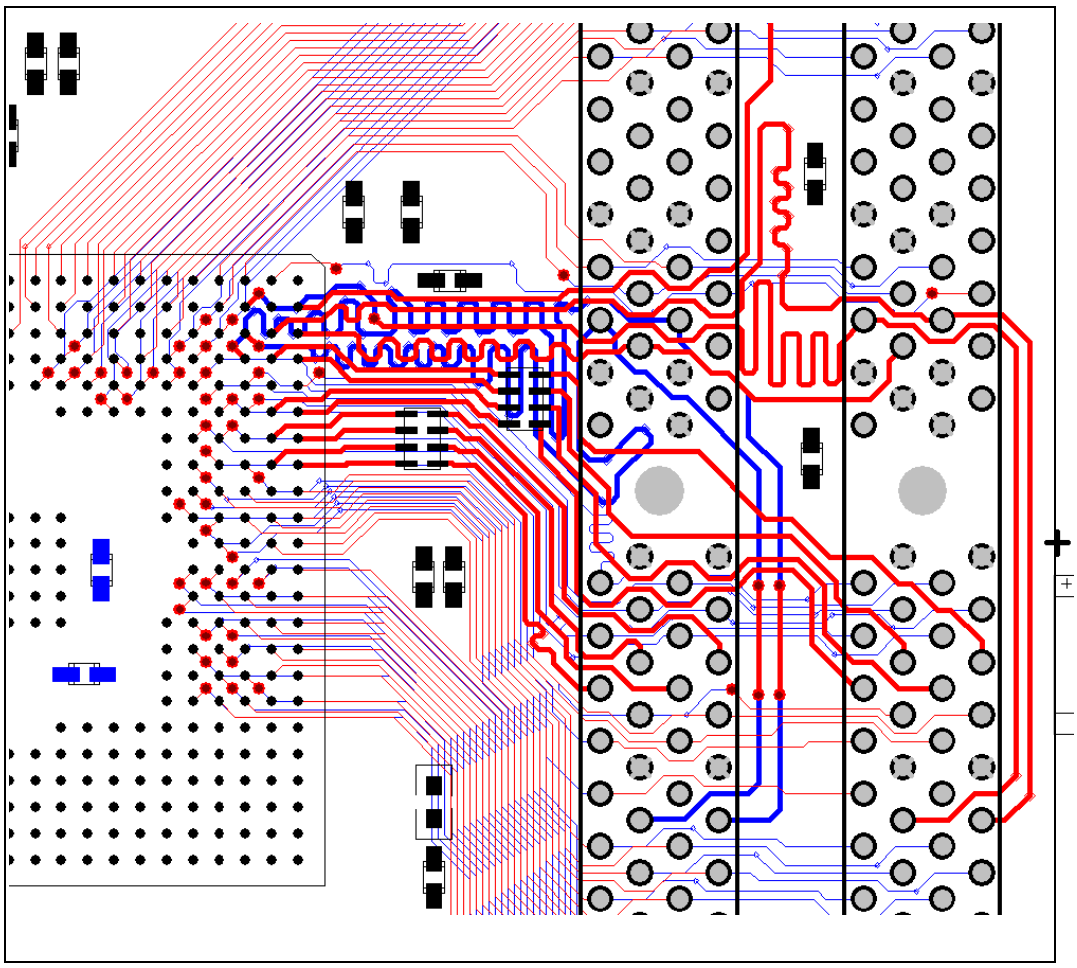
Signal		Top.	Trace (mils)		Trace Lengths (inches)												
					A	B		C		D		E		F		G	
			Width	Space	Max	Min	Max	Min	Max	Min	Max	Min	Max	Min	Max	Min	Max
SCS[3:2]#	Opt.1	5	10	8								3	5			1.5	2
	Opt.2	5	10	8								2.2	5			1.5	1.8
	Opt.3	5	10	8								1.6	5			1.15	1.5
SCS[1:0]#	Opt.1	4	10	8						3	5					1.5	2
	Opt.2	4	10	8						2.2	5					1.5	1.8
	Opt.3	4	10	8						1.6	5					1.15	1.5
SMAA[7:4]		1	10	8	0.5	0.5	2										
SMAB[7:4]#		2	10	8	0.5			0.5	2								
SCKE[1:0]		3	10	8						1	2.5			0.4	1		
SMD[63:0], SDQM[7:0]		3	5	7						1	3			0.4	1		
SCAS#, SRAS#, SWE#		3	5	7						1	3.5			0.4	1		
SBS[1:0], SMAA[11:8, 3:0]		3	5	7						1	2.5			0.4	1		

NOTES:

1. It is recommended to add 10 Ω series resistors to the MAA[7:4] and the MAB[7:4] lines, as close as possible to GMCH for signal integrity.

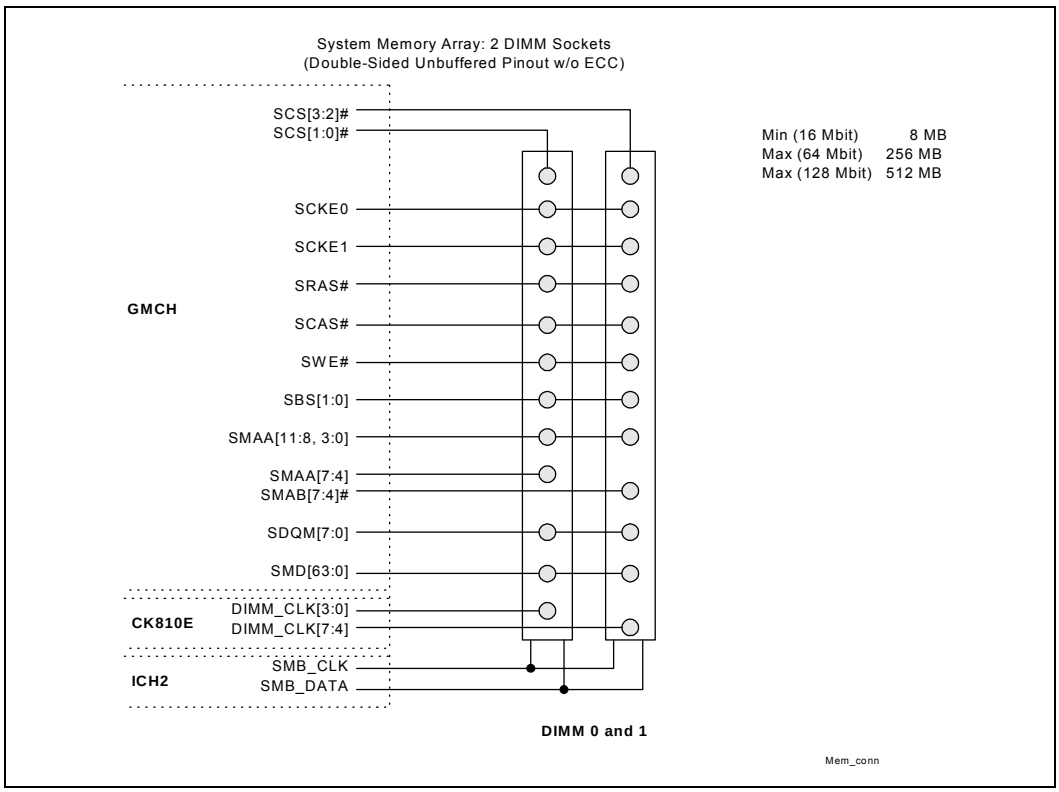
6.3.2 System Memory Routing Example

Figure 36. System Memory Routing Example



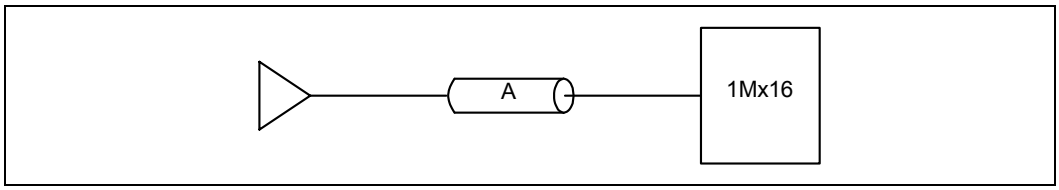
6.3.3 System Memory Connectivity

Figure 37. System Memory Connectivity



6.4 Display Cache Interface

Figure 38. Display Cache (Topology 1)



6.4.1 Display Cache Solution Space

Table 17. Display Cache Routing (Topology 1)

Signal	Topology	Trace (mils)		A (inches)	
		Width	Spacing	Min	Max
LMD[31:0], LDQM[3:0]	1	5	7	1	5

NOTE: Trace Length (inches)

Figure 39. Display Cache (Topology 2)

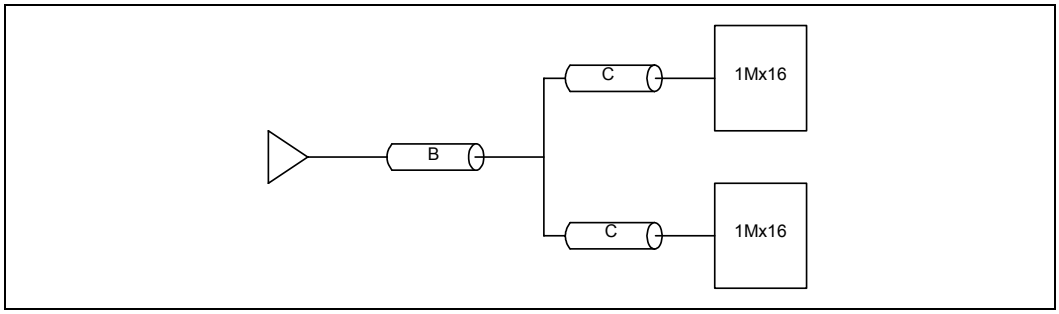


Table 18. Display Cache Routing (Topology 2)

Signal	Topology	Trace (units=mils)		B (inches)		C (inches)	
		Width	Spacing	Min	Max	Min	Max
LMA[11:0], LWE#, LCS#, LRAS#, LCAS#	2	5	7	1	3.75	0.75	1.25

Figure 40. Display Cache (Topology 3)

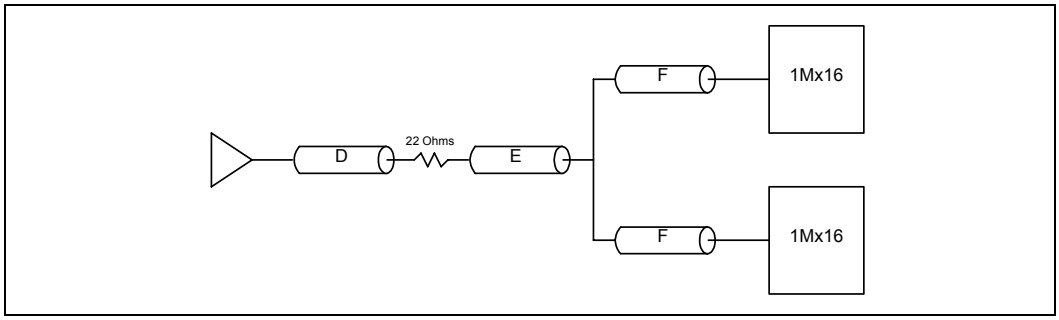


Table 19. Display Cache Routing (Topology 3)

Signal	Topology	Trace (units=mils)		D (inches)	E (inches)		F (inches)	
		Width	Spacing	Length	Min	Max	Min	Max
TCLK	3	5	7	0.5	1.5	2.5	0.75	1.25

Figure 41. Display Cache (Topology 4)

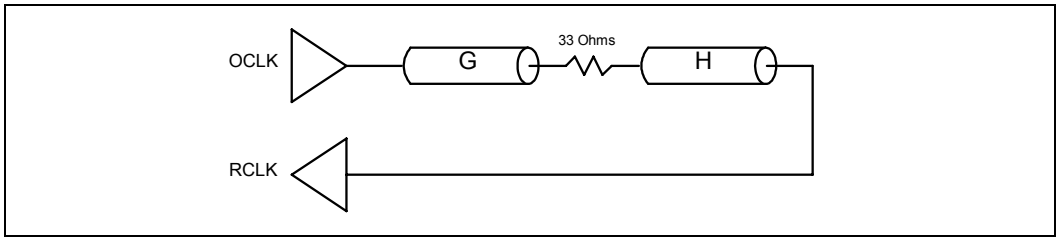


Table 20. Display Cache Routing (Topology 4)

Signal	Topology	Trace (units=mils)		G (inches)	H (inches)	
		Width	Spacing	Length	Min	Max
OCLK	4	5	6	0.5	3.25	3.75

6.5 Hub Interface

The 810ET2 chipset's GMCH 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 GMCH to the ICH2 on the top signal layer. Refer to Figure 42.

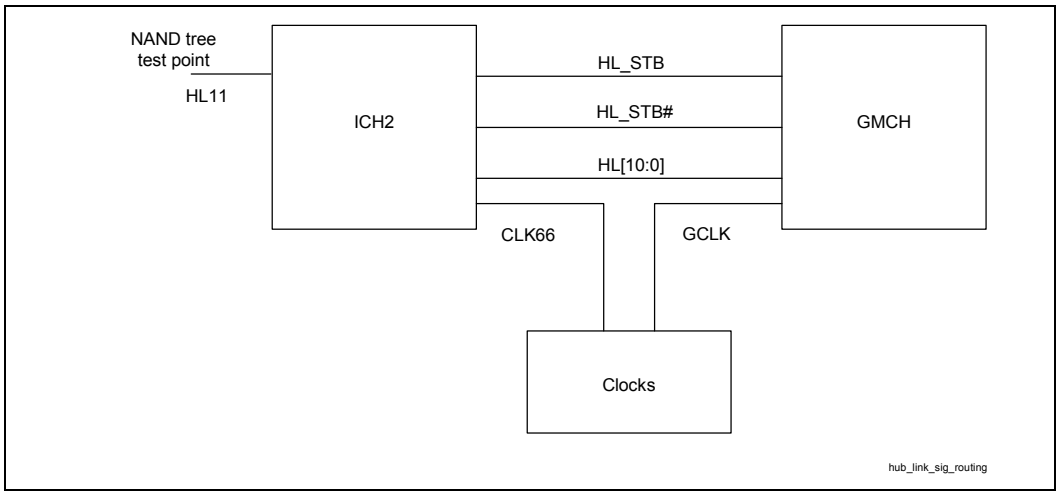
The hub interface is divided into two signal groups: data signals and strobe signals.

- 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 42. Hub Interface Signal Routing Example



6.5.1 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 GMCH 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 inch of the GMCH/ICH2 components.

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

6.5.2 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 inches, 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 inch.

6.5.3 HREF Generation/Distribution

HREF, the hub interface reference voltage, is 0.5 * 1.8 V = 0.9 V ± 2%. It can be generated using a single HREF divider or locally generated dividers (as shown in Figure 43 and Figure 44). The resistors should be equal in value and rated at 1% tolerance, to maintain 2% tolerance on 0.9 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 inches away from either GMCH or ICH2. If the single HREF divider is located more than 4 inches 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.

6.5.4 Compensation

Independent Hub interface compensation resistors are used by the 810ET2 chipset's GMCH and the ICH2 to adjust buffer characteristics to specific board characteristics. Refer to the *Intel® 810ET2 Chipset Family: 82810 Graphics and Memory Controller Hub (GMCH) Datasheet* and the *Intel® 82801BA I/O Controller Hub 2 (ICH2) and Intel® 82801 BAM I/O Controller Hub 2 Mobile (ICH2-M) 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 inch trace (targeted at a nominal trace impedance of 40 Ω). The GMCH and ICH2 each require their own RCOMP resistor.

Figure 43. Single Hub Interface Reference Divider Circuit

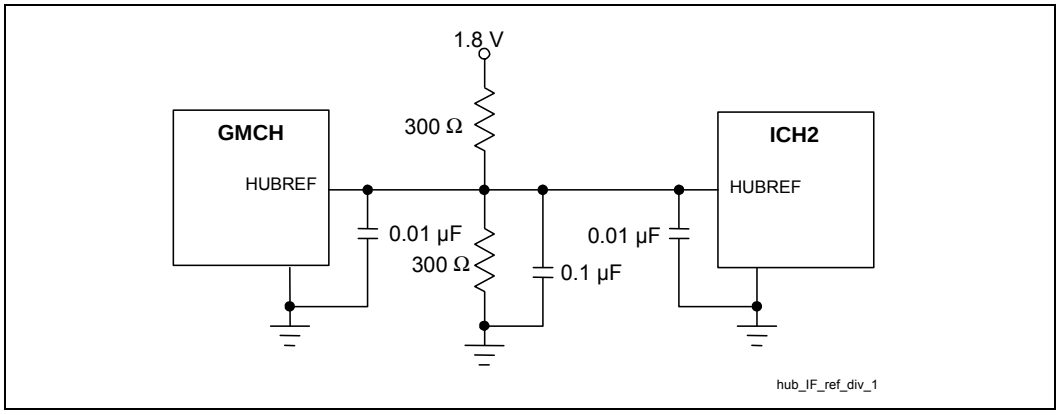
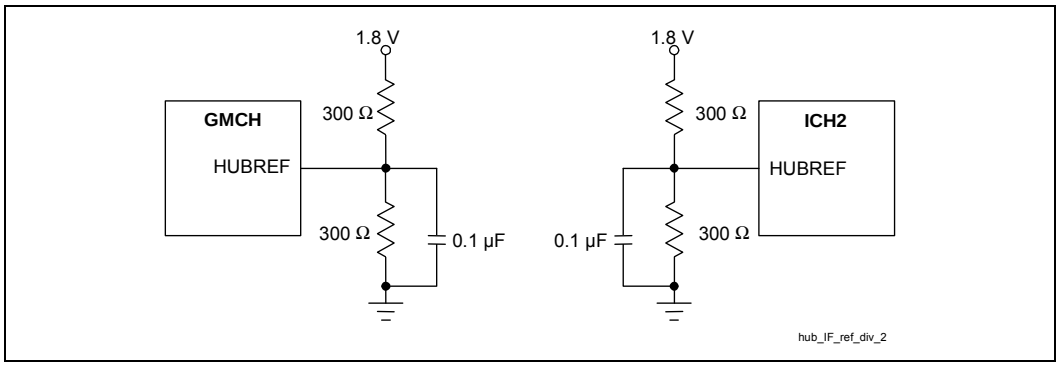


Figure 44. Locally Generated Hub Interface Reference Dividers



6.6 Intel® ICH2

6.6.1 Decoupling

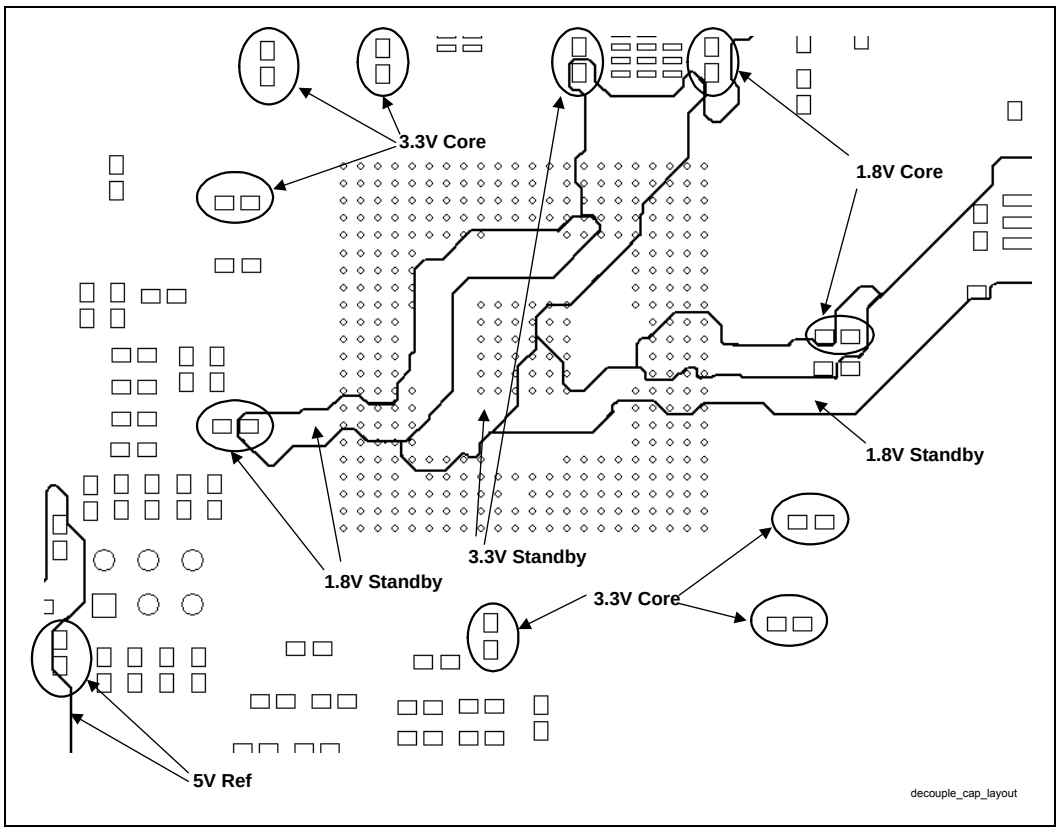
The ICH2 is capable of generating large current swings when switching between logic High and logic Low. This condition could cause the component voltage rails to drop below specified limits. To avoid this type of situation, ensure that the appropriate amount of bulk capacitance is added in parallel to the voltage input pins. It is recommended that the developer use the amount of decoupling capacitors specified in Table 21 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 Figure 45 for a layout example. It is recommended that, for prototype board designs, the designer include pads for extra power plane decoupling caps.

Table 21. Decoupling Capacitor Recommendation

Power Plane/Pins	Decoupling Capacitors	Capacitor Value
3.3 V core	6	0.1 µF
3.3 V standby	1	0.1 µF
Processor interface (1.3 ~ 2.5 V)	1	0.1 µF
1.8 V core	2	0.1 µF
1.8 V standby	1	0.1 µF
5 V reference	1	0.1 µF
5 V reference standby	1	0.1 µF

Figure 45. Intel® ICH2 Decoupling Capacitor Layout



6.7 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).

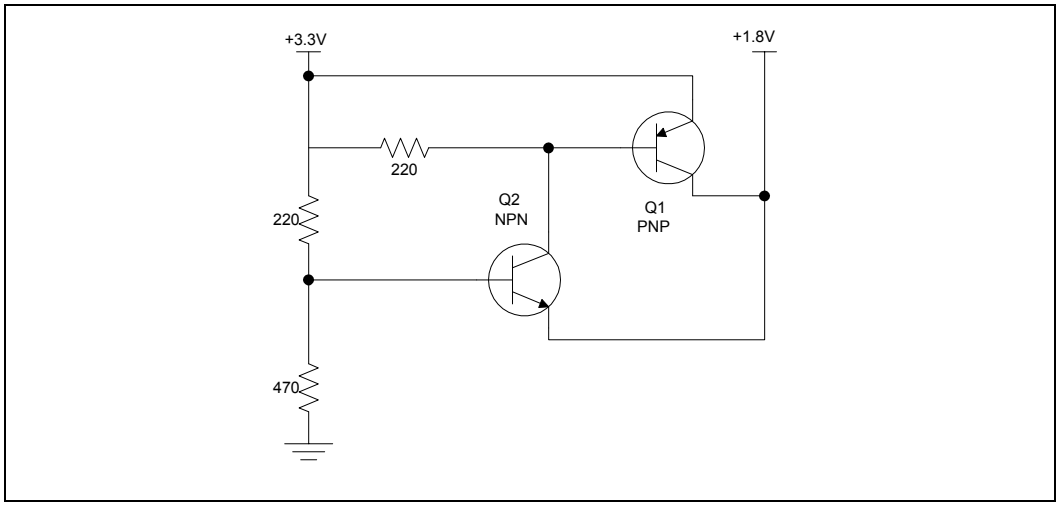
One serious consequence of violation of this "2V Rule" is electrical overstress of oxide layers, resulting in component damage.

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.

Figure 46 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 46. Example 1.8V/3.3V Power Sequencing Circuit



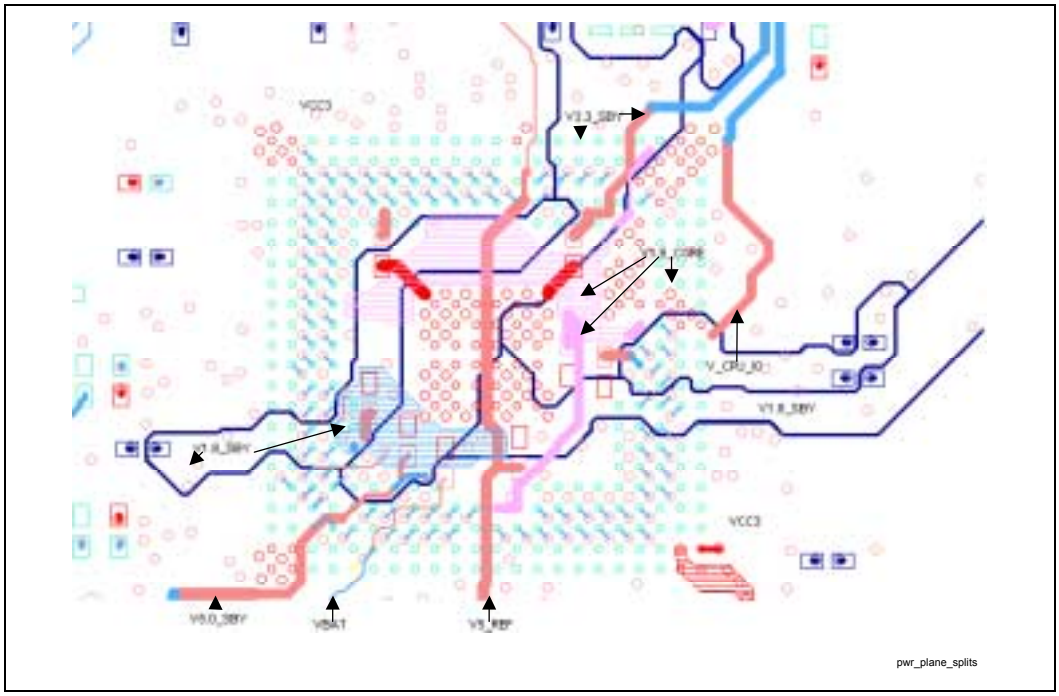
When analyzing systems that may be "marginally compliant" to the 2V Rule, pay close attention to the behavior of the ICH2's RSMRST# and PWROK signals, since these signals control internal isolation logic between the various power planes:

- 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.

6.8 Power Plane Splits

Figure 47. Power Plane Split Example



6.9 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%.

6.10 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 inches long (from ICH2 to IDE connector). Additionally, the shortest IDE signal (on a given IDE channel) must be less than 0.5 inch shorter than the longest IDE signal (on that channel).

6.10.1 Cabling

- **Length of cable:** Each IDE cable must be equal to or less than 18 inches.
- **Capacitance:** Less than 30 pF
- **Placement:** A maximum of 6 inches 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 inches 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 inches 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.

6.11 Cable Detection for Ultra ATA/66 and Ultra ATA/100

The ICH2 IDE controller supports PIO, multiword (Intel 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.

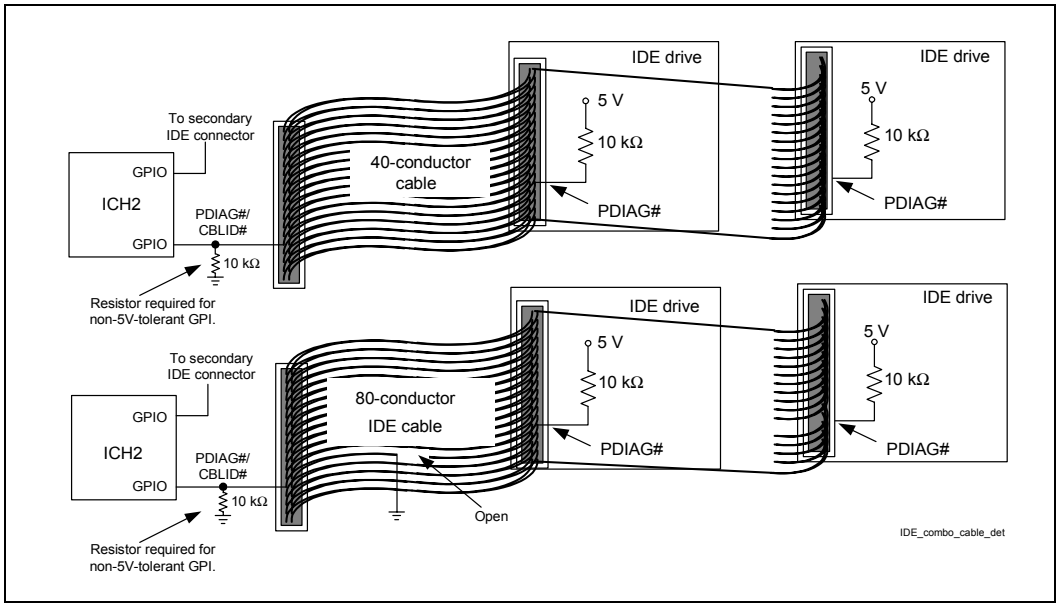
To determine whether the ATA/66 or ATA/100 mode can be enabled, the ICH2 requires that the system software attempt to determine the type of cable used in the system. If the system software detects an 80-conductor cable, it may use any Ultra DMA mode up to the highest transfer mode supported by both the 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.

6.11.1 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 Figure 48. All IDE devices have a 10 kΩ pull-up resistor to 5 volts on this signal. Not all GPI and GPIO pins on the ICH2 are 5-volt tolerant. If non 5-volt 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 Figure 48).

Figure 48. Combination Host-Side / Device-Side IDE Cable Detection



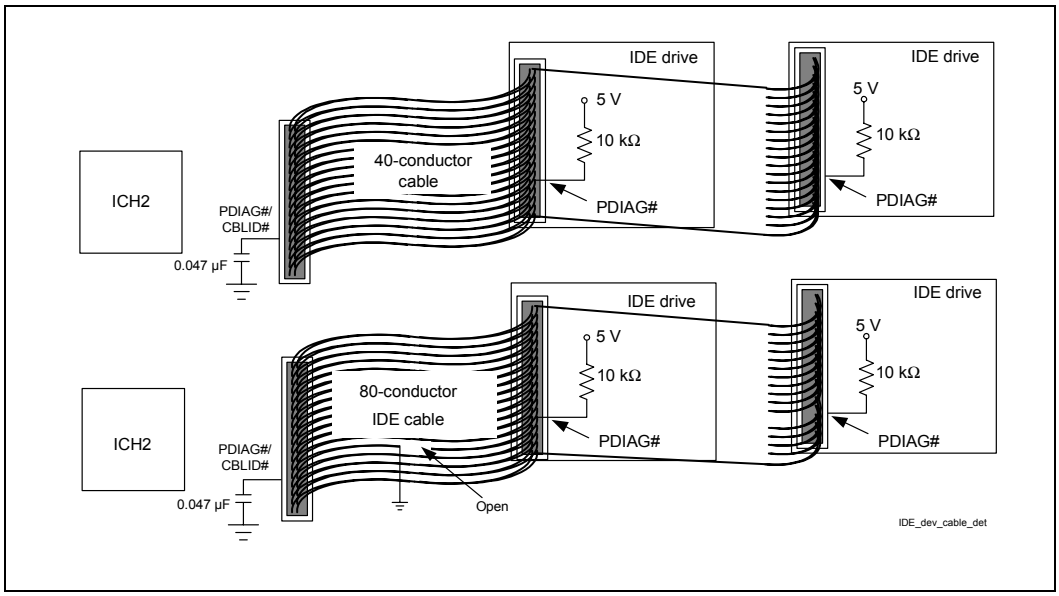
This mechanism allows the BIOS, after diagnostics, to sample PDIAG#/CBLID#. If the signal is High, then there is a 40-conductor cable in the system and ATA modes 3, 4 and 5 must not be enabled.

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.

6.11.2 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 Figure 49. This capacitor **should not be populated** when implementing the recommended combination host-side/device-side cable detection mechanism described previously.

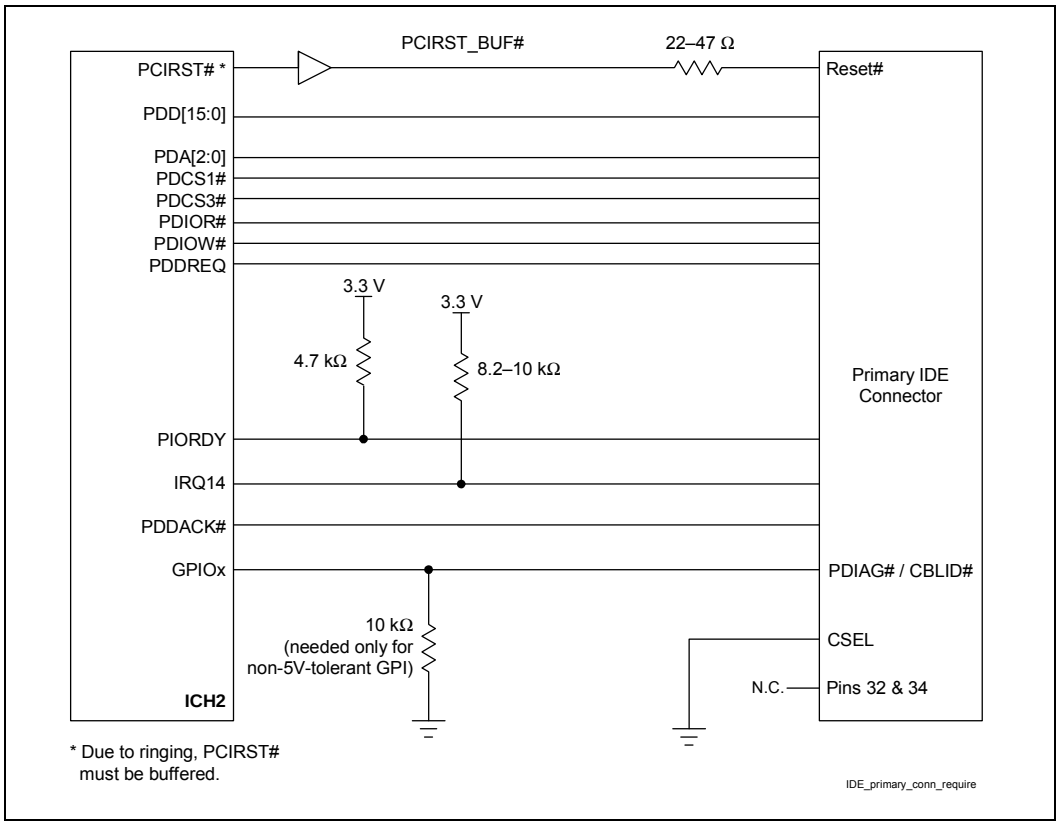
Figure 49. Device-Side IDE Cable Detection



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.

6.11.3 Primary IDE Connector Requirements

Figure 50. Connection Requirements for Primary IDE Connector

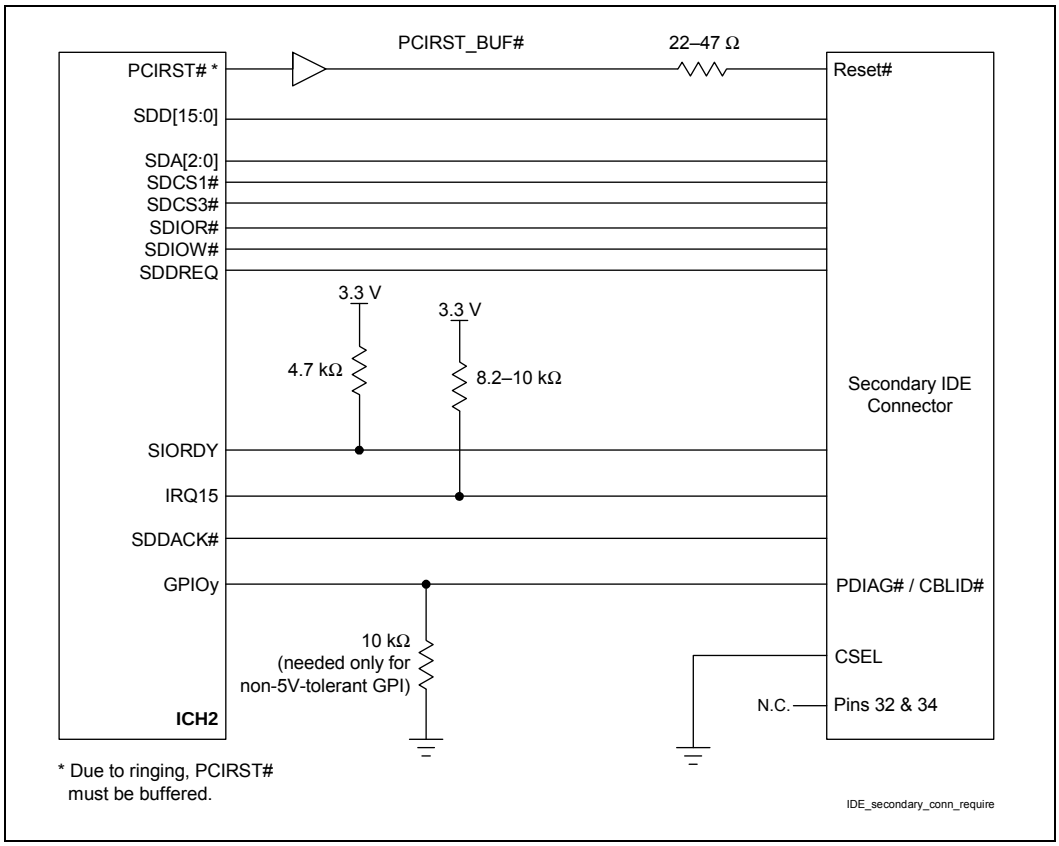


NOTES:

1. 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.
2. An 8.2 kΩ to 10 kΩ pull-up resistor is required on IRQ14 and IRQ15 to VCC3.
3. A 4.7 kΩ pull-up resistor to VCC3 is required on PIORDY and SIORDY.
4. 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.
5. 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.

6.11.4 Secondary IDE Connector Requirements

Figure 51. Connection Requirements for Secondary IDE Connector



NOTES:

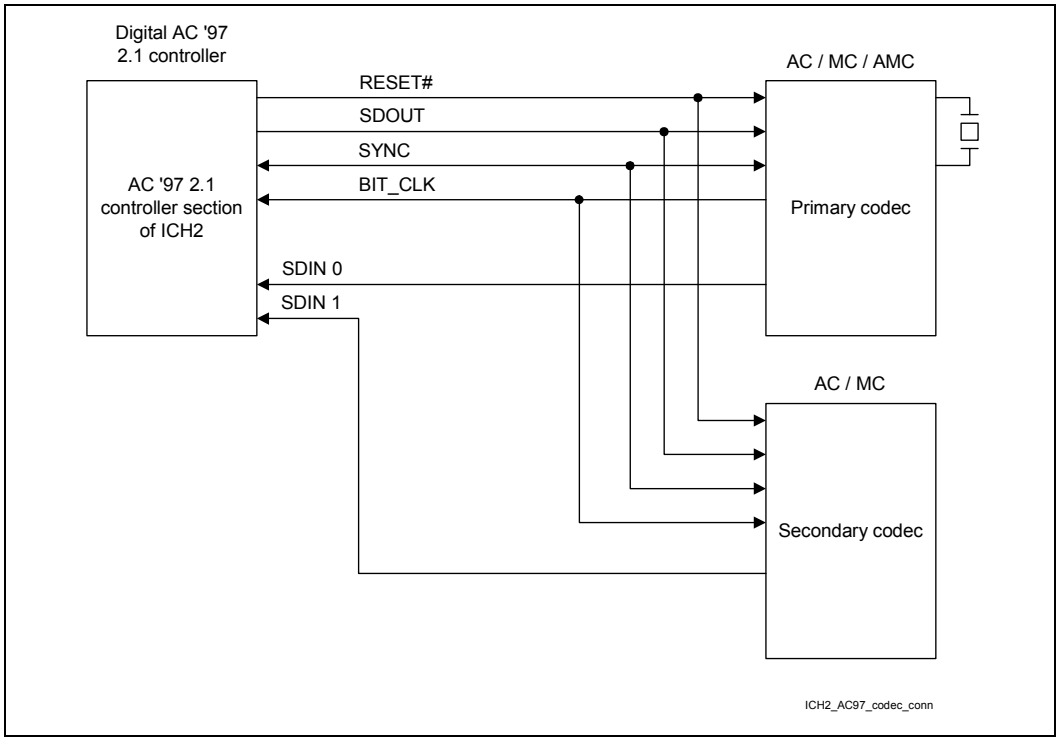
1. 22 Ω to 47 Ω series resistors are required on RESET#. The correct value should be determined for each unique motherboard design, based on signal quality.
2. An 8.2 kΩ to 10 kΩ pull-up resistor is required on IRQ14 and IRQ15 to VCC3.
3. A 4.7 kΩ pull-up resistor to VCC3 is required on PIORDY and SIORDY
4. 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.
5. The 10 kΩ resistor to ground on the PDIAG#/CBLID# signal is now required on the Secondary Connector. This change is to prevent the GPI pin from floating if a device is not present on the IDE interface.

6.12 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 bi-directional, 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. Figure 52 shows a two-codec topology of the AC-link for the ICH2.

Figure 52. 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 6.13.

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'97 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.

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 22. 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[1].

NOTES:
1. 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.

6.12.1 AC'97 Audio Codec Detect Circuit and Configuration Options

The following provides general circuits to implement a number of different codec configurations. 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.

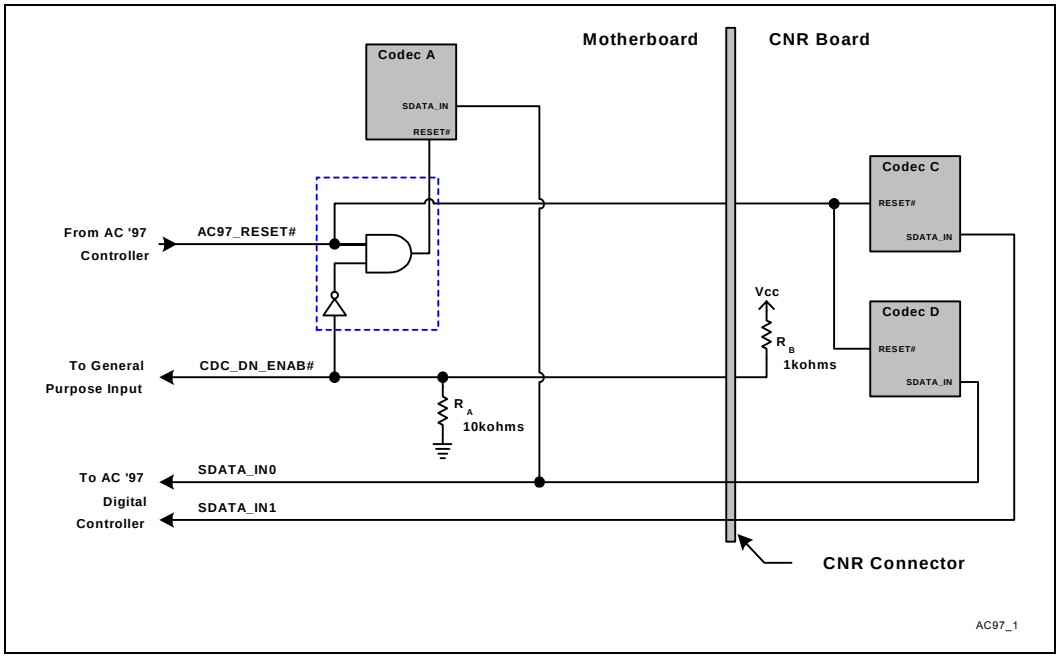
The following four circuit figures (Figure 53 to Figure 56)show the adaptability of a system with the modification of R_A and R_B combined with some basic glue logic to support multiple codec

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 BIOS so that the correct PnP IDs can be loaded.

As shown in Figure 53, 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 to disable the motherboard codec when a CNR is installed containing two AC ’97 codecs (or a single AC ’97 codec that 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.

Figure 53. CDC_DN_ENAB# Support Circuitry for a Single Codec on Motherboard



The architecture shown in Figure 54 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 54 and Figure 55 show a switch on the CNR board. This is necessary to connect the CNR board codec to the proper SDATA_IN*n* line so there is not a conflict with the motherboard codec(s).

Figure 54. CDC_DN_ENAB# Support Circuitry for Multi-Channel Audio Upgrade

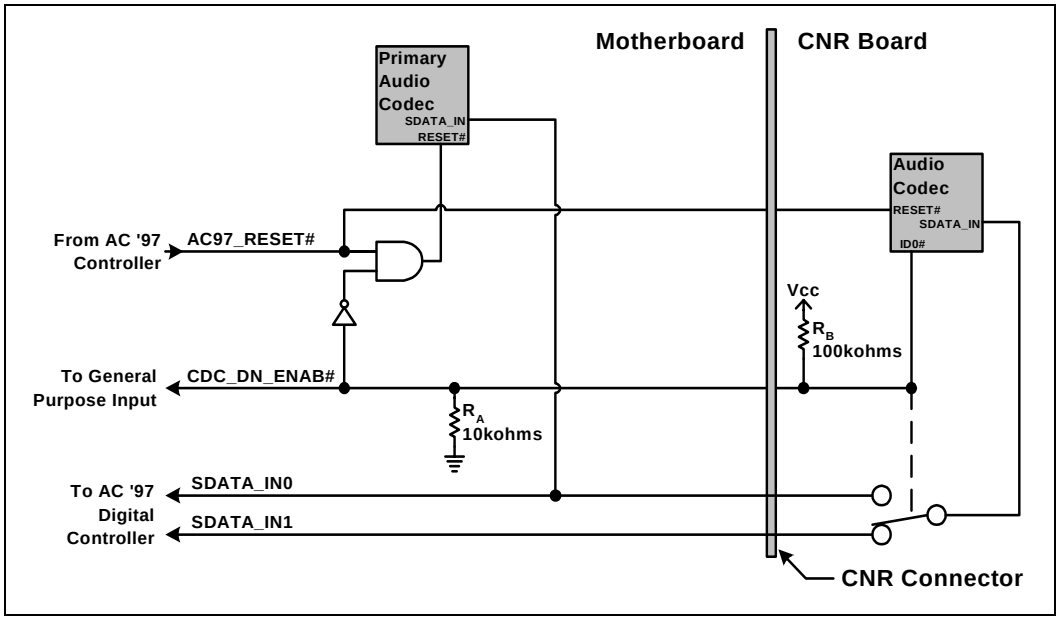


Figure 55 shows the circuitry required on the motherboard to support a two-codec down configuration. This circuitry disables the codec on a single codec CNR. Notice that in this configuration the resistor R_B has been changed to 100 kΩ.

Figure 55. CDC_DN_ENAB# Support Circuitry for Two-Codecs on Motherboard / One-Codec on CNR

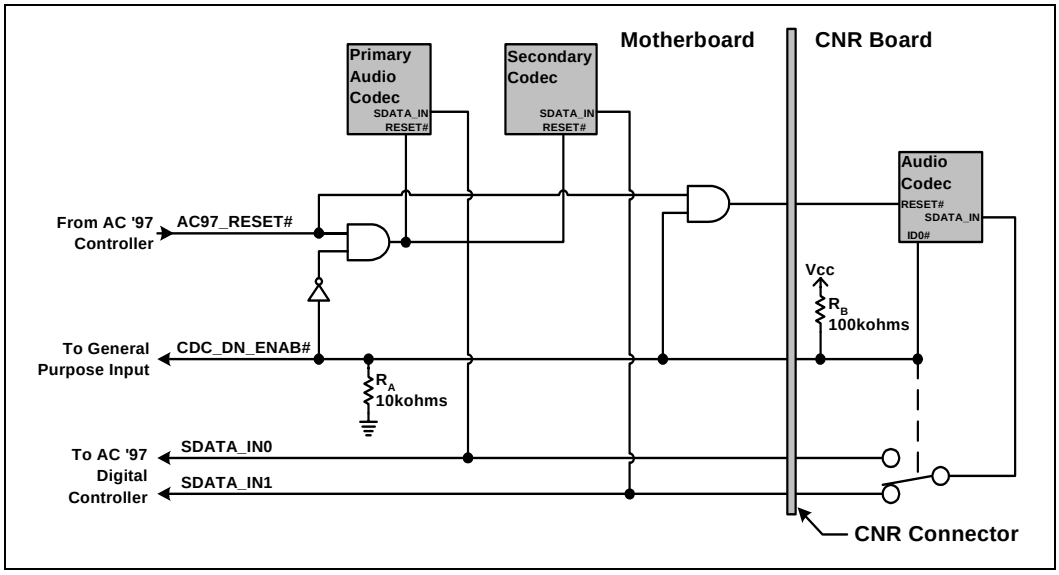
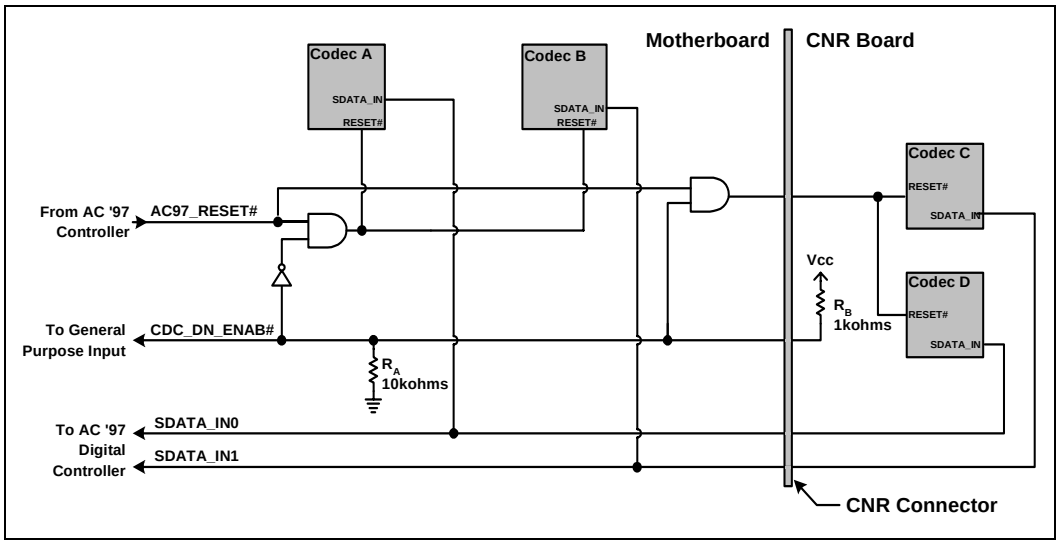


Figure 56 shows the case of two-codecs down and a dual-codec CNR. In this case, both codecs on the motherboard are disabled (while both on CNR are active) by R_A being 10 kΩ and R_B being 1 kΩ.

Figure 56. CDC_DN_ENAB# Support Circuitry for Two-Codecs on Motherboard / Two-Codecs on CNR



Circuit Notes (Figure 53 to Figure 56)

5. While it is possible to disable down codecs, as shown in Figure 53 and Figure 56, 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.
6. 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).
7. 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 when the codec has power.
8. A motherboard with one or more codecs down must implement R_A with a value of 10 kΩ.
9. 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 23. Signal Descriptions

Signal	Description
CDC_DN_ENAB#	When low, this signal indicates that the codec on the motherboard is enabled and primary on the AC97 Interface. When high, this signal 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 (ICH2).
SDATA_IN*n*	AC '97 serial data from an AC '97-compliant codec to an AC '97-compliant controller (i.e., the ICH2).

6.12.1.1 Valid Codec Configurations

Table 24. Codec Configurations

Valid Codec Configurations
AC(Primary)
MC(Primary)
AMC(Primary)
AC(Primary) + MC(Secondary)
AC(Primary) + AC(Secondary)
AC(Primary) + AMC(Secondary)

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

6.12.2 SPKR Pin Considerations

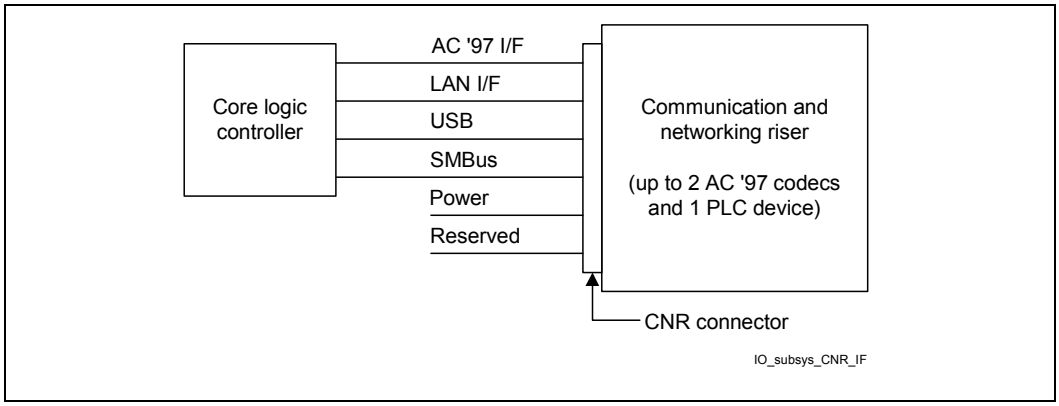
The effective impedance of the speaker and codec circuitry on the SPKR signal line must be greater than 50 kΩ. Failure to due so will cause the TCO Timer Reboot function to be erroneously disabled. SPKR is used as both the output signal to the system speaker and as a functional strap. The strap function 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, it is 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 (see figure). The value of the pull-down 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 pull-down resistor be less than 7.3 kΩ.

6.13 CNR

The Communication and Networking Riser (CNR) Specification defines a hardware-scalable Original Equipment Manufacturer (OEM) motherboard riser and interface. This interface supports multi-channel 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.

Figure 57 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 57. CNR Interface



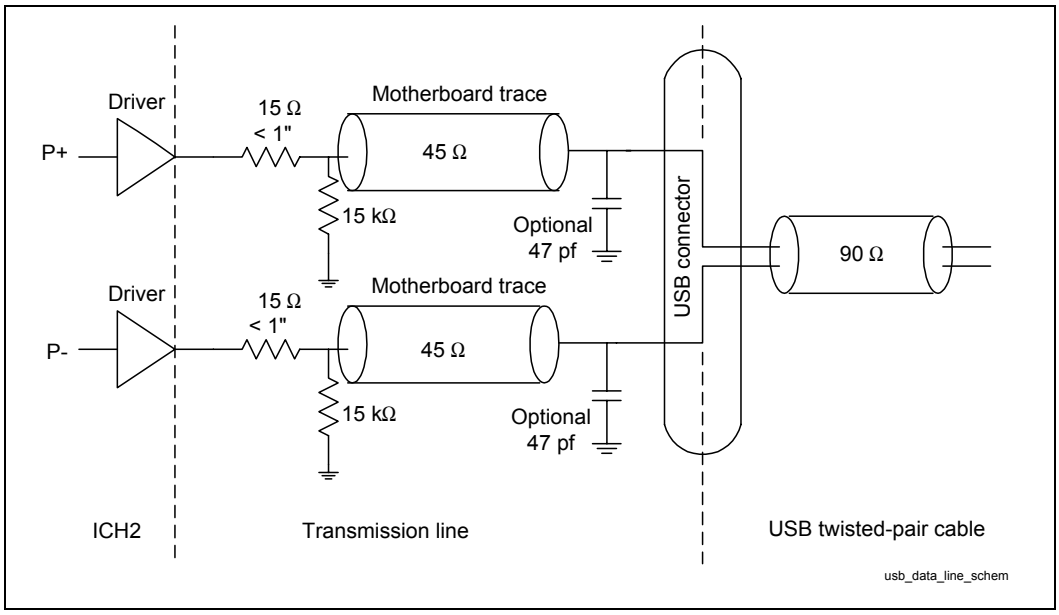
6.14 USB

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 inch). These series resistors are required for source termination of the reflected signal.
- An optional 0 to 47 pF capacitor may be placed as close to the USB connector as possible on the USB data lines (P0+/-, P1+/-, P2+/-, P3+/-). This cap can be used for signal quality (rise/fall time) and to help minimize EMI radiation.
- 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 34 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.

Figure 58 illustrates the recommended USB schematic.

Figure 58. 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

6.14.1 Disabling the Native USB Interface of 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 deasserted by pulling them up weakly to VCC3SBY, and that both function 2 & 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.

6.15 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.

6.16 I/O APIC Design Recommendation

UP systems not using the IOAPIC 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.

6.17 SMBus/SMLink Interface

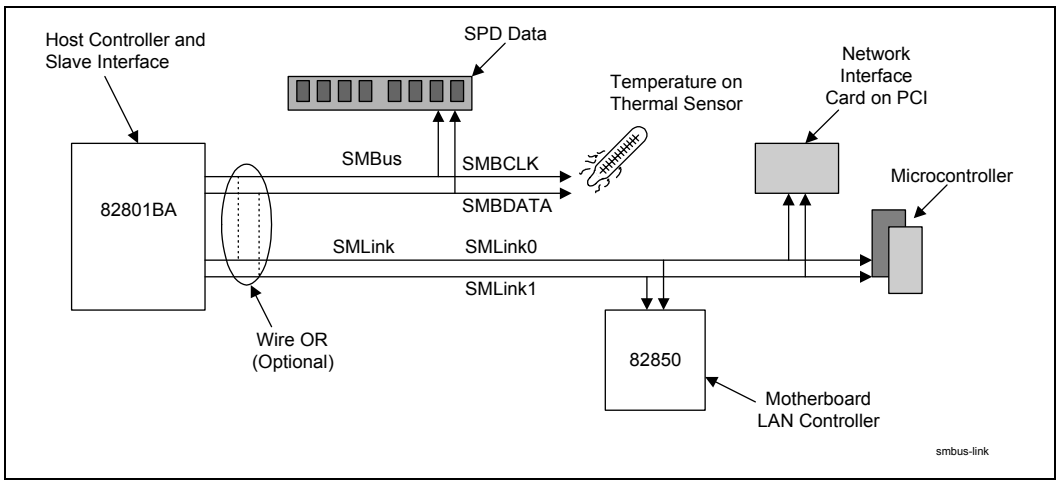
The SMBus interface on the ICH2 is the same as that on the ICH. It uses two signals (SMBCLK and SMBDATA) to send and receive data from components residing on the bus. These signals are used exclusively by the SMBus host controller. The SMBus host controller resides inside the ICH2.

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 Figure 59. 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 59. 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. Table 25 describes the pull-up requirements for different implementations of the SMBus and SMLink signals.

Table 25. Pull-Up Requirements for SMBus and SMLink

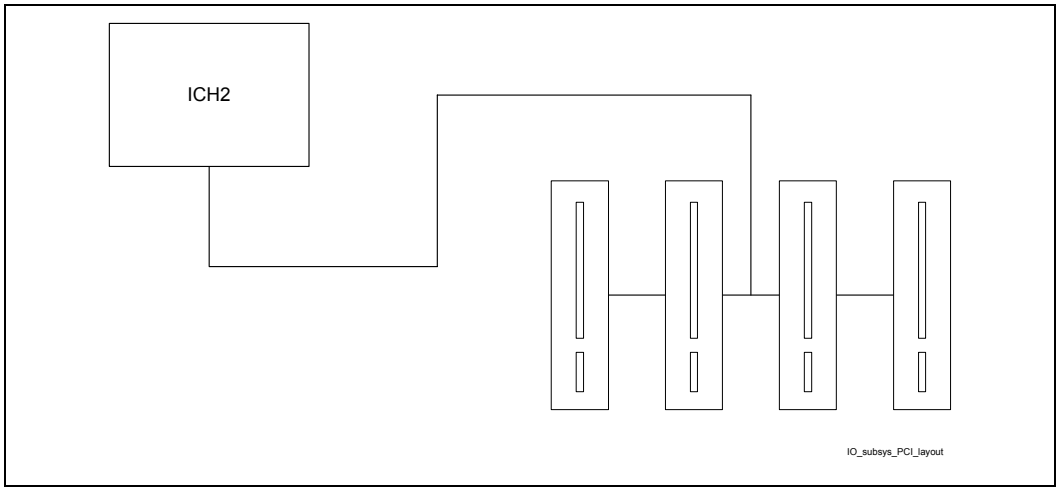
SMBus / SMLink Use	Implementation
Alert-on-LAN* signals	4.7 kΩ pull-up resistors to 3.3 VSB are required.
GPIOs	Pull-up resistors to 3.3 VSB and the signals must be allowed to change states on power-up. (For example, on power-up the Intel® ICH2 will drive **heartbeat** messages until the BIOS programs 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.

6.18 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 60. PCI Bus Layout Example



6.19 RTC

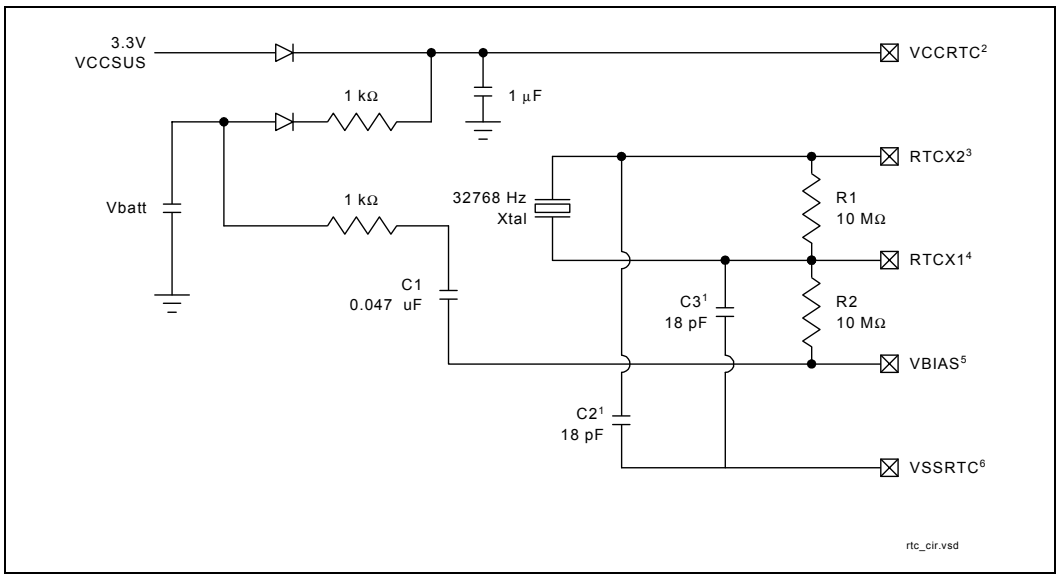
The ICH2 contains a real-time clock (RTC) with 256 bytes of battery-backed SRAM. The internal RTC module provides two key functions: keeping the date and time, and storing system data in its RAM when the system is powered down.

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

6.19.1 RTC Crystal

The ICH2 RTC module requires an external oscillating source of 32.768 kHz connected on the RTCX1 and RTCX2 pins. Figure 61 shows the external circuitry that comprises the oscillator of the ICH2 RTC.

Figure 61. External Circuitry for the Intel® ICH2 RTC



NOTES:

1. The exact capacitor value must be based on the crystal maker's recommendation. (The typical value for C2 and C3 is 18 pF for a crystal with Cload = 12.5pF)
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

6.19.2 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.

6.19.3 RTC Layout Considerations

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

6.19.4 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.

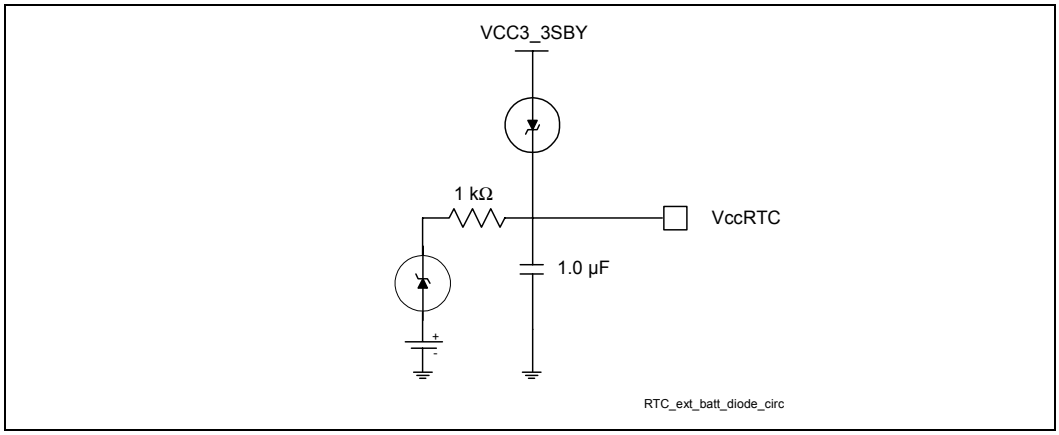
Example batteries include the Duracell* 2032, 2025 or 2016 (or equivalent), that give many years of operation. Batteries are rated by storage capacity. The battery life can be calculated by dividing the capacity by the average current required. For example, if the battery storage capacity is 170 mAh (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. Figure 62 shows an example of the used diode circuitry.

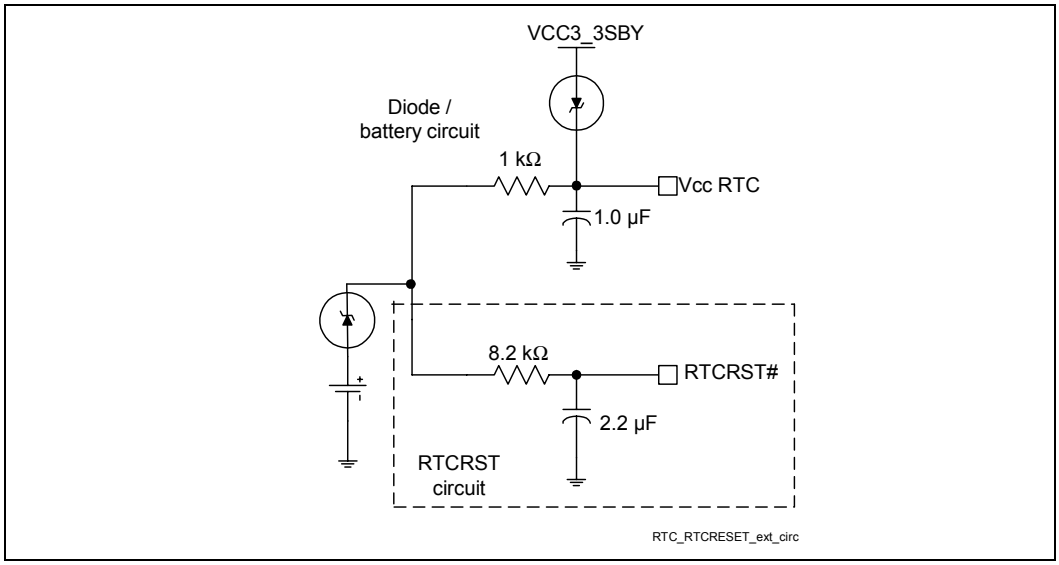
Figure 62. Diode Circuit to Connect RTC External Battery



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.

6.19.5 RTC External RTCRST Circuit

Figure 63. 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 a RC time delay such that RTCRST# goes 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 Figure 63) that allows the RTC well to be powered by the battery when system power is unavailable. Figure 63 shows an example of this circuitry when used in conjunction with the external diode circuit.

6.19.6 RTC Routing Guidelines

- All RTC OSC signals (RTCX1, RTCX2, VBIAS) should all be routed with trace lengths less than 1 inch. 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.
- Do not route any switching signals under the external components (unless on the other side of the ground plane).

6.19.7 VBIAS DC Voltage and Noise Measurements

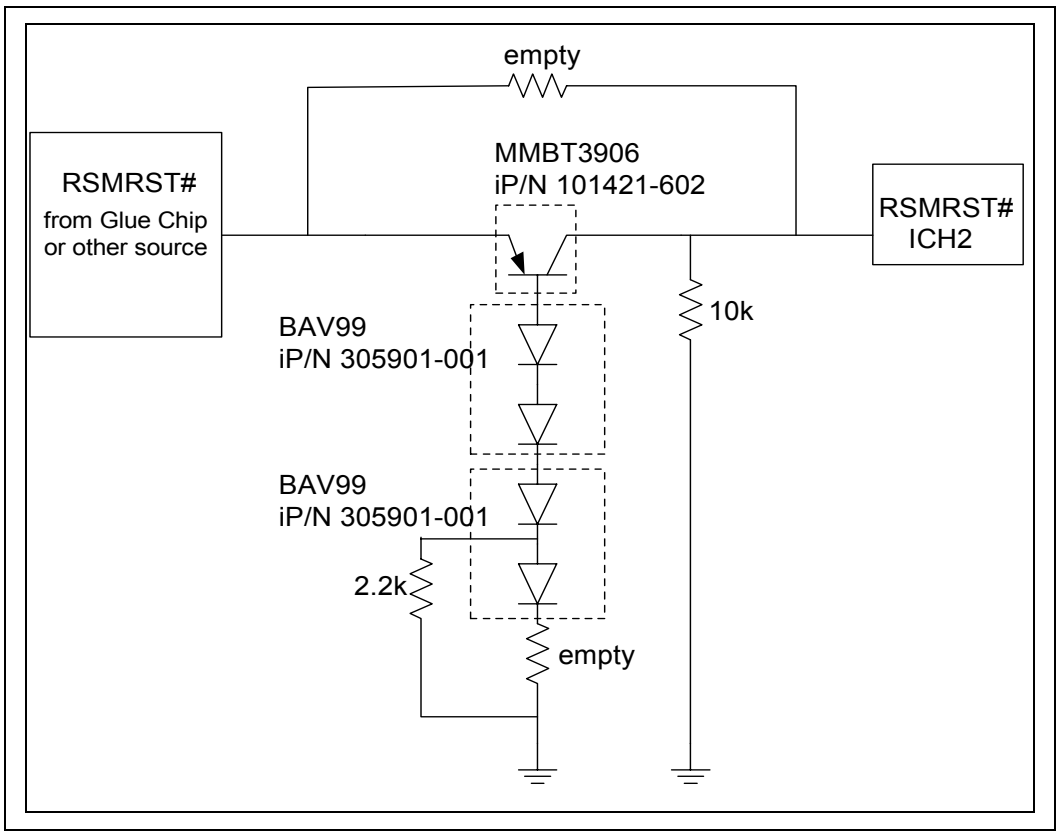
- All RTC OSC signals (RTCX1, RTCX2, VBIAS) should all be routed with trace lengths less than 1 inch. The shorter, the better.
- Steady-state VBIAS is a DC voltage of about 0.38 V ± 0.06 V.
- When the battery is inserted, VBIAS will be “kicked” to about 0.7–1.0 V, but it will return to its DC value within 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) and Intel® 82801 BAM I/O Controller Hub 2 Mobile (ICH2-M) Datasheet.*

6.19.8 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 63 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 Figure 64, 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 64. RTC Power-Well Isolation Control



6.20 LAN Layout Guidelines

The ICH2 provides several options for integrated LAN capability. The platform supports several components, depending on the target market. These guidelines use the 82562ET to refer to both the 82562ET and 82562EM. The 82562EM is specified in those cases where there is a difference.

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 the 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 Figure 65 and Table 26 for the corresponding section of the design guide. Dual footprint guidelines are in Section 6.20.5.

Figure 65. Intel® ICH2 / LAN Connect Section

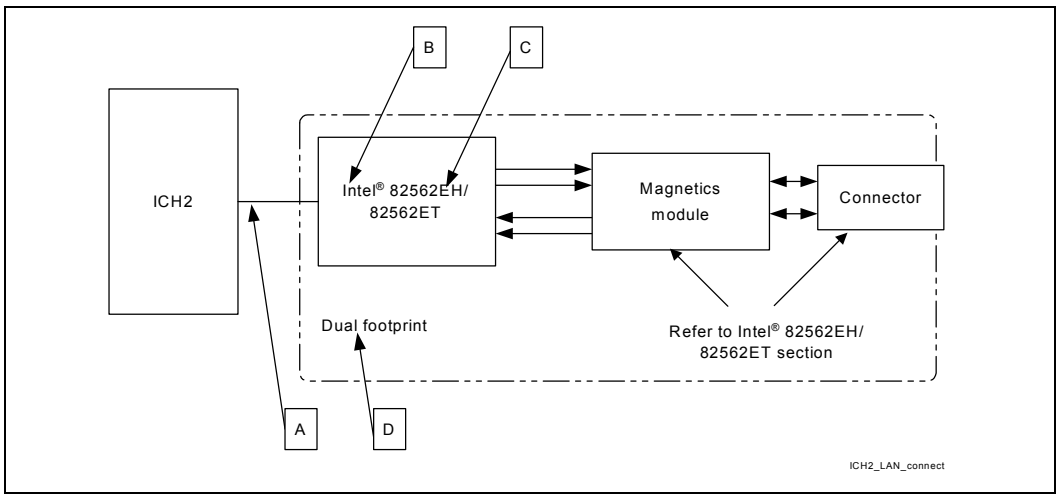


Table 26. LAN Design Guide Section Reference (see Figure 65)

Layout Section	Figure Ref.	Design Guide Section
Intel® ICH2 – LAN interconnect	A	Section 6.20.1, *Intel® ICH2 – LAN Interconnect* Guidelines
General routing guidelines	B,C,D	Section 6.20.2, *General LAN Routing Guidelines and Considerations*
Intel® 82562EH	B	Section 6.20.3, *Intel® 82562EH Home/PNA** Guidelines
Intel® 82562ET /Intel 82562EM	C	Section 6.20.4, *Intel® 82562ET / 82562EM* Guidelines
Dual layout footprint	D	Section 6.20.5, *Intel® 82562ET / 82562EH Dual Footprint* Guidelines

6.20.1 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.

6.20.1.1 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

6.20.1.2 Point-to-Point Interconnect

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

Figure 66. Single-Solution Interconnect

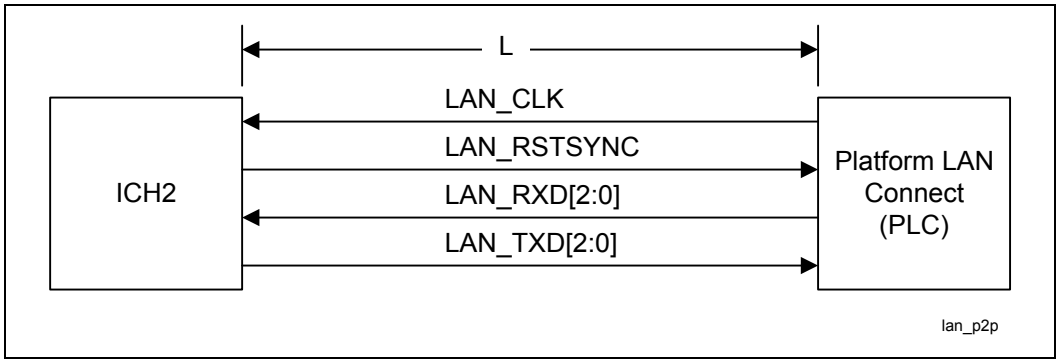


Table 27. Single-Solution Interconnect Length Requirements (see Figure 66)

Configuration	L	Comment
Intel® 82562EH	4.5” to 10”	Signal lines LAN_RXD[2:1] and LAN_TXD[2:1] are not connected.
Intel® 82562ET	3.5” to 10”	
CNR	3” to 9”	The trace length from the connector to LOM should be 0.5 inch to 3.0 inches

6.20.1.3 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 Figure 67. This figure shows the recommended trace routing lengths.

Figure 67. LOM/CNR Interconnect

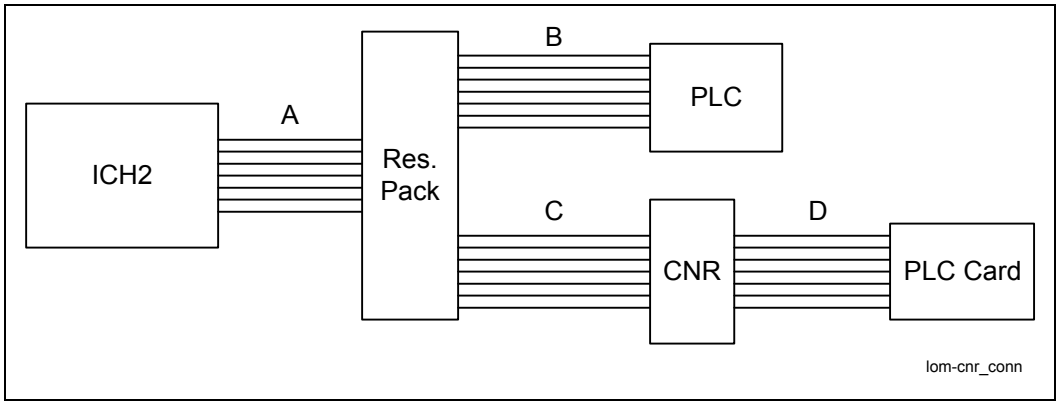


Table 28. LOM/CNR Length Requirements (see Figure 67)

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	**3.5” to 10”**	3.5” to (10.0” – A)		
Intel® 82562ET/EH card*	0.5” to 6.5”		2.5” to (9” – A)	0.5” to 3.0”

NOTE: The total trace length should not exceed 13 inches.

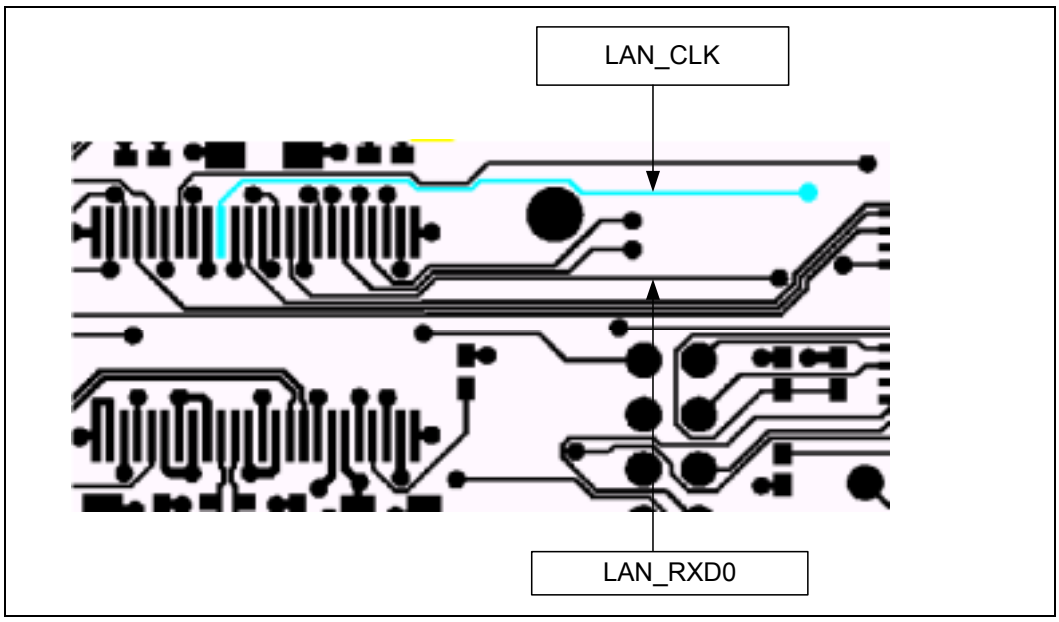
Additional guidelines for this configuration are as follows:

- 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.

6.20.1.4 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 inch shorter than the **LAN_CLK** trace. (**LAN_CLK** should always be the longest motherboard trace in each group.)

Figure 68. LAN_CLK Routing Example



6.20.1.5 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.

6.20.1.6 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.

6.20.1.7 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.

6.20.2 General LAN Routing Guidelines and Considerations

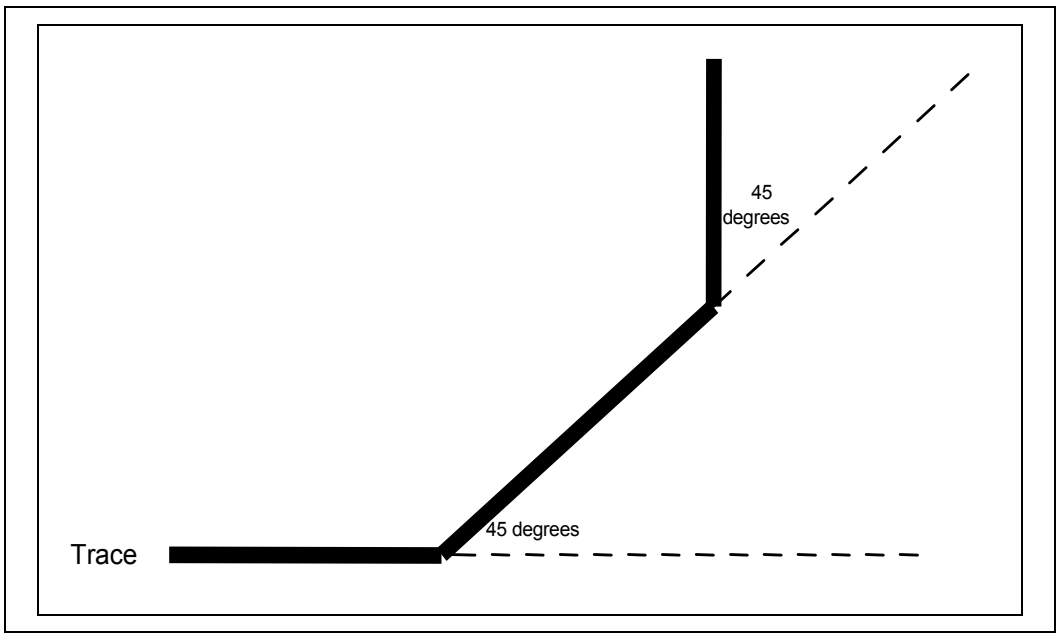
6.20.2.1 General Trace Routing Considerations

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 inch.
- 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 inches. (Many customer designs with differential traces longer than 5 inches 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 100 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 is 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° bends instead. Refer to Figure 69.
- 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 69. Trace Routing



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. This can lower the differential impedance by 10 Ω when the traces within a pair are closer than 0.030 inch (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.

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 inch 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.

6.20.2.2 Power and Ground Connections

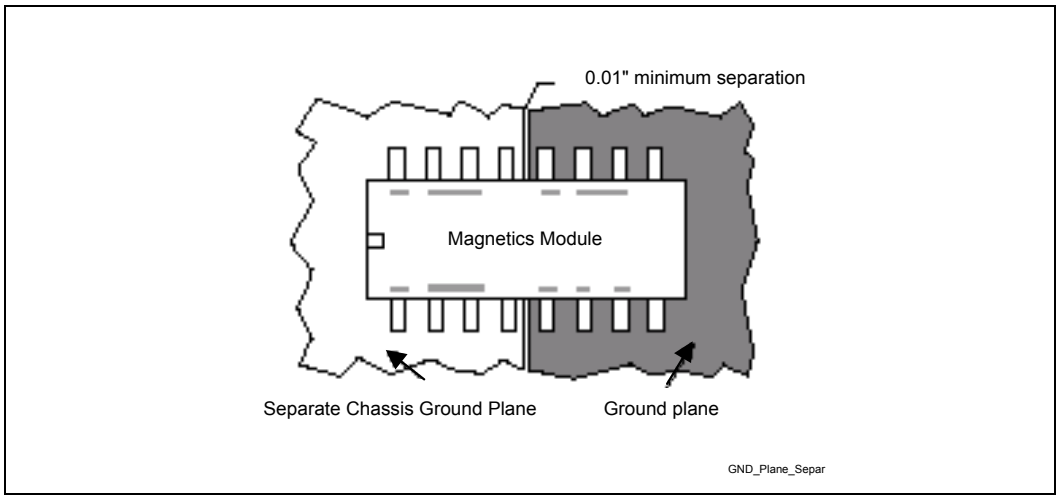
Comply with the following rules and guidelines for power and ground connections:

- 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.

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 70. 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 minimizes the loop area.
- Avoid fast rise/fall times as much as possible. Signals with fast rise and fall times contain many high-frequency harmonics that 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 1.6mm (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.

6.20.2.3 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.

6.20.2.4 Common Physical Layout Issues

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

10. Unequal length of the two traces within a differential pair. Inequalities create common-mode noise and distort transmit or receive waveforms.
11. 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.
12. 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 specification-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).
13. 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 inch from the differential traces.
14. 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; this can greatly degrade the receiver's BER over long cables. After exiting the PLC, the transmit traces should be kept 0.3 inch 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.
15. 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.)
16. 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.
17. 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.

18. 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.
19. 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 specifications. This 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).

6.20.3 Intel® 82562EH Home/PNA* Guidelines

For correct LAN performance, designers must follow the general guidelines outlined in Section 6.20.2. Additional guidelines for implementing an 82562EH Home/PNA* LAN connect component are listed in the following sections. Refer to Section 1.2 for a list of related documents.

6.20.3.1 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 the 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.

6.20.3.2 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.

6.20.3.3 Crystals and Oscillators

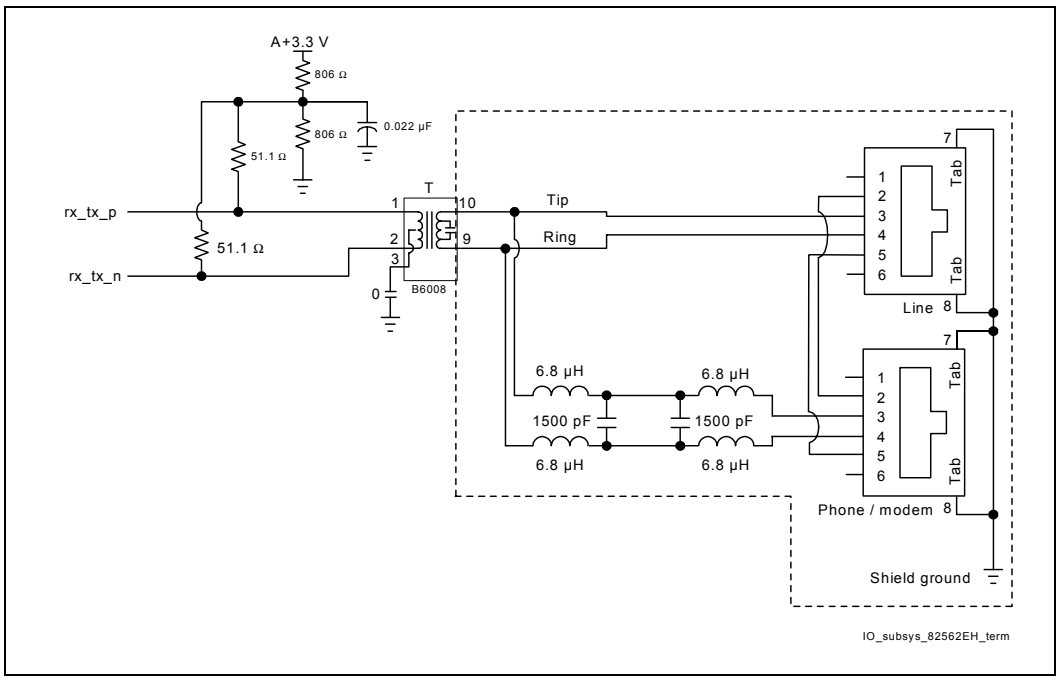
To minimize the effects of EMI, clock sources should not be placed near I/O ports or board edges. Radiation from these devices may be coupled onto the I/O ports or out of the system chassis. Crystals should also be kept away from the HomePNA magnetics module to prevent communication interference. If 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.

6.20.3.4 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 Intel 82562EH. The center, common point between the 51.1 Ω resistors is connected to a voltage-divider network. The termination is shown in Figure 71.

Figure 71. 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.

6.20.3.5 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 Figure 72).

Figure 72. Critical Dimensions for Component Placement

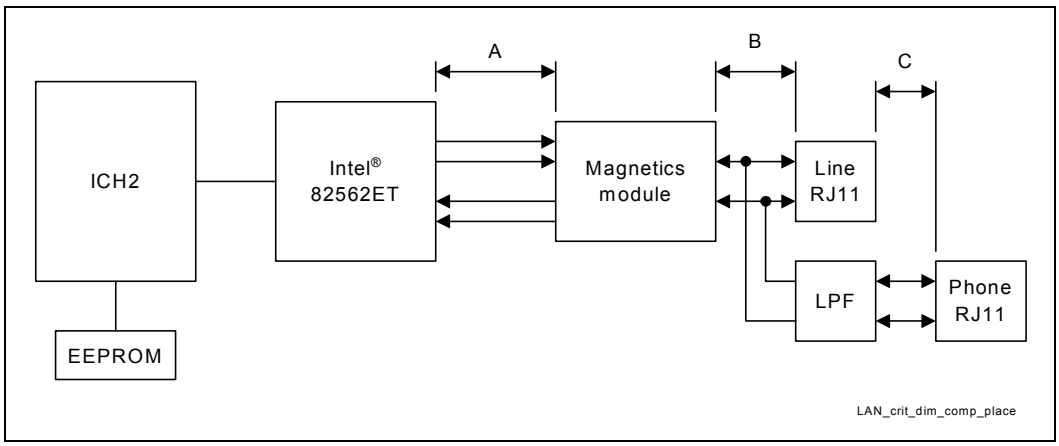


Table 29. Critical Dimensions for Component Placement (see Figure 72)

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

Distance from Magnetics Module to Line RJ11

This distance ‘B’ should be given highest priority and should be less then 1 inch. 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 receive circuit performance and contribute to emissions radiated from the transmit side.

Distance from Intel® 82562EH to Magnetics Module

Due to the high speed of signals present, distance ‘A’ between the 82562EH and the magnetic should also be less than 1 inch, but should be second priority relative to distance from connects to the magnetic 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 trace route. The reflections of a signal may have a high-frequency component that may contribute more EMI than the original signal itself.

Distance from LPF to Phone RJ11

This distance ‘C’ should be less then 1 inch. 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.

6.20.4 Intel® 82562ET / 82562EM Guidelines

For correct LAN performance, designers must follow the general guidelines outlined in Section 6.20.2. Additional guidelines for implementing an 82562ET or 82562EM LAN connect component are provided in the following sub-sections. For additional information, refer to the *Intel® 82562ET Platform LAN Connect (PLC) Datasheet and PCB Design for the Intel® 82562 ET/EM Platform LAN Connect.*

6.20.4.1 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.

6.20.4.2 Crystals and Oscillators

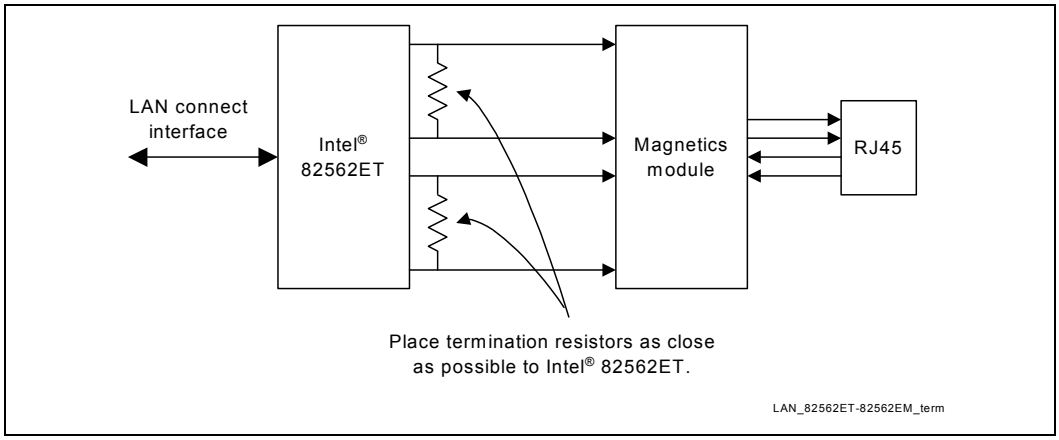
To minimize the effects of EMI, clock sources should not be placed near I/O ports or board edges. Radiation from these devices may be coupled onto the I/O ports or out of the system chassis. Crystals should also be kept away from the Ethernet magnetics module to prevent interference with communication. 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.

6.20.4.3 Intel® 82562ET / 82562EM Termination Resistors

The 100 Ω (1%) resistor used to terminate the differential transmit pairs (TDP/TDN) and the 120 Ω (1%) receive differential pairs (RDP/RDN) should be placed as close as possible to the LAN connect component (82562ET or 82562EM) as possible. 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 73. Intel® 82562ET/82562EM Termination



6.20.4.4 Critical Dimensions

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 Figure 74).

Figure 74. Critical Dimensions for Component Placement

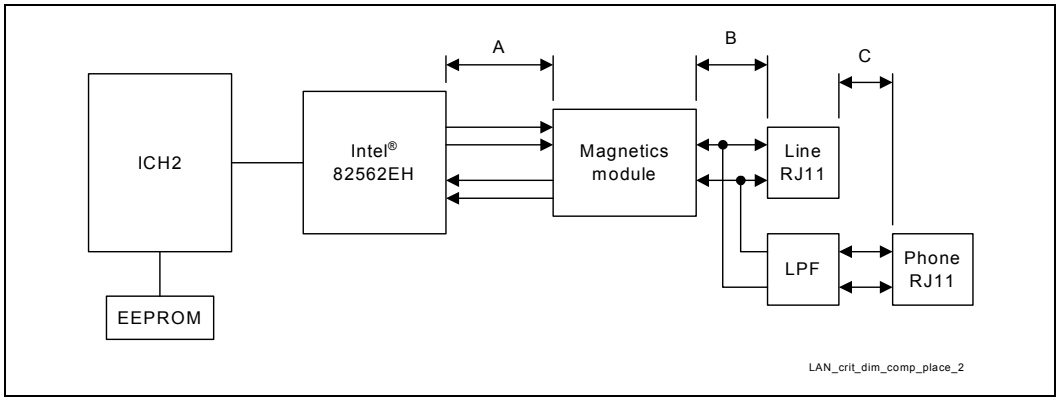


Table 30. Critical Dimensions for Component Placement (see Figure 74)

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

Distance from Magnetics Module to RJ45

The distance A in Figure 74 should be given the highest priority in board layout. The separation between the magnetic module and the RJ45 connector should be kept less than 1 inch. 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 RJ45 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 inch 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.

6.20.4.5 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, they 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.

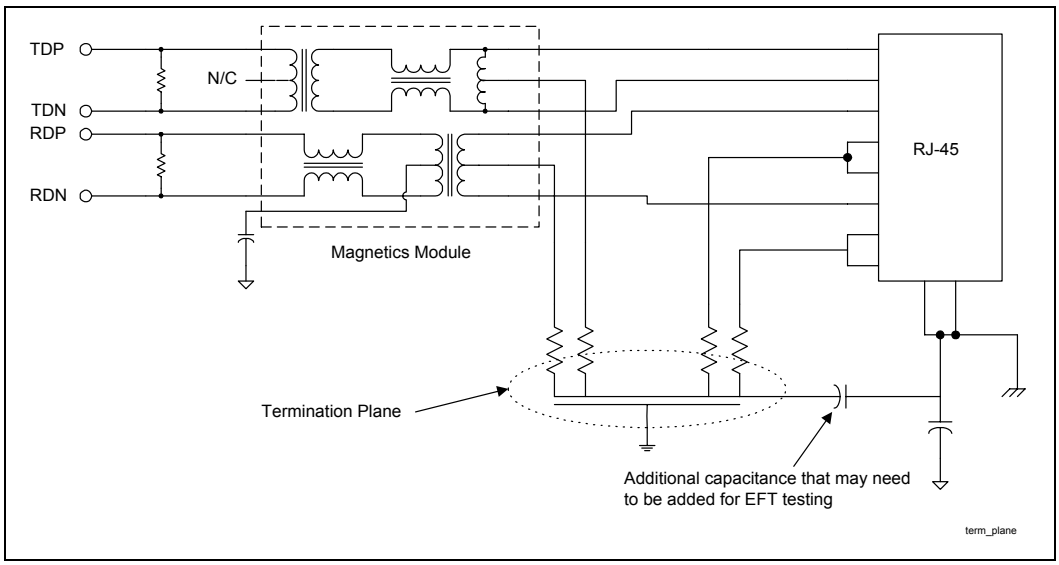
Terminating Unused Connections

In Ethernet designs, it is common practice to terminate to ground both unused connections on the RJ45 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 75. Termination Plane



6.20.5 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 6.20.1 through 6.20.4 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 Figure 76 and Figure 77.

Figure 76. Dual-Footprint LAN Connect Interface

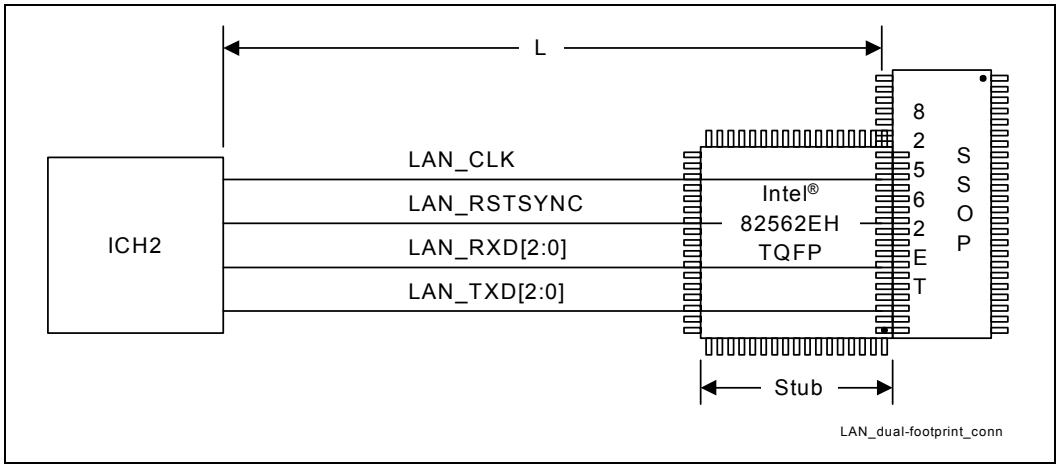
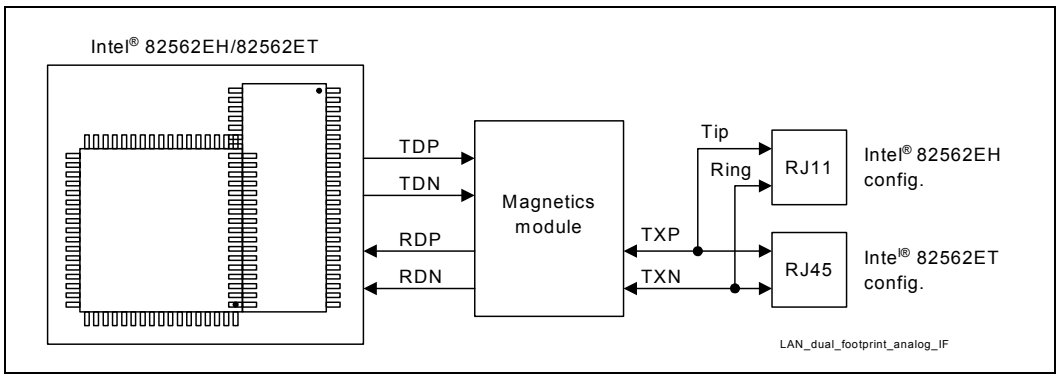


Figure 77. Dual-Footprint Analog Interface



The following are additional guidelines for this configuration:

- L = 0.5 inch to 6.5 inches
- Stub < 0.5 inch
- Either 82562EH or 82562ET/82562EM can be installed, but not both.
- Intel 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.

6.21 LPC/FWH

The following provides general guidelines for compatibility and design recommendations for supporting the FWH flash BIOS device. The majority of the changes will be incorporated in the BIOS.

6.21.1 In-Circuit FWH Programming

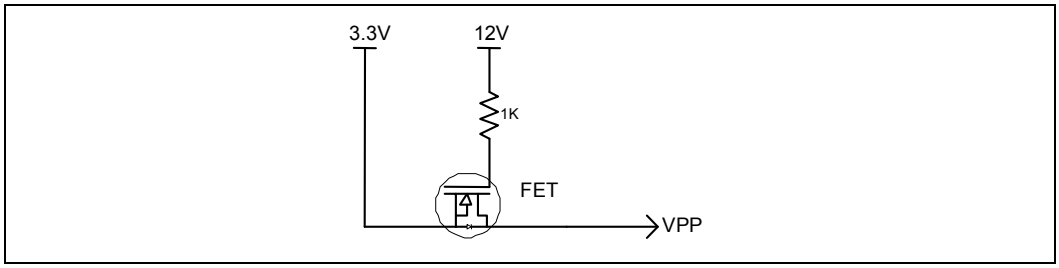
All cycles destined for the FWH will appear on the PCI. The ICH2 hub interface-to-PCI Bridge puts all processor boot cycles out on the PCI (before sending them out on the FWH interface). If the ICH2 is set for subtractive decode, these boot cycles can be accepted by a positive decode agent on PCI. This 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.

6.21.2 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. To decrease programming time it becomes necessary to apply 12 V to the V_{PP} pin. Figure 78 shows a circuit that allows 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 78. FWH VPP Isolation Circuitry



6.21.3 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.

6.22 Processor PLL Filter Recommendation

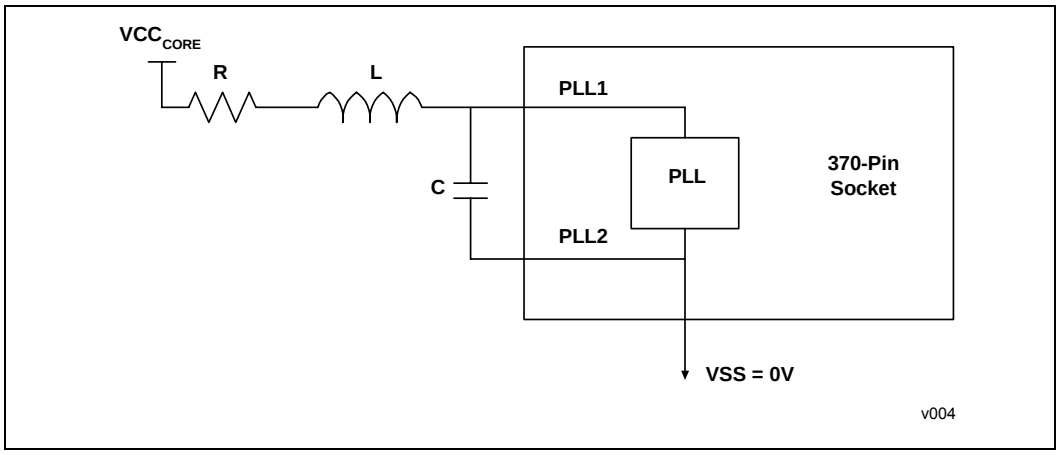
6.22.1 Processor PLL Filter Recommendation

All Celeron processors have internal PLL clock generators that are analog and require quiet power supplies to minimize jitter.

6.22.2 Topology

The general desired topology is shown in Figure 79. Not shown are parasitic routing and local decoupling capacitors. Excluded from the external circuitry are parasitics associated with each component.

Figure 79. Filter Topology



6.22.3 Filter Specification

The function of the filter is to protect the PLL from external noise through low-pass attenuation. In general, the low-pass description forms an adequate description for the filter.

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

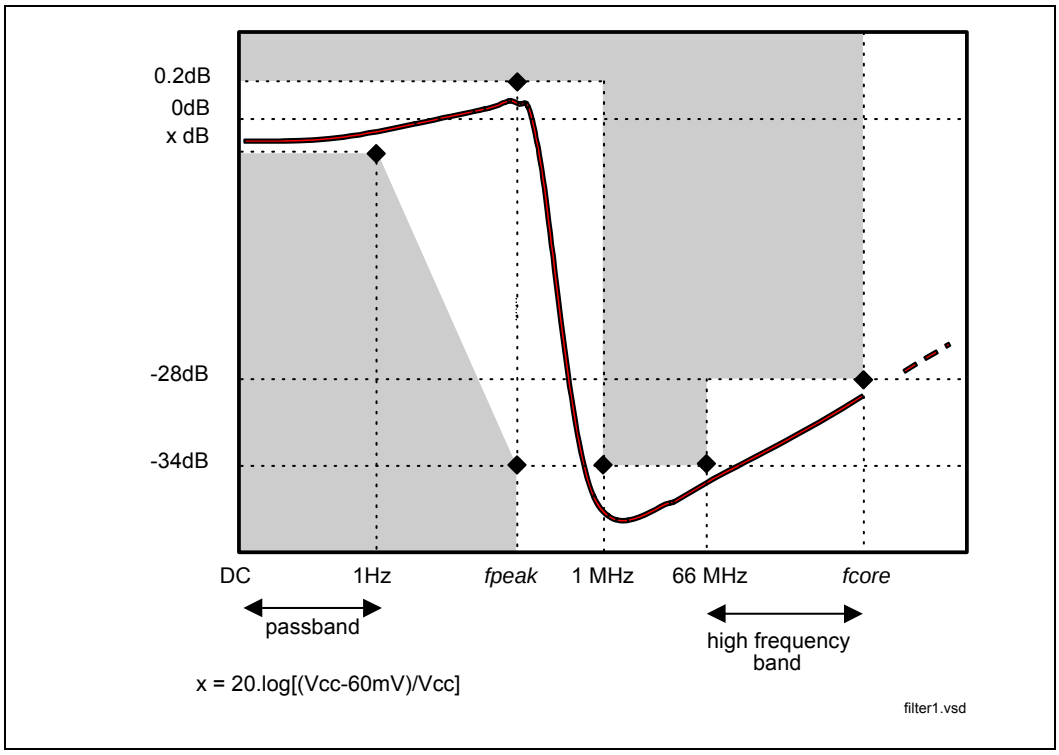
< 0.5 dB attenuation in pass band (see DC drop in next set of requirements)

> 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 Figure 80.

Figure 80. Filter Specification



NOTES:
1. Diagram not to scale.
2. No specification for frequencies beyond fcore.
3. fpeak, if it exists, should be less than 0.05 MHz.

Other requirements:

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

6.22.4 Recommendation for Intel® Platforms

The following tables are examples of components that meet Intel's recommendations, when configured in the topology presented in Figure 79.

Table 31. Inductor

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

Table 32. 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 33. Resistor

Value	Tolerance	Power	Note
1 Ω	10%	1/16W	Resistor may be implemented with trace resistance in which 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 a 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, the maximum DCR of the L should be less than 2.0 - 1.1 = 0.9 Ω, which precludes using some inductors.

Other routing requirements:

- C should be close to PLL1 and PLL2 pins, < 0.1 Ω per route. These routes do not count towards the minimum damping R requirement.
- PLL2 route should be parallel and next to PLL1 route (minimize loop area).
- L should be close to C; any routing resistance should be inserted between VCC_{CORE} and L.
- Any discrete R should be inserted between VCC_{CORE} and L.

Figure 81. Using Discrete R

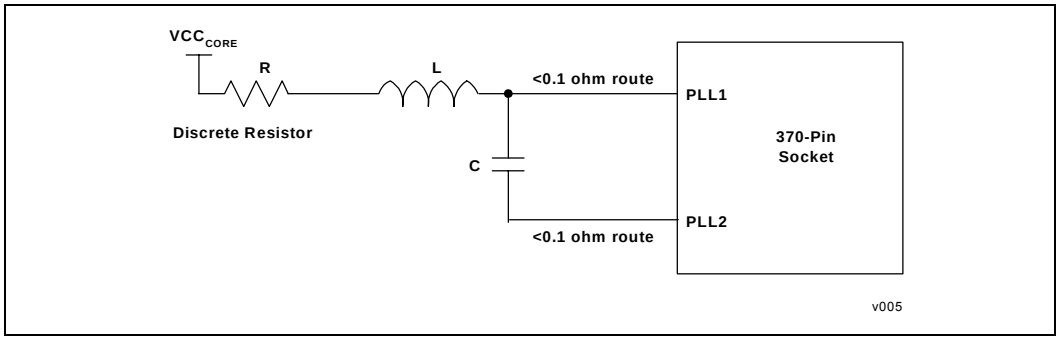
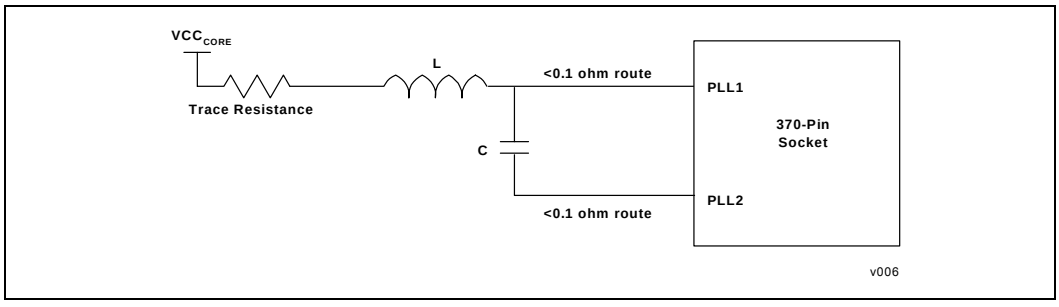


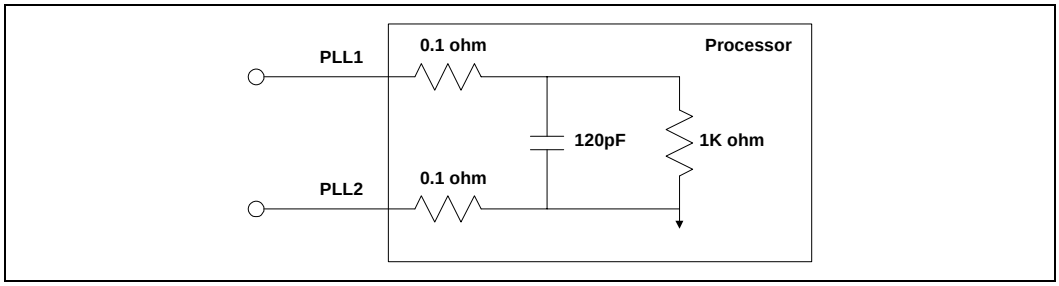
Figure 82. No Discrete R



6.22.5 Custom Solutions

As long as filter performance as specified in the previous “Filter Specification” figure and other requirements outlined in Section 6.22.1. are satisfied, other solutions are acceptable. Custom solutions should be simulated against a standard reference core model, which is shown in Figure 83.

Figure 83. Core Reference Model



NOTES:

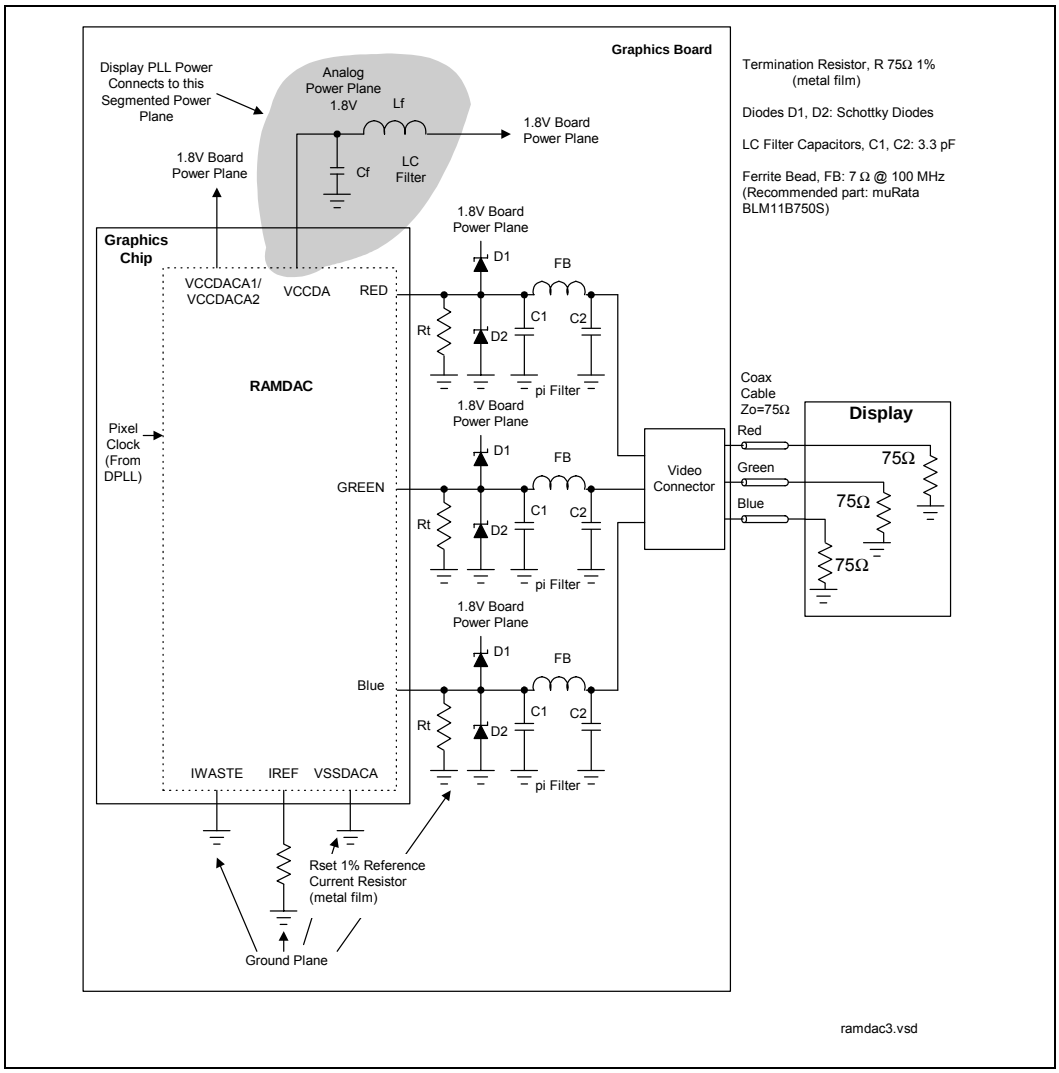
1. 0.1 Ω resistors represent package routing1.
2. 120 pF capacitor represents internal decoupling capacitor.
3. 1 kΩ resistor represents small signal PLL resistance.
4. Be sure to include all component and routing parasitics.
5. Please sweep across component/parasitic tolerances.
6. To observe IR drop, use DC current of 30 mA and minimum VCC_{CORE} level.

1 For other modules (interposer, DMM, etc), adjust routing resistor if desired, but use minimum numbers.

6.23 RAMDAC/Display Interface

Figure 84 shows the interface of the RAMDAC analog current outputs with the display. Each DAC output is doubly-terminated with a 75 Ω resistance; one 75 Ω resistance from the DAC output to the board ground and the other termination resistance exists within the display. The equivalent DC resistance at the output of each DAC output is 37.5 Ω. The output current of each DAC flows into this equivalent resistive load to produce a video voltage without the need for external buffering. There is also an LC pi-filter that is used to reduce high-frequency glitches and noise, and reduce EMI. To maximize the performance, the filter impedance, cable impedance, and load impedance should be the same. The LC pi-filter consists of two 3.3 pF capacitors and a ferrite bead with a 75 Ω impedance at 100 MHz. The LC pi-filter is designed to filter glitches produced by the RAMDAC while maintaining adequate edge rates to support high-end display resolutions.

Figure 84. Schematic of RAMDAC Video Interface



NOTE: Diodes D_1, D_2 are clamping diodes and may not be necessary to populate.

In addition to the termination resistance and LC pi-filter, there are protection diodes connected to the RAMDAC outputs to help prevent latch-up. The protection diodes must be connected to the same power supply rails as the RAMDAC. An LC filter is recommended to connect the segmented analog 1.8V power plane of the RAMDAC to the 1.8V board power plane. The LC filter is recommended to be designed for a cut-off frequency of 100 kHz.

6.23.1 Reference Resistor (Rset) Calculation

The full-swing video output is designed to be 0.7V according to the VESA video standard. With an equivalent DC resistance of 37.5 Ω (two 75 Ω in parallel - one 75 Ω termination on the board and one 75 Ω termination within the display), the full-scale output current of a RAMDAC channel is 0.7/37.5 Ω = 18.67 mA. Since the RAMDAC is an 8-bit current-steering DAC, this full-scale current is equivalent 255I, where I is a unit of current. Therefore, the unit current or LSB current of the DAC signals equals 73.2 µA. The reference circuitry generates a voltage across this R_{set} resistor equal to a bandgap voltage divided-by-three (409 mV). The RAMDAC reference current generation circuitry is designed to generate a 32I reference current using the reference voltage and the R_{set} value. To generate a 32I reference current for the RAMDAC, the reference current setting resistor, R_{set}, is calculated from the following equation:

$$R_{set} = VREF/32I = 0.409V/32*73.2\mu A = 174\ \Omega$$

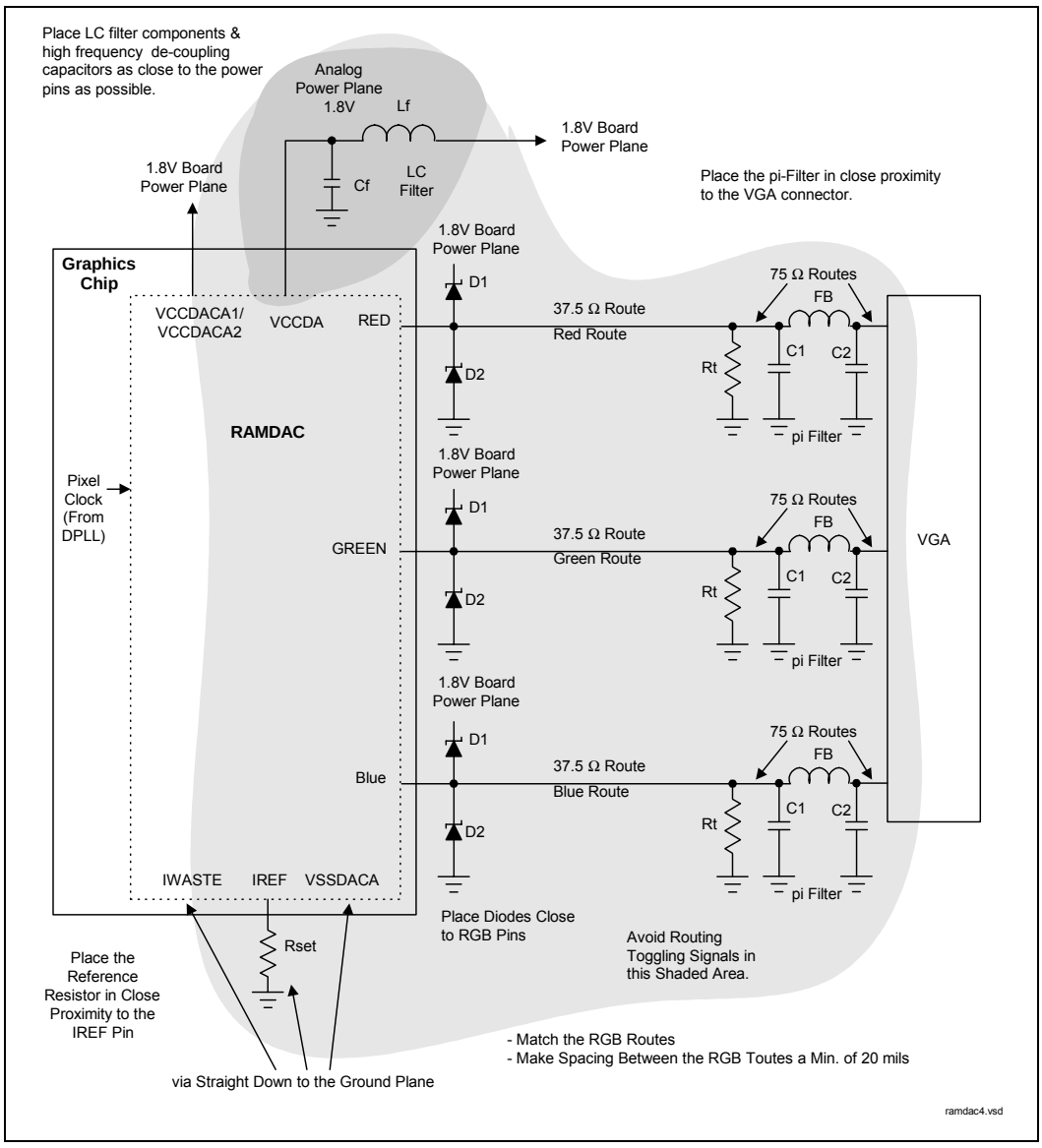
6.23.2 RAMDAC Board Design Guidelines

The RAMDAC layout is shown in Figure 85, Figure 86, and Figure 87. An RLC network should be used between the board power plane and the board ground plane. The recommended RAMDAC routing for a four layer board is such that the red, green and blue video outputs be routed on the top (bottom) layer over (under) a solid ground plane to maximize noise rejection characteristics of the video outputs. It is essential to avoid toggling signals from being routed next to the video output signals to the VGA connector. A 20 mil spacing between any video route and any other routes is recommended.

Matching of the video routes (red, green, and blue) from the RAMDAC to the VGA connector is also essential. The routing for these signals should be as similar as possible (i.e., same routing layer(s), same number of vias, same routing length, same bends and jogs).

Figure 85 shows recommended RAMDAC component placement and routing. The termination resistance can be placed anywhere along the video route from the RAMDAC output to the VGA connector as long as the impedance of the traces are designed as indicated in Figure 85. The pi-filters are recommended to be placed in close proximity to the VGA connector to maximize EMI filtering effectiveness. The LC filter components for the RAMDAC/PLL power plane, de-coupling capacitors, latch-up protection diodes, and the reference resistor are recommended to be placed in close proximity to the respective pins.

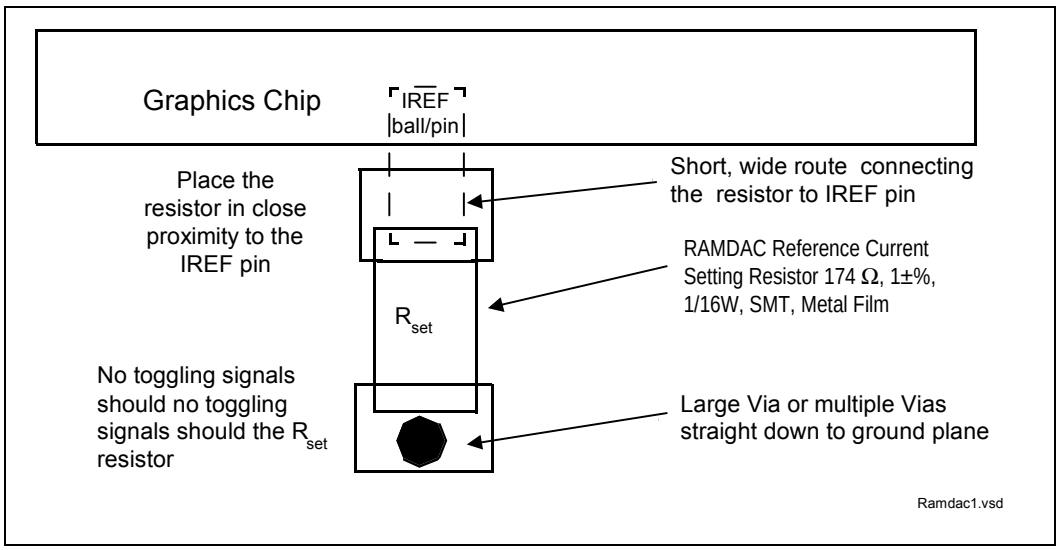
Figure 85. RAMDAC Component and Routing Guidelines



NOTE: Diodes D_1, D_2 are clamping diodes and may not be necessary to populate.

Figure 86 shows the recommended reference resistor placement and connections.

Figure 86. Recommended RAMDAC Reference Resistor Placement and Connections



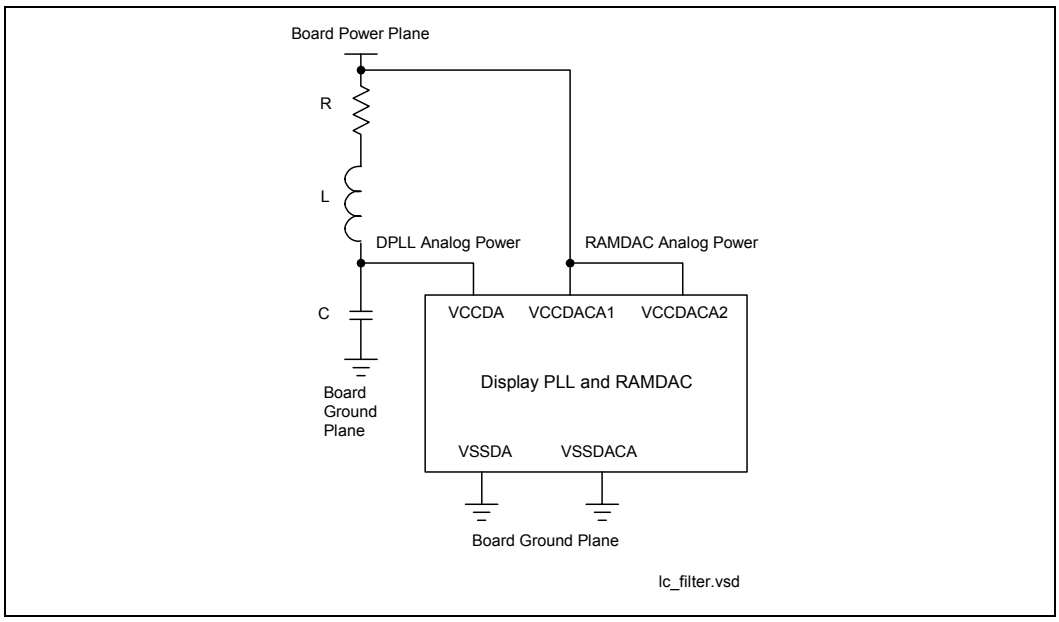
6.24 DPLL Filter Design Guidelines

The 810ET2 chipset contains sensitive phase-locked loop circuitry (the DPLL) that can cause excessive dot clock jitter. Excessive jitter on the dot clock may result in a "jittery" image. An LC filter network connected to the DPLL analog power supply is recommended to reduce dot clock jitter.

The DPLL bandwidth varies with the resolution of the display and can be as low as 100 kHz. In addition, the DPLL jitter transfer function can exhibit jitter peaking effects in the range from 100 kHz to a few megahertz. A low-pass LC filter is recommended for the display PLL analog power supply designed to attenuate power supply noise with frequency content from 100 kHz and above so that jitter amplification is minimized.

Figure 87 is a block diagram showing the recommended topology of the filter connection (parasitics not shown). The display PLL analog power rail (VCCDA) is connected to the board power plane through an LC filter. The RAMDAC analog power rails (VCCDACA1 and VCCDACA2) are connected directly to the 1.8V board power plane.

Figure 87. Recommended LC Filter Connection



The resistance from the inductor to the board 1.8V power plane represents the total resistance from the board power plane to the filter capacitor. This resistance, which can be a physical resistor, routing/via resistance, parasitic resistance of the inductor or combinations of these, acts as a damping resistance for the filter and effects the response of the filter.

The LC filter topology shown Figure 87 is the preferred choice since the RAMDAC minimum voltage level requirement does not place constraints on the LC filter for the DPLL. The maximum current flowing into the DPLL analog power is approximately 30 mA, much less than that of the RAMDAC, and therefore, a filter inductor with a higher DC resistance can be tolerated. With the topology in the above figure, the filter inductor DC current rating must be at least 30 mA and the maximum IR drop from the board power plane to the VCCDA ball should be 100 mV or less (corresponds to a series resistance equal to or less than 3.3 Ω. This larger dc resistance tolerance improves the damping and the filter response.

6.24.1 Filter Specification

The low-pass filter specification with the input being the board power plane and the output measured across the filter capacitor is defined as follows for the filter topology shown in the above figure.

- pass band gain < 0.2 dB
- DC IR drop from board power plane to the DPLL VCCDA ball < 100 mV (and a maximum DC resistance < 3.3 Ω)
- filter should support a DC current > 30 mA
- minimum attenuation from 100 kHz to 10 MHz = 10 dB (desired attenuation > 20 dB)
- a magnetically shielded inductor is recommended

The resistance from the board power plane to the filter capacitor node should be designed to meet the filter specifications outlined above. This resistance acts as a damping resistance for the filter and affects the filter characteristics. This resistance includes the routing resistance from the board power plane connection to the filter inductor, the filter inductor parasitic resistance, the routing from the filter inductor to the filter capacitor, and resistance of the associated vias. Part of this resistance can be a physical resistor. A physical resistor may not be needed depending on the resistance of the inductor and the routing/via resistance.

The filter capacitance should be chosen with as low of an ESR (equivalent series resistance) and ESL (equivalent series inductance) as possible to achieve the best filter performance. The parasitics of the filter capacitor can alter the characteristics of the filter significantly and even cause the filter to be ineffective at the frequencies of interest. The LC filter must be simulated with all the parasitics of the inductor, capacitor, and associated routing parasitics along with tolerances.

6.24.2 Recommended Routing/Component Placement

- The filter capacitance should be placed as close to the VCCDA ball as possible so that the routing resistance from the filter capacitor lead to the package VCCDA ball is < 0.1 Ω.
- The VSSDA ball should via straight down to the board ground plane.
- The filter inductor should be placed in close proximity to the filter capacitor and any routing resistance should be inserted between the board power plane connection and the filter inductor.
- If a discrete resistor is used for the LC filter, the resistor should be placed between the board power plane connection and the filter inductor.

6.24.3 Example LC Filter Components

Table 34 and Table 35 show example LC components and resistance for the LC filter topology shown in the “Recommended LC Filter Connection” figure.

Table 34. DPLL LC Filter Component Example

Component	Manufacturer	Part No.	Description
Capacitor	KEMET	T495D336MD16AS	33 µF ±20%, 16VDC, ESR=0.225 Ω @ 100 kHz, ESL=2.5 nH
Inductor	muRATA	LQG11A68NJ00	68 nH ±5%, 300 mA, Max dc resistance = 0.8 Ω, size=0603
Resistance			< 3.3 Ω

The resistance of the filter is defined as the total resistance from the board power plane to the filter inductor. If a discrete resistor is used as part of this resistance, the tolerance and temperature coefficient should be accounted for so that the maximum DC resistance in this path from the board power plane connection to the DPLL VCCDA ball is less than 3.3 Ω to meet the IR drop requirement.

Table 35. Additional DPLL LC Filter Component Example

Component	Manufacturer	Part No.	Description
Capacitor	KEMET	T495D336MD16AS	33 µF ±20%, 16VDC, ESR=0.225 Ω @ 100 kHz, ESL=2.5 nH
Inductor	muRATA	LQG21NR10K10	100 nH ±10%, 250 mA, Max dc resistance = 0.26 Ω, size=0805, magnetically shielded

As an example, Figure 88 is a Bode plot showing the frequency response using the capacitor and inductor values shown in Table 36. The capacitor and inductor values were held constant while the resistance was swept for four different combinations of resistance (the resistance of the discrete/trace resistor and the resistance of the inductor), each resulting in a different series resistance. In addition, different values for the resistance of the inductor were assumed based on its max and typical DC resistance. This is summarized in Table 36. This yielded the four different frequency response curves shown in Figure 88.

Figure 88. Frequency Response (see Table 36)

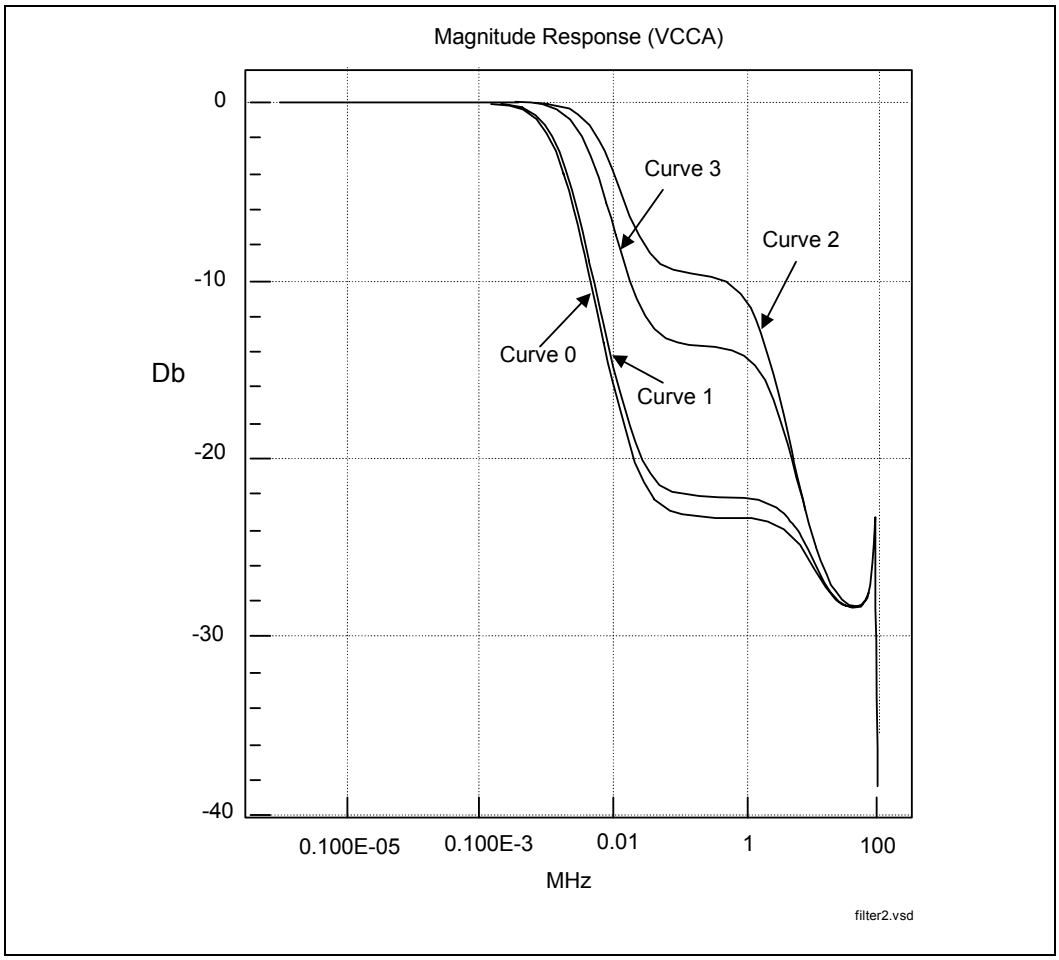


Table 36. Resistance Values for Frequency Response Curves

Curve	RTRACE + RDISCRETE	RIND
0	2.2 Ω	0.8 Ω
1	2.2 Ω	0.4 Ω
2	0 Ω	0.4 Ω
3	0 Ω	0.8 Ω

As series resistance (R_{TRACE} + $R_{DISCRETE}$ + R_{IND}) increases, the filter response (i.e., attenuation in PLL bandwidth) improves. There is a limit of 3.3 Ω total series resistance of the filter to limit DC voltage drop.

7 Clocking

7.1 Clock Generation

There is only one clock generator component required in an Intel 810ET2 chipset system. The CK810E clock chip is pin compatible with the CK810 clock chip, which comes in a single 56-pin SSOP package.

There is one pin function change in the CK810E relative to the CK810, the REFCLK Reset Strap:

Table 37. REFCLK Reset Strap for CK810 vs. CK810E

At reset	APIC Clock Strap	System Bus Freq Select (SEL1)
After reset	14 MHz Clock	14 MHz Clock
Reset default	Internal Pull-up for 33 MHz APIC Clock Populate External 10 kΩ Resistor to Ground for 16 MHz	Internal Pull-down for 66 MHz or 100 MHz Bus Freq Select (SEL0) External Drive to 1 to Select 133 MHz System Bus

The CK810E is a mixed voltage component. Some of the output clocks are 3.3V and some of the output clocks are 2.5V. As a result, the CK810E device requires both 3.3V and 2.5V. These power supplies should be as clean as possible. Noise in the power delivery system for the clock driver can cause noise on the clock lines. The CK810E provides the clock frequencies indicated in Table 38.

Table 38. Intel® 810ET2 Chipset Clocks (2-DIMM)

Number	Clock	Frequency
3	Processor Clocks	66/100/133 MHz
9	SDRAM Clocks	100 MHz
8	PCI Clocks	33 MHz
2	APIC Clocks	16.67/33 MHz
2	48 MHz Clocks	48 MHz
2	3V66 MHz Clocks	66 MHz
1	REF Clock	14.31818 MHz

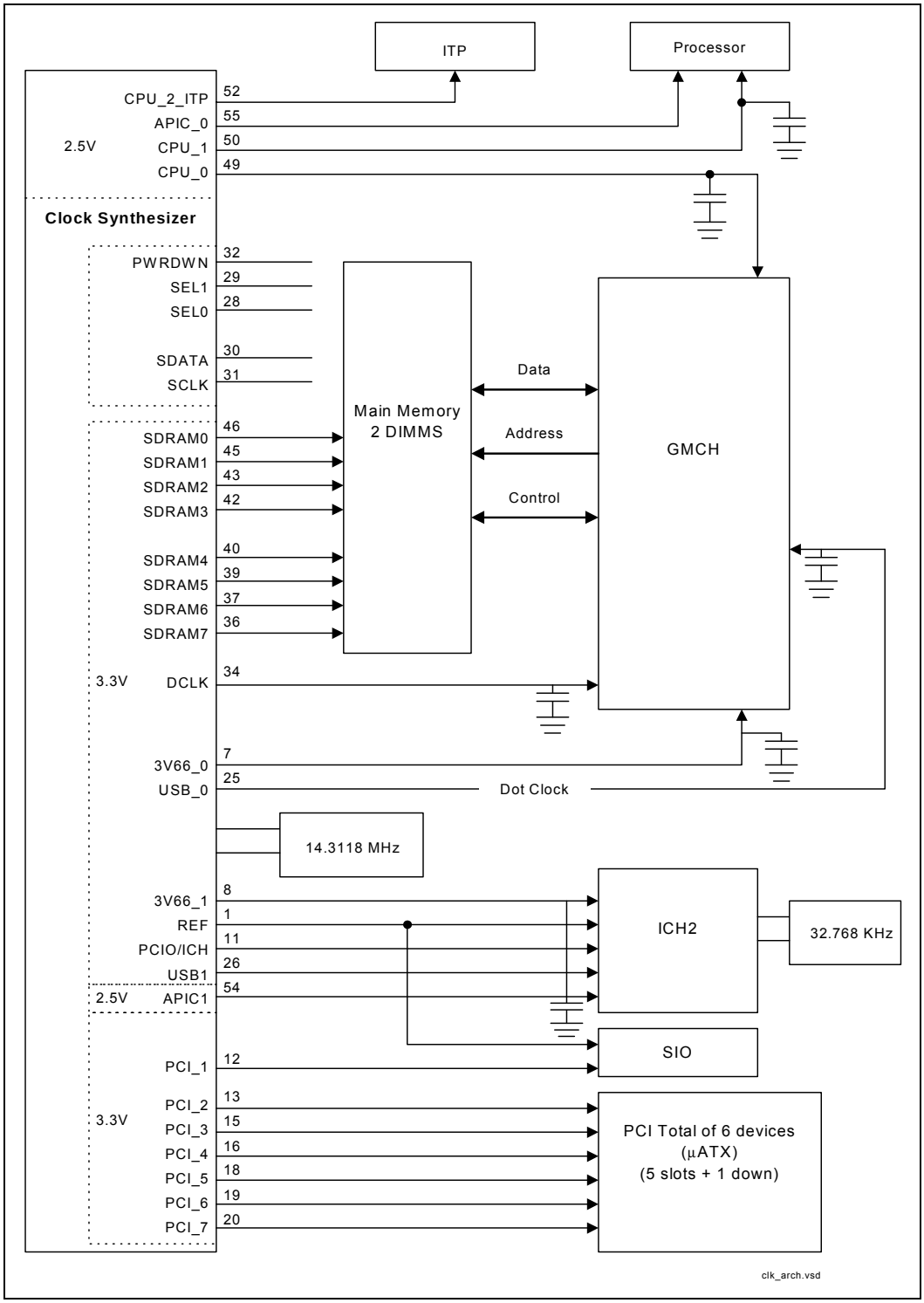
The DCLKREF signal from the external clock synthesizer to the GMCH is a 48 MHz signal. This signal has no length requirements, except those specified in the Design Guide. However, care in routing this signal relative to the DIMM slots is important. Future board designs should attempt to route the DCLKREF trace so that the trace is not parallel to the DIMM slots or does not pass underneath the DIMM slots. This prevents noise coupling of memory-related signals into the 48 MHz clock signal.

Features (56 Pin SSOP Package)

- Three copies of processor clock 66/100/133 MHz (2.5V) (Processor, GMCH, ITP)
- Nine copies of 100 MHz (all the time) SDRAM clock (3.3V) (SDRAM[0:7], DClk)
- Eight copies of PCI clock (33 MHz) (3.3V)
- Two copies of APIC clock @16.67 MHz or 33 MHz, synchronous to processor clock (2.5V)
- Two copy of 48 MHz clock (3.3V) [Non SSC]
- Two copies of 3V66 MHz clock (3.3V)
- One copy of REF clock @14.31818 MHz (3.3V) also used as input strap to determine APIC frequency
- 66/100/133 MHz processor operation (selectable at power up only)
- Ref. 14.31818 MHz Xtal oscillator input
- Power Down Pin
- Spread spectrum support
- I^2C Support for turning off unused clocks

7.2 Clock Architecture

Figure 89. Intel® 810ET2 Chipset Clock Architecture



7.3 Clock Routing Guidelines

Table 39 shows the group skew and jitter limits.

Table 39. Group Skew and Jitter Limits at the Pins of the Clock Chip

Signal Group	Pin-Pin Skew	Cycle-Cycle Jitter	Nominal Vdd	Skew, Jitter Measure Point
Processor	175 ps	250 pS	2.5V	1.25V
SDRAM	250 ps	250 pS	3.3V	1.50V
APIC	250 ps	500 pS	2.5V	1.25V
48 MHz	250 ps	500 pS	3.3V	1.50V
3V66	175 ps	500 pS	3.3V	1.50V
PCI	500 ps	500 pS	3.3V	1.50V
REF	N/A	1000 pS	3.3V	1.50V

Table 40 shows the signal group and resistor tolerance.

Table 40. Signal Group and Resistor

Signal Group	Resistor
Processor	33 Ω ± 5%
SDRAM	22 Ω ± 5%
DCLK	33 Ω ± 5%
3V66	22 Ω ± 5%
PCI	33 Ω ± 5%
TCLK	22 Ω ± 5%
OCLK/RCLK	33 Ω ± 5%
48 MHz	33 Ω ± 5%
APIC	33 Ω ± 5%
REF	10 Ω ± 5%

Table 41 shows the layout dimensions for the clock routing.

Note: All the clock signals must be routed on the same layer which reference to a ground plane.

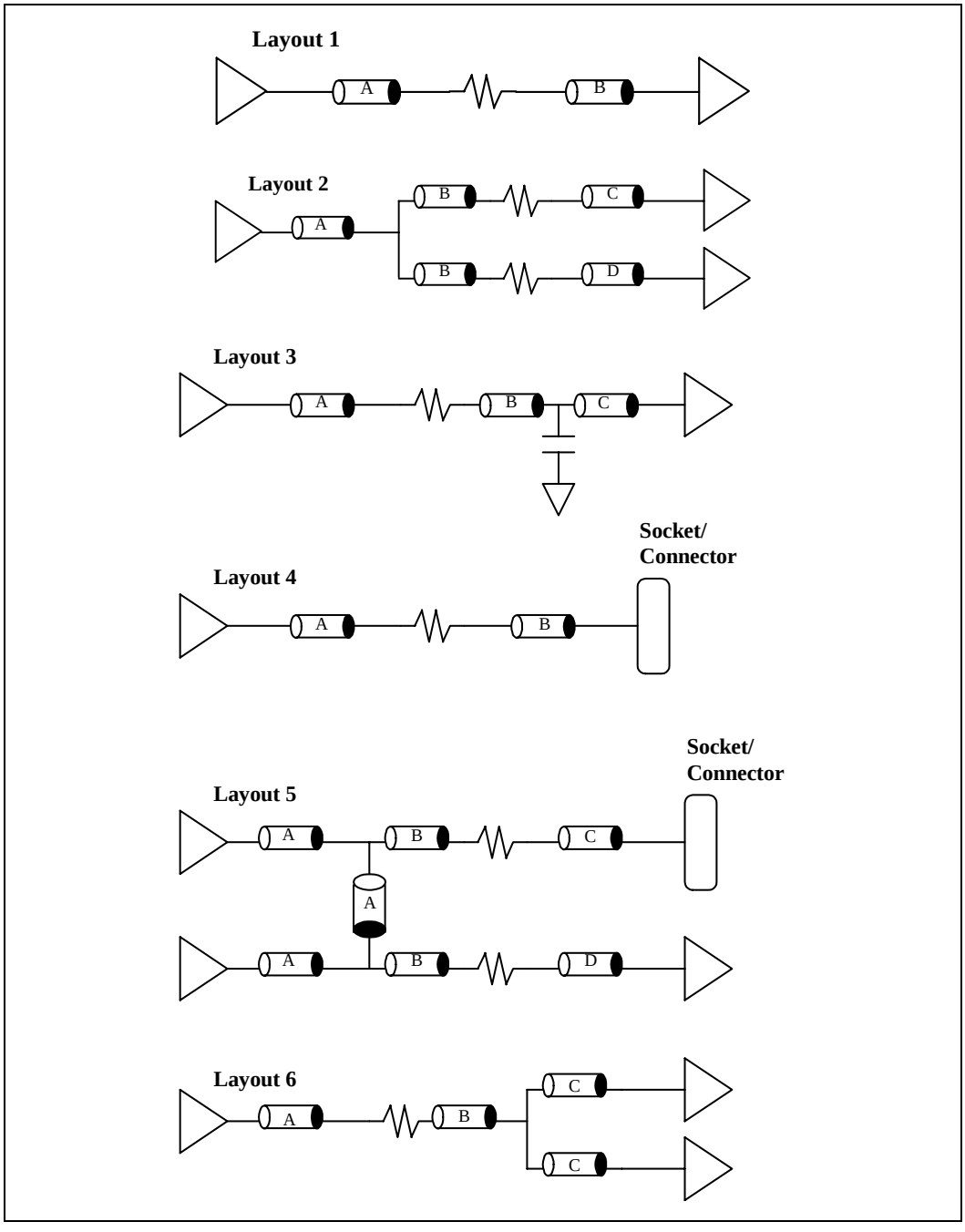
Table 41. Layout Dimensions

Group	Receiver	Resistor	Cap	Topology	A	B	C	D
MCLK	DIMM	22 Ω	N/A	Layout 1	0.5”	X	N/A	N/A
Processor: Intel® Pentium® III FC-PGA Processor 100/133 MHz	Segment C => Pentium III FC-PGA Processor Segment D => GMCH	33 Ω	N/A	Layout 5	0.1”	0.5”	X+4.8"	X+7.1”
Processor: Intel® Celeron® processor 66/100 MHz	Segment C => Celeron processor socket Segment D => GMCH	33 Ω	N/A	Layout 5	0.1”	0.5”	X+5.4"	X+7.1”
DCLK	GMCH	33 Ω	22 pF	Layout 3	0.5”	X+3.2”	0.5”	N/A
3V66	GMCH	22 Ω	18 pF	Layout 3	0.5”	X+1.4”	0.5”	N/A
3V66	Intel® ICH2	22 Ω	18 pF	Layout 3	0.5”	X+1.4”	0.5”	N/A
PCI	PCI device	33 Ω	N/A	Layout 1	0.5”	X+3.0” to X+9.3”	N/A	N/A
PCI	PCI socket	33 Ω	N/A	Layout 4	0.5”	X+0.0” to X+6.0”	N/A	N/A
PCI	ICH2	33 Ω	N/A	Layout 1	0.5”	X+4.4”	N/A	N/A
TCLK	SDRAM	22 Ω	N/A	Layout 6	0.5”	1.5” to 2.5”	0.75” to 1.25”	N/A
OCLK/RCLK	GMCH	33 Ω	N/A	Layout 1	0.5”	3.25” to 3.75”	N/A	N/A
APIC	PPGA	33 Ω	N/A	Layout 4	0.5”	Y	N/A	N/A
APIC	ICH2	33 Ω	N/A	Layout 1	0.5”	Y+2.4”	N/A	N/A

NOTES:
1. W, X, Y and Z trace lengths are arbitrary. Below are some suggested values: X=5.0 inches, Y=4.2 inches.

Figure 90 shows the different topologies used for the clock routing guidelines.

Figure 90. Different Topologies for the Clock Routing Guidelines

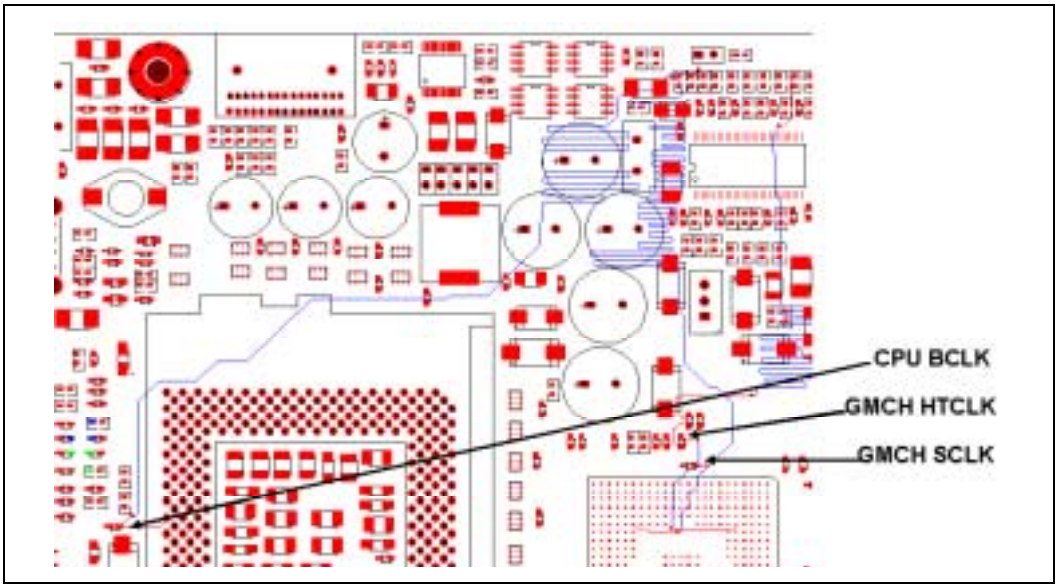


7.4 Capacitor Sites

Intel recommends 0603 package capacitor sites placed as close as possible to the clock input receivers for AC tuning for the following signal groups:

- GMCH
- Processor
- SDRAM/DCLK
- 3V66
- 3V66 to the ICH2

Figure 91. Example of Capacitor Placement Near Clock Input Receiver



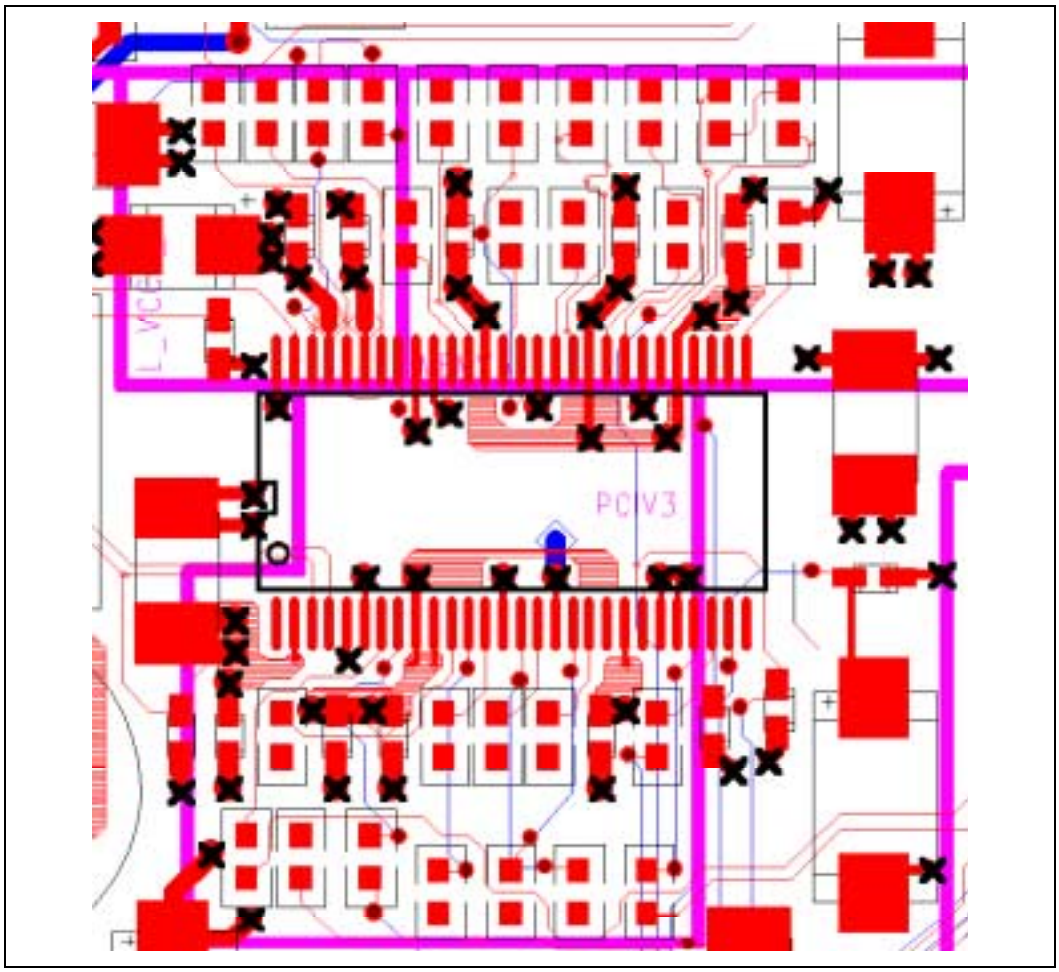
7.5 Clock Power Decoupling Guidelines

Several general layout guidelines should be followed when laying out the power planes for the CK810E clock generator.

- Isolate power planes to the each of the clock groups.
- Place local decoupling as close to power pins as possible 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 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 24–26 mil path. An example power via is an 18 mil finished hole with a 33–38 mil path. For large decoupling or power planes with large current transients it is recommended to use a larger power via.

An example of clock power layout is presented in Figure 92.

Figure 92. Example of Clock Power Plane Splits and Decoupling



7.6 Clock Skew Requirements

To ensure correct system functionality, certain clocks must maintain a skew relationship to other clocks as summarized in Section 7.6.1.

7.6.1 IntraGroup Skew Limits

Clocks within each group must maintain appropriate skew relationship to each other. These requirements are summarized in Table 42.

Table 42. Clock Skew Requirements

Group Pair	Skew Limit	Measurement Point of Receiver
Processor BCLK to GMCH HTCLK	350 ps window	Pin on top of PPGA PKG GMCH Ball
GMCH SCLK to DIMM Clocks	±630 ps Referenced to GMCH SCLK	GMCH Ball DRAM Component Pin on Module
GMCH HubCLK to Intel® ICH2 HubCLK	575 ps window	GMCH Ball ICH Ball

This page is intentionally left blank.

8 Power Delivery

This chapter contains power delivery guidelines. Table 43 provides definitions for power delivery terms used in this chapter.

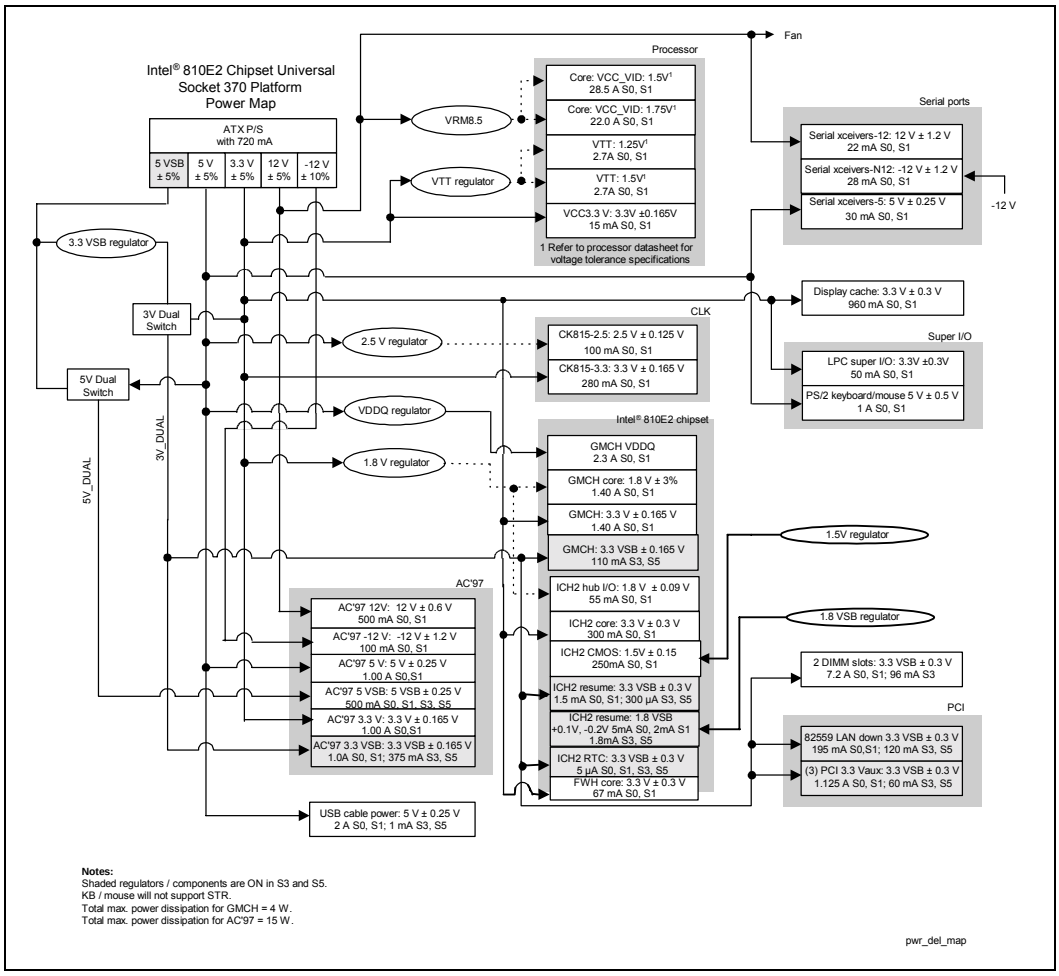
Table 43. Power Delivery Definitions

Term	Description
Suspend-To-RAM (STR)	In the STR state, the system state is stored in main memory and all unnecessary system logic is turned off. Only main memory and logic required to wake the system remain powered. This state is used in the Customer Reference Board (CRB) to satisfy the S3 ACPI power management state.
Full-power operation	During full-power operation, all components on the motherboard remain powered. Note that full-power operation includes both the full-on operating state and the S1 (CPU stop-grant state) state.
Suspend operation	During suspend operation, power is removed from some components on the motherboard. The CRB supports two suspend states: Suspend-to-RAM (S3) and Soft-off (S5).
Power rails	An ATX power supply has 6 power rails: +5V, -5V, +12V, -12V, +3.3V, 5VSB. In addition to these power rails, several other power rails are created with voltage regulators on the CRB.
Core power rail	A power rail that is only on during full-power operation. These power rails are on when the PSON signal is asserted to the ATX power supply. The core power rails that are distributed directly from the ATX power supply are: ± 5V, ±12V and +3.3V.
Standby power rail	A power rail that in on during suspend operation (these rails are also on during full-power operation). These rails are on at all times (when the power supply is plugged into AC power). The only standby power rail that is distributed directly from the ATX power supply is: 5VSB (5V Standby). There are other standby rails that are created with voltage regulators on the motherboard.
Derived power rail	A derived power rail is any power rail that is generated from another power rail. For example, 3.3VSB is usually derived (on the motherboard) from 5VSB using a voltage regulator (on the CRB, 3.3VSB is derived from 5V_DUAL).
Dual power rail	A dual power rail is derived from different rails at different times (depending on the power state of the system). Usually, a dual power rail is derived from a standby supply during suspend operation and derived from a core supply during full-power operation. Note that the voltage on a dual power rail may be misleading.

Figure 93 shows the power delivery architecture for an example 810E2 chipset universal platform-based system. 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 93. Power Delivery Map



In addition to the power planes provided by the ATX power supply, an **instantly available** 810E2 chipset for use with universal socket 370 platform (using Suspend-to-RAM) requires six power planes to be generated on the board. The requirements for each power plane are documented in this section. In addition to on-board voltage regulators, the CRB will have a 5V Dual Switch.

5V Dual Switch

This switch will power the 5V Dual plane from the 5V core ATX supply during full-power operation. During Suspend-to-RAM, the 5V Dual plane will be powered from the 5V Standby power supply. Note: the voltage on the 5V Dual plane **is not 5V!** There is a resistive drop through the 5V Dual Switch that must be considered. Therefore, NO COMPONENTS should be connected directly to the 5V Dual plane. On the CRB, the only devices connected to the 5V Dual plane are voltage regulators (to regulate to lower voltages).

Note: This switch is not required in an Intel 810E2 chipset for use with universal socket 370 platform that does not support Suspend-to-RAM (STR).

VTT

This power plane is used to power the AGTL/AGTL+ termination resistors. Refer to the latest revisions of the Pentium III processor (CPUID=068xh) and Celeron processor (CPUID=068xh) datasheets.

Note: This regulator is required in ALL designs.

1.8V

The 1.8V plane powers the GMCH core and the ICH2 hub interface I/O buffers. This power plane has a total power requirement of approximately 1.7A. The 1.8V plane should be decoupled with a 0.1 µF and a 0.01 µF chip capacitor at *each* corner of the GMCH and with a single 1 µF and 0.1 µF capacitor at the ICH2.

Note: This regulator is required in ALL designs.

VDDQ

The VDDQ plane is used to power the GMCH AGP interface and the graphics component AGP interface. Refer to the AGP Interface Specification Revision 2.0 (http://www.agpforum.org) and ECR#43 and ECR#44 for specific VDDQ delivery requirements.

For the consideration of component long-term reliability, the following power sequence is strongly recommended while the AGP interface of GMCH is running at 3.3V. If the AGP interface is running at 1.5V, the following power sequence recommendation is no longer applicable. The power sequence recommendations are:

- During the power-up sequence, the 1.8V must ramp up to 1.0V **Before** 3.3V ramps up to 2.2V
- During the powerdown sequence, the 1.8V CAN NOT ramp below 1.0V **Before** 3.3V ramps below 2.2V
- The same power sequence recommendation also applies to the entrance and exit of S3 state, since MCH power is compete off during the S3 state.

Refer to Section 8.3.2 for more information on the power ramp sequence requirement between 3.3V and 1.8V. System designers need to be aware of this requirement while designing the voltage regulators and selecting the power supply. For further details on the voltage sequencing

requirements, refer to the *Intel® 810 Chipset Family: 82810 Graphics and Memory Controller Hub (GMCH) Datasheet*.

3.3VSB

The 3.3VSB plane powers the I/O buffers in the resume well of the ICH2 and the PCI 3.3Vaux suspend power pins. The 3.3Vaux requirement state that during suspend, the system must deliver 375 mA to each *wake-enabled* card and 20 mA to each *non wake-enabled* card. During *full-power* operation, the system must be able to supply 375 mA to each card. Therefore, the total current requirement is:

- *Full-power Operation:* 375 mA * number of PCI slots
- *Suspend Operation:* 375 mA + 20 mA * (number of PCI slots – 1)

In addition to the PCI 3.3Vaux, the ICH2 suspend well power requirements must be considered as shown in Figure 93.

Note: This regulator is required in ALL designs.

1.8VSB

The 1.8VSB plane powers the logic to the resume well of the ICH2. This should not be used for VCMOS.

8.1 Thermal Design Power

The 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 82810E2 GMCH component is 5.1W.

The TDP of the 82801BA ICH2 is 1.5 W ±15%.

8.1.1 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:

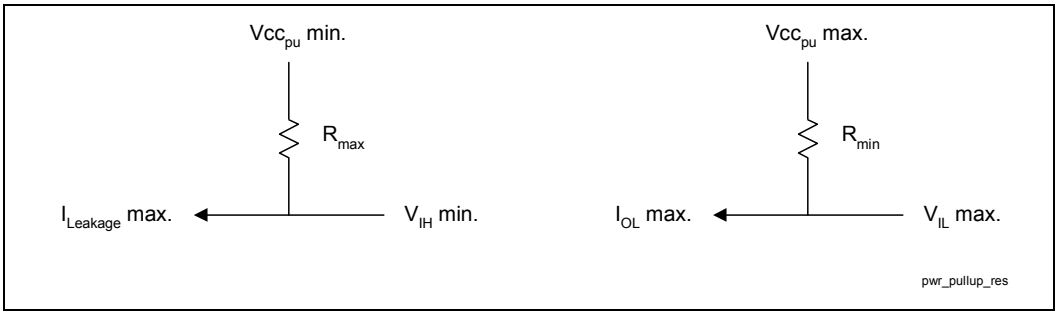
$$R_{MAX} = (Vcc_{PU}\ MIN - V_{IH}\ MIN) / I_{LEAKAGE}\ MAX$$

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

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 94. Pull-Up Resistor Example



8.2 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 powerdown, 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.

8.3 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 ATXPWROK 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.
- PWROK 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.

8.3.1 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.

8.3.2 1.8V/3.3V Power Sequencing

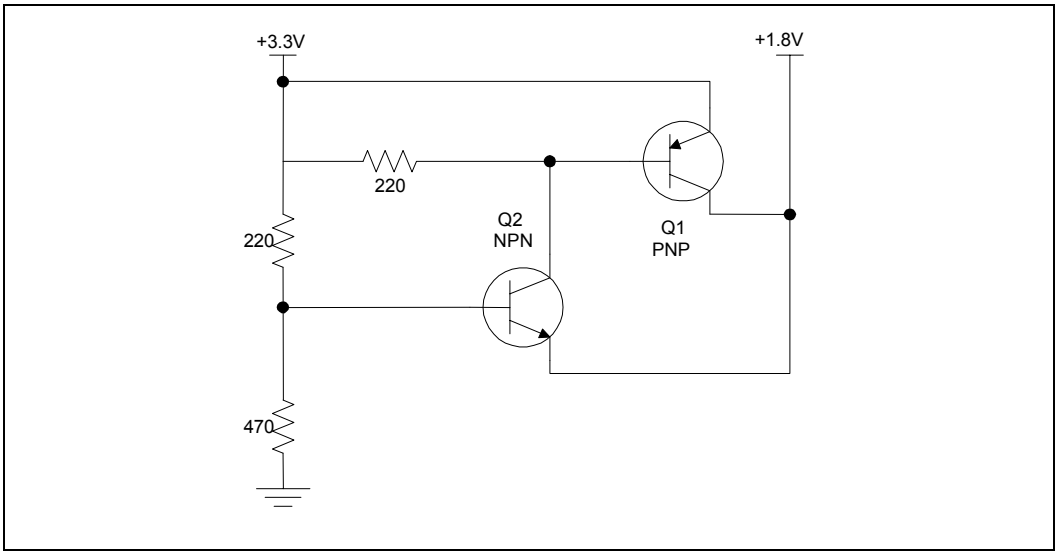
The ICH2 has two pairs of associated 1.8V and 3.3V supplies. These are {VCC1.8, VCC3.3} and {VCCSus1_8, VCCSus3_3}. These pairs are assumed to power up and power down together. **The difference between the two associated supplies must never be greater than 2.0V.** The 1.85V supply may come up before 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.

Figure 95 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 95. 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

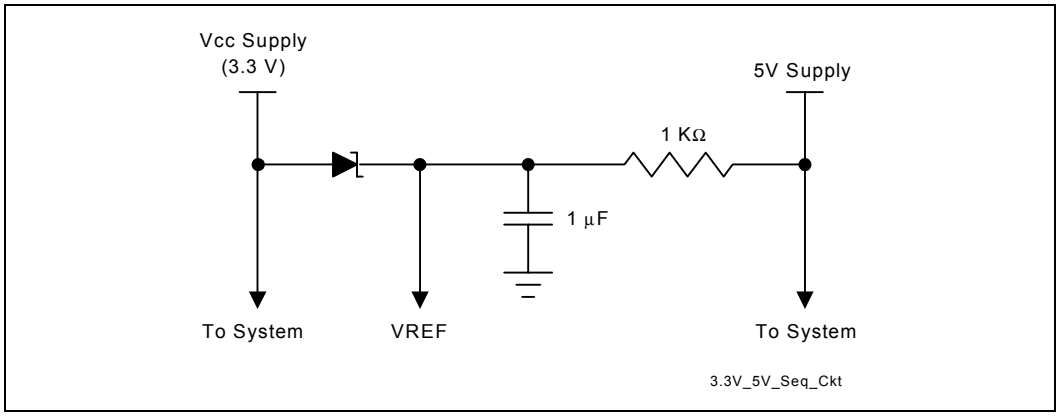
If one of these signals goes high while one of its associated power planes is active and the other is not, a leakage path will exist between the active and inactive power wells. This could result in high, possibly damaging, internal currents.

8.3.3 3.3V/V5REF Sequencing

V5REF is the reference voltage for 5V tolerance on inputs to the ICH2. V5REF must be powered up before or simultaneously to VCC3.3. It must also power down after or simultaneous to VCC3.3. 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 96 shows a sample implementation of how to satisfy the V5REF/3.3V sequencing rule.

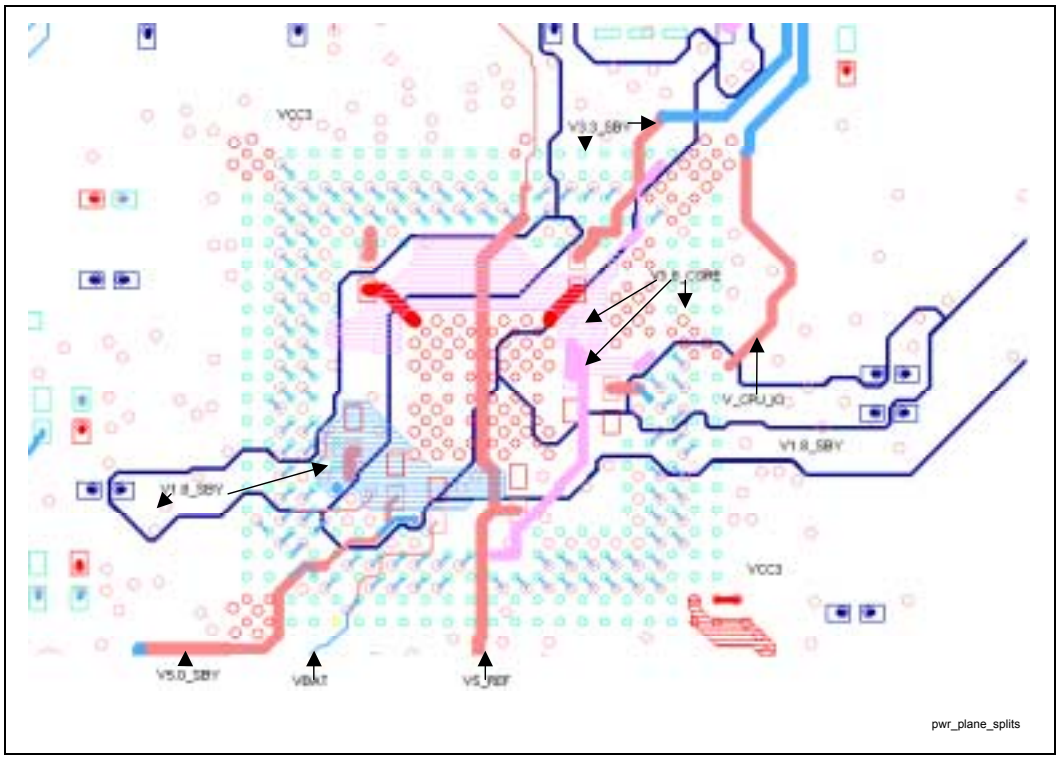
As an additional consideration, during suspend, the only signals that are 5V tolerant capable are USB OC:[3:0]#. If these signals are not needed during suspend, V5REF_SUS can be connected to either VccSus3_3 or 5V_Always/5V_AUX. If OC:[3:0]# is needed during suspend and 5V tolerance is required then V5REF_SUS should be connected to 5V_Always/5V_AUX, but if 5V tolerance is not needed in suspend, then V5REF_SUS can be connected to either VccSus3_3 or 5V_Always/5V_AUX rails.

Figure 96. 3.3V/V5REF Sequencing Circuitry



8.4 Power Plane Splits

Figure 97. Power Plane Split Example



8.5 Power_Supply PS_ON Considerations

- If a pulse on SLP_S3# or SLP_S5# is short enough (~ 10-100mS) 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.

9 Design Checklist

9.1 Design Review Checklist

This checklist highlights design considerations that should be reviewed prior to manufacturing a motherboard that implements an 810E2 chipset universal socket 370 platform. This is not a complete list and does not guarantee that a design will function properly. Beyond the items contained in the following text, refer to the most recent version of the Design Guide for more detailed instructions on designing a motherboard.

9.1.1 Design Checklist Summary

The following tables provide 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 for questions or issues regarding interpretation of the information contained in these tables.

Table 44. AGTL+ Connectivity Checklist for 370-Pin Socket Processors

Processor Pin	I/O	Recommendations
A[35:3]# [1]	I/O	• Connect A[31:3]# to GMCH. Leave A[35:32]# as No Connect (not supported by chipset).
ADS# [1]	I/O	• Connect to GMCH.
AERR#	I/O	• Leave as No Connect (not supported by chipset).
AP[1:0]#	I/O	• Leave as No Connect (not supported by chipset).
BERR#	I/O	• Leave as No Connect (not supported by chipset).
BINIT#	I/O	• Leave as No Connect (not supported by chipset).
BNR# [1]	I/O	• Connect to GMCH.
BP[3:2]#	I/O	• Leave as No Connect.
BPM[1:0]	I/O	• Leave as No Connect.
BPRI# [1]	I	• Connect to GMCH.
BREQ[0]# (BR0#)	I/O	• 10 Ω pull-down resistor to ground.
D[63:0]# [1]	I/O	• Connect to GMCH.
DBSY# [1]	I/O	• Connect to GMCH.
DEFER# [1]	I	• Connect to GMCH.
DEP[7:0]#	I/O	• Leave as No Connect (not supported by chipset).
DRDY# [1]	I/O	• Connect to GMCH.
HIT# [1]	I/O	• Connect to GMCH.

Processor Pin	I/O	Recommendations
HITM# [1]	I/O	• Connect to GMCH.
LOCK# [1]	I/O	• Connect to GMCH.
REQ[4:0]# [1]	I/O	• Connect to GMCH.
RESET#	I	• 56 Ω pull-up resistor to VTT, connect to GMCH, 240 Ω series resistor to ITP.
RESET2# [2]	I	• Driven by same signal as RESET#.
RP#	I/O	• Leave as No Connect (not supported by chipset).
RS[2:0]#	I	• Connect to GMCH.
RSP#	I	• Leave as No Connect (not supported by chipset).
TRDY# [1]	I	• Connect to GMCH.

Table 45. CMOS Connectivity Checklist for 370-Pin Socket Processors

Processor Pin	I/O	Recommendations
A20M#	I	• Connect to Intel® ICH2.
FERR#	O	• 150 Ω pull-up resistor to VCC_{CMOS} / Connect to ICH2.
FLUSH#	I	• 150 Ω pull-up resistor to VCC_{CMOS} (not used by chipset).
IERR#	O	• 150 Ω pull-up resistor to VCC_{CMOS} if tied to custom logic or leave as No Connect (not used by chipset).
IGNNE#	I	• Connect to ICH2.
INIT#	I	• Connect to ICH2 & FWH Flash BIOS.
LINT0/INTR	I	• Connect to ICH2.
LINT1/NMI	I	• Connect to ICH2.
PICD[1:0]	I/O	• 150 Ω pull-up resistor to VCC_{CMOS} / Connect to ICH2.
PREQ#	O	• ~200–330 Ω pull-up resistor to VCC_{CMOS} / Connect to ITP.
PWRGOOD	I	• 150–330 Ω pull-up to 2.5V, output from the PWRGOOD logic.
SLP#	I	• Connect to ICH2.
SMI#	I	• Connect to ICH2.
STPCLK#	I	• Connect to ICH2.
THERMTRIP#	O	• 150 Ω pull-up resistor to VCC_{CMOS} and connect to power off logic, or leave as No Connect.

Table 46. TAP Checklist for a 370-Pin Socket Processor

Processor Pin	I/O	Recommendations
TCK	I	• 39 Ω pull-down resistor to ground/ Connect to ITP.
TDI	I	• 200–330 Ω pull-up resistor to VCC_{CMOS} / Connect to ITP.
TDO	O	• 150 Ω pull-up resistor to VCC_{CMOS} / Connect to ITP.
TMS	I	• 39 Ω pull-up resistor to VCC_{CMOS} / 47 Ω series resistor to ITP.
TRST#	I	• 500-680 Ω pull-down resistor to ground / Connect to ITP.
PRDY#	I	• 150 Ω pull-up resistor to VTT / 240 Ω series resistor to ITP.

NOTES:
1. The ITP connector is different than the one previously specified for other Intel IA-32 processors. It is the female counterpart to the previously specified connector and is specifically for use with processors utilizing 1.5V CMOS TAP I/O signals.
2. The Pentium III processor requires an ITP with a 1.5V tolerant buffer board. Previous ITPs are designed to work higher voltages and may damage the processor if they are connected to a Pentium III processor.

Table 47. Miscellaneous Checklist for 370-Pin Socket Processors

Processor Pin	I/O	Recommendations
BCLK	I	• 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 GMCH and processor.
BSEL0	I/O	• **Case 1, 66/100/133 MHz support:** 1 kΩ pull-up resistor to 3.3V, connect to CK810E SEL0 input, connect to GMCH LMD29 pin via 10 kΩ series resistor. • **Case 2, 100/133 MHz support:** 1 kΩ pull-up resistor to 3.3V, connect to PWRGOOD logic such that a logic low on BSEL0 negates PWRGOOD.
BSEL1	I/O	• 1 kΩ pull-up resistor to 3.3V, connect to CK810E REF pin via 10 kΩ series resistor, connect to GMCH LMD13 pin via 10 kΩ series resistor.
CLKREF	I	• Connect to divider on VCC2.5 or VCC3.3 to create 1.25V 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 could be connected to PWRGOOD logic to gate system from powering on if no processor is present. If used, 1 kΩ–10 kΩ pull-up resistor to any voltage.
EDGCTRL	I	• 51 Ω ±5% pull-up resistor to VCC_{CORE}.
PICCLK	I	• Connect to clock generator / 22–33 Ω series resistor (though OEM needs to simulate based on driver characteristics).
PLL1, PLL2	I	• Low pass filter on VCC_{CORE} provided on motherboard. Typically a 4.7 uH inductor in series with VCC_{CORE} is connected to PLL1 then through a series 33 µF capacitor to PLL2.
RTTCTRL5[1] (S35)		• 110 Ω ±1% pull-down resistor to ground.
SLEWCTRL (E27)		• 110 Ω ±1% pull-down resistor to ground.

Processor Pin	I/O	Recommendations
THERMDN	O	• No Connect if not used; otherwise connect to thermal sensor using vendor guidelines.
THERMDP	I	• No Connect if not used; otherwise connect to thermal sensor using vendor guidelines.
VCC1.5	I	• Connected to same voltage source as VTT. Must have some high and low frequency decoupling.
VCC2.5	I	• Connected to 2.5V voltage source. Should have some high and low frequency decoupling.
VCC_{CMOS}	O	• 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}	I	• 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)	O	• For the Intel® Celeron® processor (CPUID=068xh), Intel® Pentium® III processor (CPUID=068xh), and future 0.13 micron socket 370 processors, $VCORE_{DET}$ must float.
VID[3:0]	O	• Connect to on-board VR or VRM. For on-board VR, 10 kΩ pull-up resistor to power-solution compatible voltage 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]	N/A	• Connect regulator controller pin to ground (not on processor).
VREF[7:0]	I	• Connect to VREF voltage divider made up of 75 Ω and 150 Ω 1% resistors connected to VTT. • Decoupling Guidelines: • Four ea. (min) 0.1 µF in 0603 package placed within 500 mils of VREF pins.
VTT	I	• Connect AH20, AK16, AL13, AL21, AN11, AN15, and G35 to 1.5V 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
Additional VTT	I	• AA33, AA35, AN21, E23, S33, S37, U35, U37
GND	N/A	• AJ3
No Connects	N/A	• The following pins must be left as no-connects: AK30, AM2, F10, G37, L33, N33, N35, N37, Q33, Q35, Q37, R2, W35, X2, Y1

Table 48. GMCH Checklist

Checklist Line Items	Comments
VCCDA	• VCCDA needs to be connected to an isolated power plane.
HCLK, SCLK	• 22 pF capacitor to ground as close as possible to GMCH.
GTLREFA, GTLREFB	• Refer to the latest design guide for the correct GTLREF generation circuit.
HUBREF	• Refer Section 6.5.4 for the correct HUBREF generation circuit. Also, place a 0.1 µF capacitor as close as possible to GMCH to ground.
IWASTE	• Tie to ground.
IREF	• Place a resistor as close as possible to GMCH and via straight to VSS plane. A 174 Ω 1% resistor is recommended.
LTVCL, LTVDA	• 10 kΩ (approximate) pull-up resistor to 3.3V if digital video out is not implemented.
LTCLK	• Series resistor 22 Ω ± 2%.
OCLK/RCLK	• Series resistor 33 Ω ± 2%.
LMD[27:31] Reset strapping options:	**Strapping options:** For a 1, use a 10 kΩ (approximate) pull-up resistor to 3.3V; a 0 is default (due to internal pull-down resistors). LMD31: 0 = Normal operation 1 = XOR TREE for testing purposes LMD30: 0 = Normal operation 1 = Tri-state mode for testing purposes (will tri-state all signals) LMD29: 0 = System bus frequency = 66 MHz 1 = System bus frequency = 100 MHz LMD28: The value on LMD28 sampled at the rising edge of CPURST# reflects if the IOQD (In-Order Queue Depth) is set to 1 or 4. 0 = IOQD = 4 1 = IOQD = 1 LMD27: The PMOS-Kicker should be “OFF” for either processor. This is accomplished by treating LMD27 the same as any other LMD pin. Do not externally pull it up or down. LMD26: **Set to socket** for Intel® Celeron® processors (CPUID=068xh) and Intel® Pentium® III processors (CPUID=068xh). **Set to slot** for Pentium III processors that use 0.13 micron technology. LMD13: 0= LMD29 determines host bus frequency 1= host bus frequency is 133 MHz
HCOMP	• **Option 1—RCOMP Method:** Tie the HCOMP pin to a 40 Ω 1% or 2% (or 39 Ω 1%) pull-up resistor to 1.8V via a 10 mil wide, very short (~0.5 inch) trace. • **Option 2—ZCOMP Method:** The HCOMP pin must be tied to a 10-mil trace that is AT LEAST 18 inches long. This trace must be un-terminated and care should be taken when routing the signal to avoid crosstalk (1520 mil separation between this signal and any adjacent signals is recommended). This signal may not cross power plane splits.

Table 49. System Memory Checklist

Checklist Line Items	Recommendations
Pin 147	• Connect to Ground (since the Intel® 810E2 chipset does not support registered DIMMs).
WP (Pin 81 on the DIMMs)	• Add a 4.7 kΩ pull-up resistor to 3.3V. This is a recommendation to write-protect the DIMM's EEPROM.
MAA[7:4], MAB[7:4]	• Add 10 Ω series resistors to the MAA[7:4], MAB[7:4] as close as possible to GMCH for signal integrity.

Table 50. Display Cache Checklist

Checklist Line Items	Recommendations
CKE	• 4.7 kΩ pull-up resistor to VCC3.

9.2 Intel® ICH2 Checklist

9.2.1 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 Ω 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. This signal has integrated pull-up resistors of 24 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.

9.2.2 Hub Interface

Checklist Items	Recommendations
HL[11]	• No pull-up resistor required. Use a no-stuff or a test point to put the Intel® ICH2 into NAND chain mode testing
HL_COMP	• Tie the COMP pin to a 40 Ω 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.

9.2.3 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 inch to 9 inch (0.5 inch to 3 inches on card)	To meet timing requirements.
3	• Stubs due to R-pak CNR/LOM stuffing option should not be present.	To minimize inductance.
4	• Maximum Trace Lengths: ICH2 to Intel® 82562EH: L = 4.5 inches to 10 inches; Intel® 82562ET: L = 3.5 inches to 10 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 inch (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 s.	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-degree bend is required, it is recommended to use two 45-degree 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.

Checklist Items	Recommendations	Comments
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, processor, 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.
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 capacitor (~470 pF) on each of the four LED pins. Reduces emissions attributed to LAN subsystem.

9.2.4 EEPROM Interface

Checklist Items	Recommendations
EE_DOUT	• Prototype Boards should include a placeholder for a pull-down 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 an 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).

9.2.5 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.

9.2.6 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 of the Intel® ICH2 datasheet. 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 of the ICH2 datasheet. 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	• Intel® Pentium® 4 processor-based systems: These processors do not have APIC pins so all platforms using this processor should both tie APICCLK to ground and tie APICD:[1:0] to ground via a 1K-10K pull-down resistor. • Non-Pentium 4 processor-based systems: If the APIC is used: - 150 Ω pull-up resistors on APICD[1:0] - Connect APICCLK to CK133 with a 20-33Ω 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[1:0] to GND through 10 kΩ pull-down resistors.

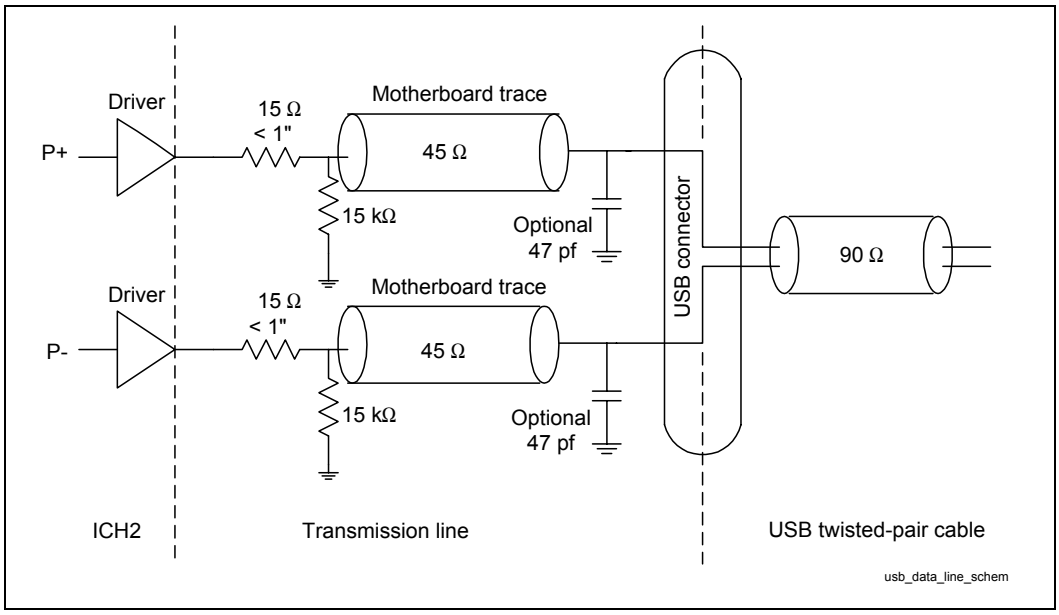
9.2.7 GPIO Checklist

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 REQ5#.
[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.

9.2.8 USB

Checklist Items	Recommendations
USBP[3:0]P USBP[3:0]N	• See Figure 98 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 98. USB Data Line Schematic



9.2.9 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 24 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#	• This signal should be connected 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. Can be tied to RSMPWROK on Desktop Platforms.

9.2.10 Processor Signals

Checklist Items	Recommendations
A20M#, CPUSLP#, IGNNE#, INIT#, INTR, NMI, SMI#, STPCLK#	• Internal circuitry has been added to the Intel® 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.

9.2.11 System Management

Checklist Items	Recommendations
SMBDATA SMBCLK	• Requires external pull-up resistors. See Section 6.17 to determine the appropriate power well to use to tie the pull-up 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 Section 6.17 to determine the appropriate power well to use to tie the pull-up 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.

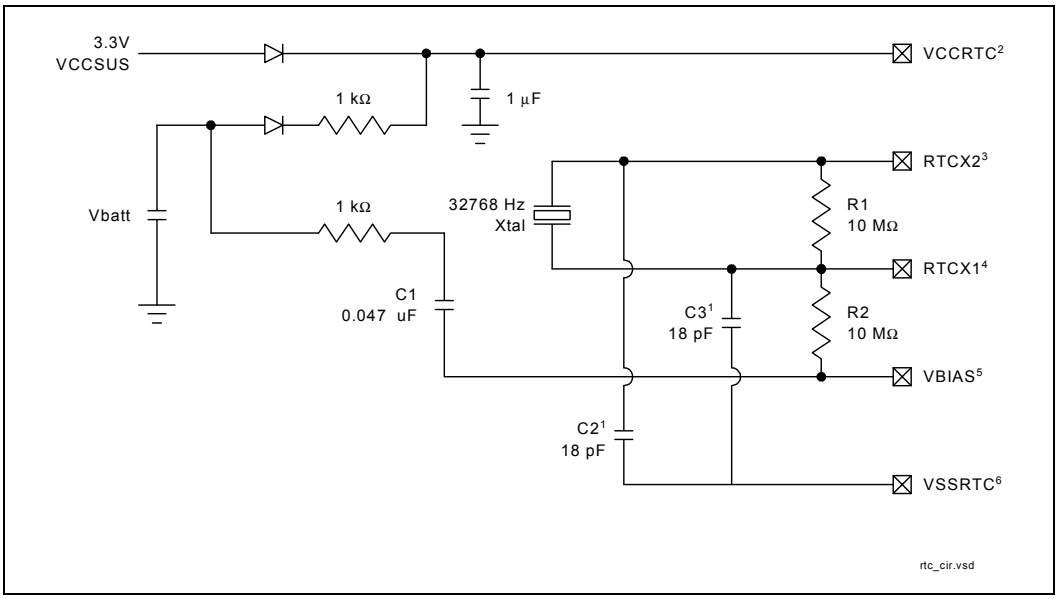
9.2.12 ISA Bridge Checklist

Checklist Items	Recommendations
Intel® ICH2 GPO[21] / MISA NOGO input	• Connect 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.

9.2.13 RTC

Checklist Items	Recommendations
VBIAS	• The VBIAS pin of the Intel® ICH2 is connected to a .047 µF capacitor. See Figure 99
RTCX1 RTCX2	• Connect a 32.768 kHz crystal oscillator across these pins with a 10 MΩ resistor and use 18 pF decoupling capacitors 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 99 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 10–20 ms RC delay (8.2 kΩ & 2.2 µF) See Figure 99.
SUSCLK	• To assist in RTC circuit debug, route SUSCLK to a test point if it is unused

Figure 99. Intel® ICH2 RTC Oscillator Circuitry



NOTES:
1. The exact capacitor value must be based on the crystal maker's recommendation. (The typical value for C2 and C3 is 18 pF for a crystal with Cload = 12.5pF)
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

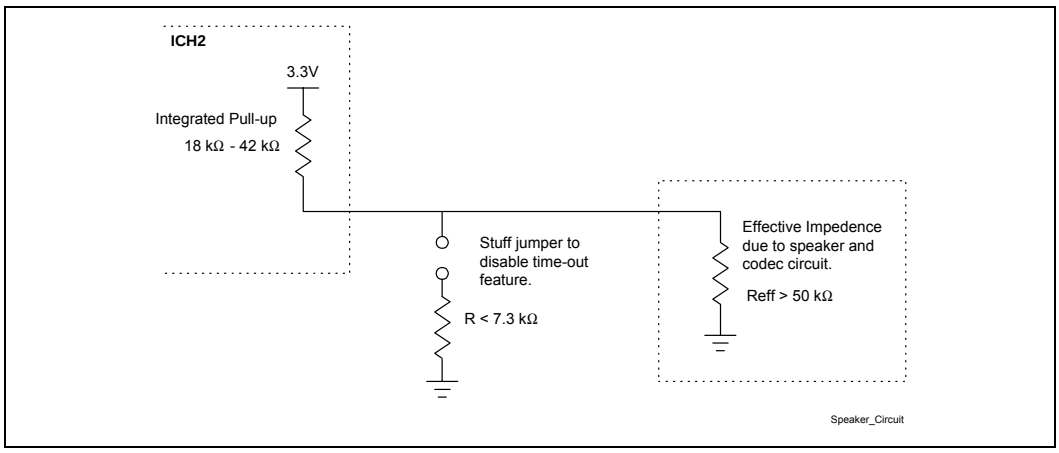
9.2.14 AC'97

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_O AC97 = 60 Ω ± 15%
	• 5 mil trace width, 5 mil spacing between traces
	• Max Trace Length ICH2/Codec/CNR = 14 inches

9.2.15 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 Figure 100.
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.

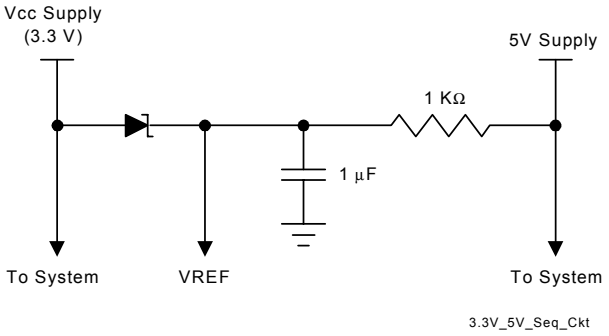
Figure 100. SPKR Circuitry



9.2.16 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 capacitor.
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 capacitors
VCCSus3.3	• Requires one 0.1 µF decoupling capacitor.
VCC1.8	• Requires two 0.1 µF decoupling capacitors.
VCCSus1.8	• Requires one 0.1 µF decoupling capacitor.
5VREF_SUS	• Requires one 0.1 µF decoupling capacitor. • V5REF_SUS only affects 5V-tolerance for USB OC:[3:0]# pins and can be connected to either VccSUS3_3 or 5V_Always/5V_AUX if 5V tolerance on these OC:[3:0]# is not needed. If 5V tolerance on OC:[3:0]# is needed then V5REF_SUS USB must be connected to 5V_Always/5V_AUX which remains powered during S5.
5VREF	• 5VREF is the reference voltage for 5V tolerant inputs in the Intel® ICH2. Tie to pins VREF[2:1]. 5VREF must power up before or simultaneous to VCC3.3. It must power down after or simultaneous to VCC3.3. Refer to Figure 101 for an example circuit schematic that may be used to ensure the proper 5VREF sequencing.

Figure 101. V5REF Circuitry



9.2.17 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 Intel® 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

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

Figure 102. Host/Device Side Detection Circuitry

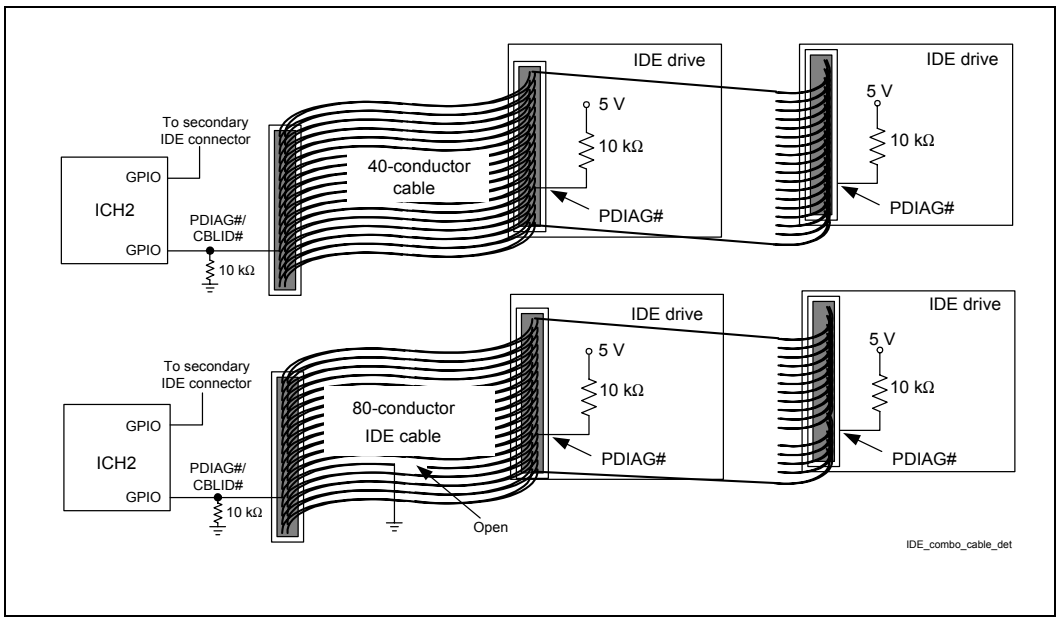
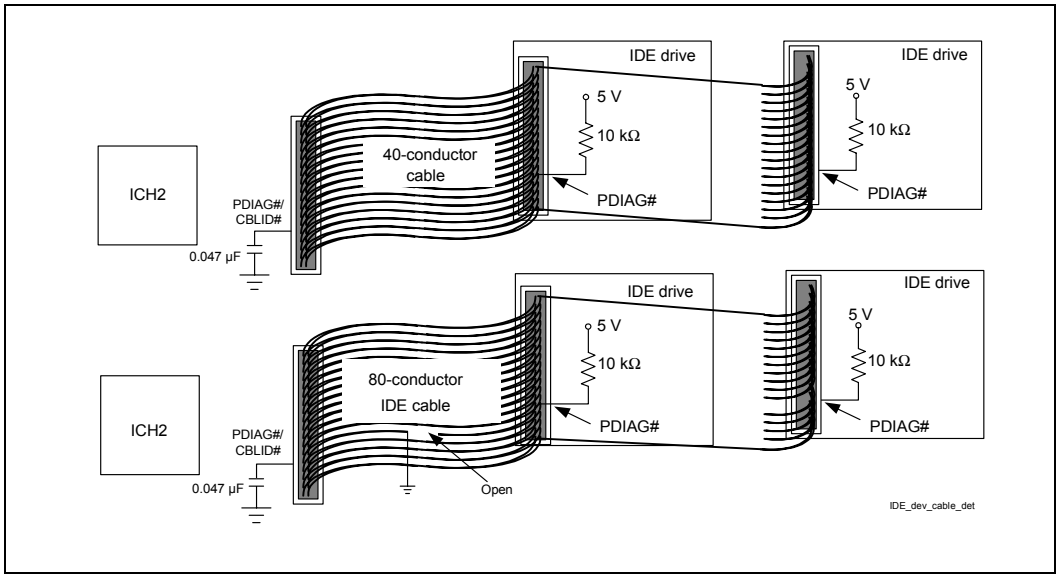


Figure 103. Device Side Only Cable Detection



9.3 LPC Checklist

Checklist Items	Recommendations
RCIN#	• Pull up through 8.2 kΩ resistor to VCC3.3.
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 capacitor)
J1BUTTON1, JPBUTTON2, J2BUTTON1, J2BUTTON2	• Pull up through 1 kΩ resistor to VCC5. Decouple through 47 pF capacitor 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 pF capacitor 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 capacitor to GND.
L_SMBD	• Pass through 150 Ω resistor to 82559.
SLCT#, PE, BUSY, ACK#, ERROR#	• Pull up through 2.2 kΩ resistor to VCC5_DB25_DR. • Decouple through 180 pF capacitor 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 capacitor to GND.
PWM1, PWM2	• Pull up to 4.7 kΩ to VCC3.3 and connected to jumper for decouple with 0.1 µF capacitor 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 capacitor to GND.
SYSOPT	• Pull down with 4.7 kΩ resistor to GND or IO address of 02Eh.

9.4 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 capacitor 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 capacitor to GND
FP_PD	• Decouple through a 470 pF capacitor to GND. • Pull up 470 Ω to VCC5.
PWM1, PWM2	• Pull up through a 4.7 kΩ resistor to VCC3.3.

9.5 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 capacitors 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.

9.6 Clock Synthesizer Checklist

Checklist Items	Recommendations
REFCLK	• Connects to R-RefCLK, USB_CLK, SIO_CLK14, and ICHCLK14.
ICH_3V66/3V66_0, DOTCLK	• Passes through 33 Ω resistor. • When signal is input for Intel® ICH2, it is pulled down through a 18 pF capacitor to GND.
DCLK/DCLK_WR	• Passes through 33 Ω resistor. • When signal is input for GMCH, it is pulled down through a 22 pF capacitor to GND.
CPUHCLK/CPU_0_1	• Passes through 33 Ω resistor. • When signal is input for 370PGA, decouple through a 18 pF capacitor 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 capacitor 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.
L_VCC2_5	• Connects to VDD2_5[0…1] through ferrite bead to VCC2.5.
GMCHHCLK/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, GMCH_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, SCLK	• Pass through 22 Ω resistor.

9.7 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 this design guide.
PLL2	• See this design guide.

9.8 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 1.8 V and be between 1.71 V to 1.89 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.

10 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 810ET2 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.

Table 51. Super I/O

Vendors	Contact	Phone
SMC	Dave Jenoff	(909) 244-4937
National	Robert Reneau	(408) 721-2981
ITE	Don Gardenhire	(512)388-7880
Winbond	James Chen	(02) 27190505 - Taipei office

Table 52. Clock Generation

Vendors	Contact	Phone
Cypress	John Wunner	206-821-9202 x325
ICS	Raju Shah	408-925-9493
IC Works	Jeff Keip	408-922-0202, x1185
IMI	Elie Ayache	408-263-6300, x235
PERICOM	Ken Buntaran	408-435-1000

Table 53. Memory Vendors

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

Table 54. 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

Table 55. Flat Panel

Vendors	Contact	Phone
Silicon Images Inc	John Nelson	408-873-3111

Table 56. AC'97

Vendors	Contact	Phone
Analog Devices	Dave Babicz	781-461-3237
AKM	George Hill	408-436-8580
Cirrus Logic (Crystal)	David Crowell	512-912-3587
Creative Technologies Ltd./ Ensoniq Corp.	Steve Erickson	408-428-6600 x6945
Diamond Multimedia Systems	Theresa Leonard	360-604-1478
ESS Technology	Bill Windsor	510-492-1708
Euphonics, Inc.	David Taylor	408-554-7201
IC Ensemble Inc.	Steve Allen	408-969-0888 x106
Motorola	Pat Casey	508-261-4649
PCTel, Inc.	Steve Manuel	410-965-2172
Conexant (formerly Rockwell)	Tom Eichenberg	949-221-4164
SigmaTel	Spence Jackson Arron Lyman	512-343-6636 512-343-6636 x11
Staccato Systems	Bob Starr	650-853-7035
Tritech Microelectronics, Inc.	Rod Maier	408-941-1360
Yamaha	Jose Villafuerte (US) Kazunari Fukaya (Japan)	408-467-2300 (0539) 62-6081

Table 57. TMDS Transmitters

Vendors	Component	Contact	Phone
Silicon Images	SII164	John Nelson	(408) 873- 3111
Texas Instrument	TFP420	Greg Davis (gdavis@ti.com)	(214) 480-3662
Chrontel	CH7301	Chi Tai Hong (cthong@chrontel.com)	(408) 544-2150

Table 58. TV Encoders

Vendors	Component	Contact	Phone
Chrontel	CH7007 / CH7008	Chi Tai Hong (cthong@chrontel.com)	(408)544-2150

Vendors	Component	Contact	Phone
Chrontel	CH7010 / CH7011	Chi Tai Hong (cthong@chrontel.com)	(408)544-2150
Conexant	CN870 / CN871	Eileen Carlson (eileen.carlson@conexant.com)	(858) 713-3203
Focus	FS450 / FS451	Bill Schillhammer (billhammer@focusinfo.com)	(978) 661-0146
Philips	SAA7102A	Marcus Rosin (marcus.rosin@philips.com)	None
Texas Instrument	TFP6022/ TFP6024	Greg Davis (gdavis@ti.com)	(214) 480-3662

Table 59. Combo TMDS Transmitters/TV Encoders

Vendors	Component	Contact	Phone
Chrontel	CH7009/ CH7010	Chi Tai Hong (cthong@chrontel.com)	(408) 544-2150
Texas Instrument	TFP6422/ TFP6424	Greg Davis (gdavis@ti.com)	(214) 480-3662

Table 60. LVDS Transmitter

Vendors	Component	Contact	Phone
National Semiconductor	387R	Jason Lu (Jason.Lu@nsc.com)	(408) 721-7540

INTEL® PENTIUM® III & INTEL® CELERON (R) PROCESSOR/ 810E2 CHIPSET UNIVERSAL SOCKET 370 PLATFORM

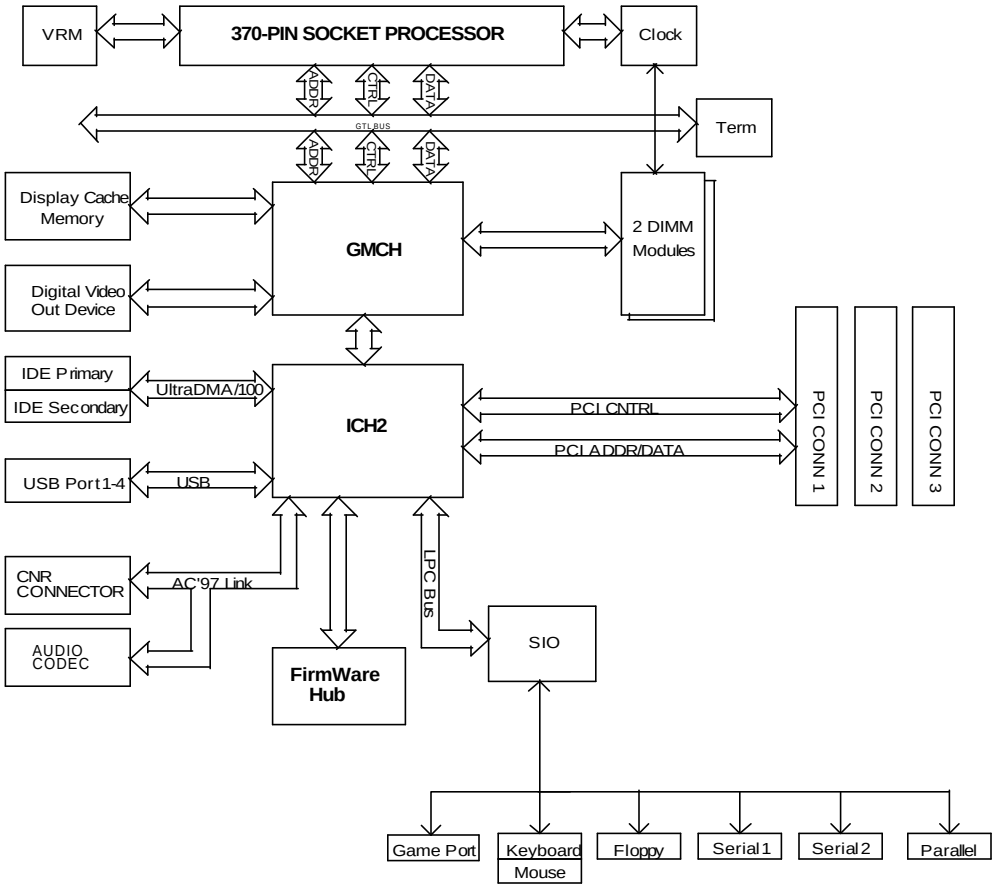
UNIPROCESSOR CUSTOMER REFERENCE SCHEMATICS

REVISION 1.0

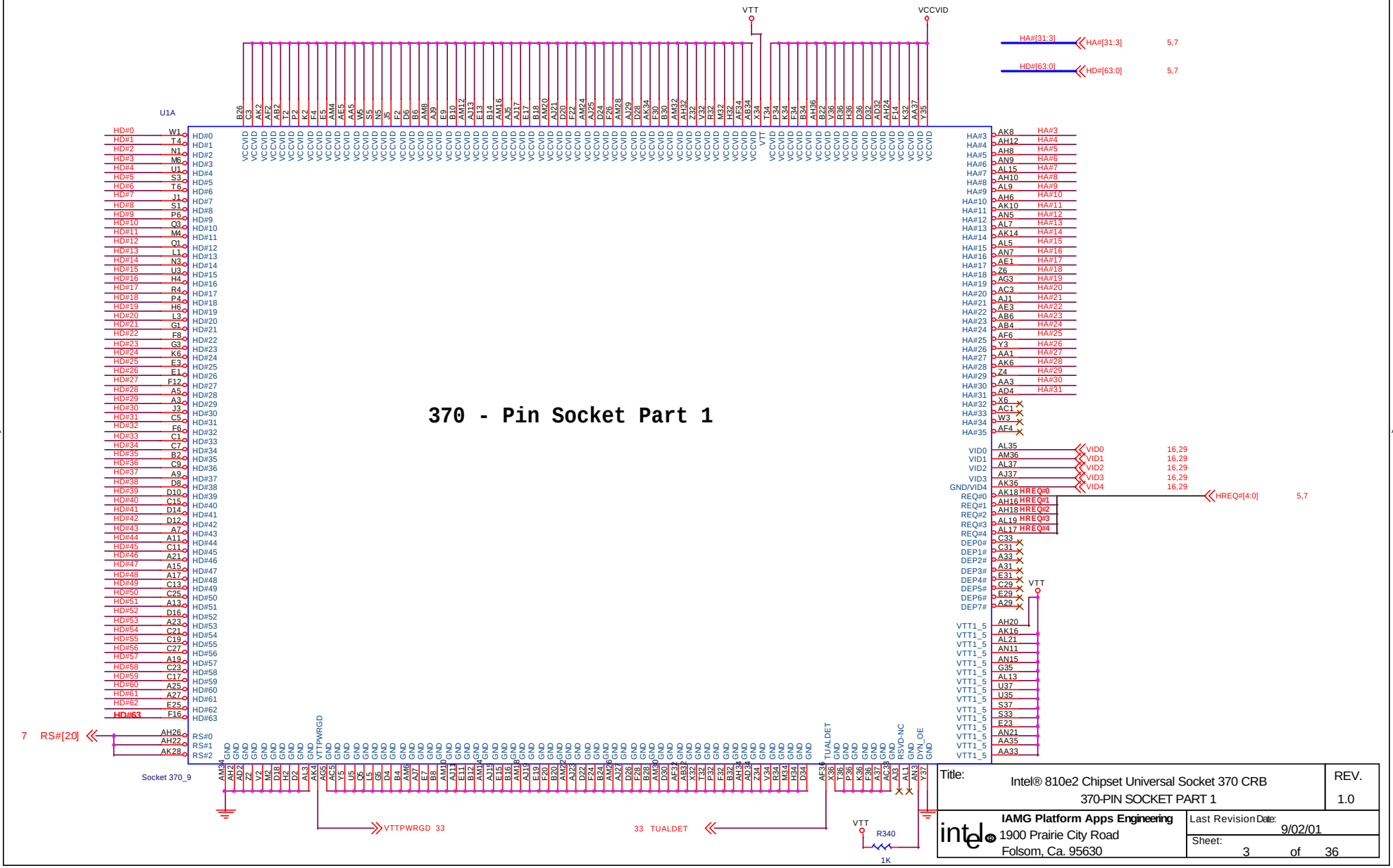
** Please note these schematics are subject to change.

Title: Intel® 810e2 Chipset Universal Socket 370 CRB Cover Sheet		REV. 1.0
intel IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date 9/02/01	
	Sheet: 1 of 36	

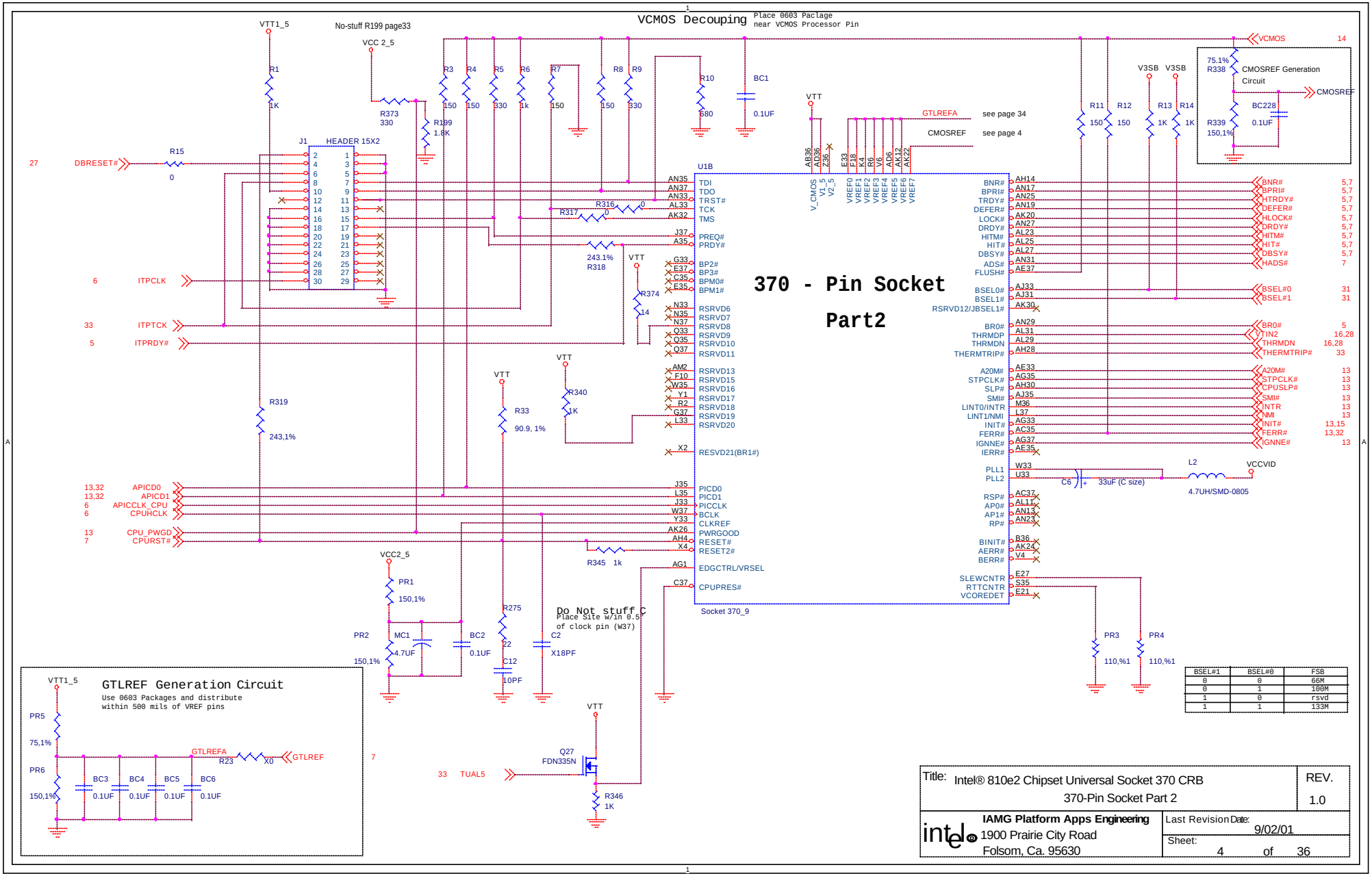
Block Diagram

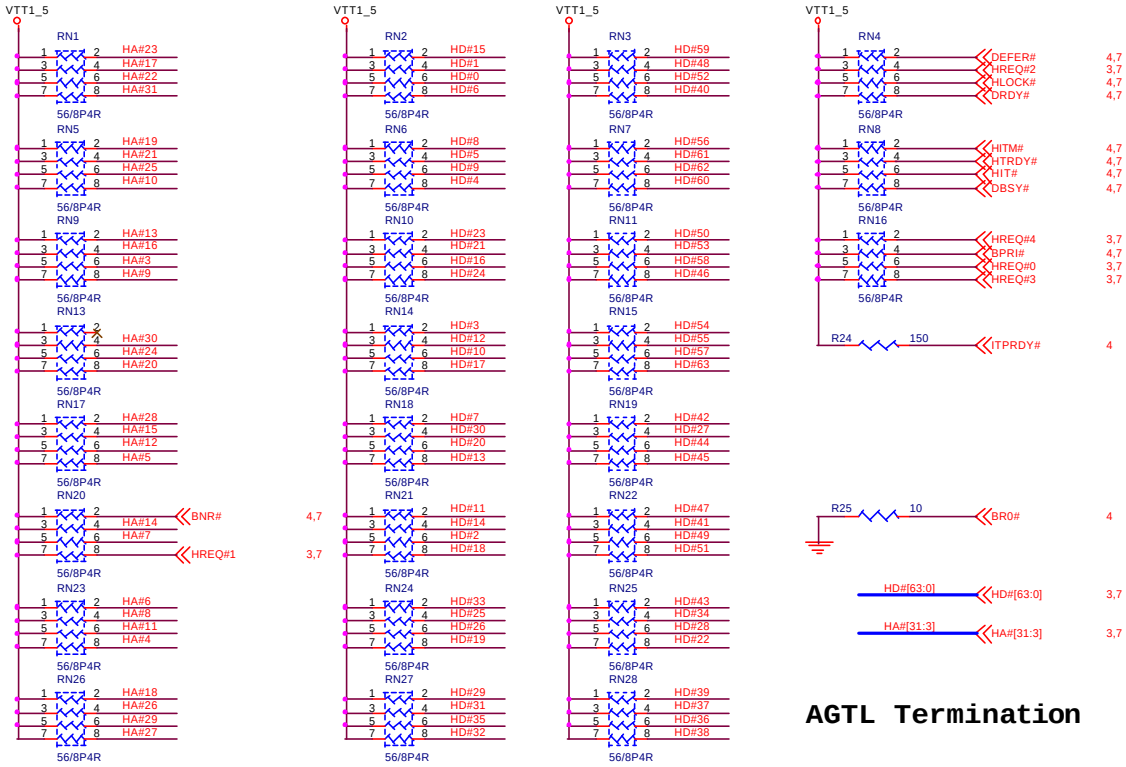


Title: Intel® 810e2 Chipset Universal Socket370CRB Block Diagram		REV. 1.0
intel IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date 9/02/01	
	Sheet: 2 of 36	



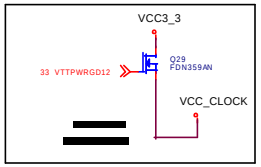
Title:	Intel® 810e2 Chipset Universal Socket 370 CRB 370-PIN SOCKET PART 1
REV.	1.0
IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 3 of 36


VCMOS Decouping
Place 0603 Paclage
near VCMOS Processor Pin
No-stuff R199 page33
VTT1_5
VCC 2_5
R1 1K
R3 150
R4 150
R5 330
R6 1k
R7 150
R8 150
R9 330
R10 680
BC1 0.1UF
R373 330
R199 1.8K
VTT
GTLREFA see page 34
CMOSREF see page 4
V3SB
R11 150
R12 150
R13 1K
R14 1K
VCMOS 14
75.1% R338
CMOSREF Generation Circuit
CMOSREF
R339 150,1%
BC228 0.1UF
J1 HEADER 15X2
27 DBRESET# R15 0
6 ITPCLK
33 ITPTCK
5 ITPRDY#
U1B
TDI
TDO
TRST#
TCK
TMS
PREQ#
PRDY#
BP2#
BP3#
BPM0#
BPM1#
RSRVD6
RSRVD7
RSRVD8
RSRVD9
RSRVD10
RSRVD11
RSRVD13
RSRVD15
RSRVD16
RSRVD17
RSRVD18
RSRVD19
RSRVD20
RESVD21(BR1#)
PICD0
PICD1
PICCLK
BCLK
CLKREF
PWRGOOD
RESET#
RESET2#
EDGCTRL/VRSEL
CPUPRES#
R316 0
R317 0
R318 243.1%
R374 14
R319 243,1%
R33 90.9, 1%
R340 1K
R345 1k
X4
X2
13,32 APICD0
13,32 APICD1
6 APICCLK_CPU
6 CPUHCLK
13 CPU_PWGD
7 CPURST#
VCC2_5
PR1 150,1%
PR2 150,1%
MC1 4.7UF
BC2 0.1UF
R275 22
C12 10PF
C2 X18PF
Do Not stuff C
Place Site w/in 0.5"
of clock pin (W37)
Q27 FDN335N
33 TUAL5
R346 1K
BNR#
BPRI#
TRDY#
DEFER#
LOCK#
DRDY#
HITM#
HIT#
DBSY#
ADS#
FLUSH#
BSEL0#
BSEL1#
RSRVD12/JBSEL1#
BR0#
THRMDP
THRMDN
THERMTRIP#
A20M#
STPCLK#
SLP#
SMI#
LINT0/INTR
LINT1/NMI
INIT#
FERR#
IGNNE#
IERR#
PLL1
PLL2
C6 33uF (C size)
L2 4.7UH/SMD-0805
VCCVID
RSP#
AP0#
AP1#
RP#
BINIT#
AERR#
BERR#
SLEWCNTR
RTTCNTR
VCOREDET
PR3 110,%1
PR4 110,%1
BNR# 5,7
BPRI# 5,7
HTRDY# 5,7
DEFER# 5,7
HLOCK# 5,7
DRDY# 5,7
HITM# 5,7
HIT# 5,7
DBSY# 5,7
HADS# 7
BSEL#0 31
BSEL#1 31
BR0# 5
VTIN2 16,28
THRMDN 16,28
THERMTRIP# 33
A20M# 13
STPCLK# 13
CPUSLP# 13
SMI# 13
INTR 13
NMI 13
INIT# 13,15
FERR# 13,32
IGNNE# 13
Socket 370_9
370 - Pin Socket
Part2
GTLREF Generation Circuit
Use 0603 Packages and distribute
within 500 mils of VREF pins
VTT1_5
PR5 75,1%
PR6 150,1%
GTLREFA
R23 X0
GTLREF
BC3 0.1UF
BC4 0.1UF
BC5 0.1UF
BC6 0.1UF
BSEL#1 BSEL#0 FSB
0 0 66M
0 1 100M
1 0 rsvd
1 1 133M
Title: Intel® 810e2 Chipset Universal Socket 370 CRB
370-Pin Socket Part 2
REV. 1.0
IAMG Platform Apps Engineering
1900 Prairie City Road
Folsom, Ca. 95630
Last Revision Date: 9/02/01
Sheet: 4 of 36



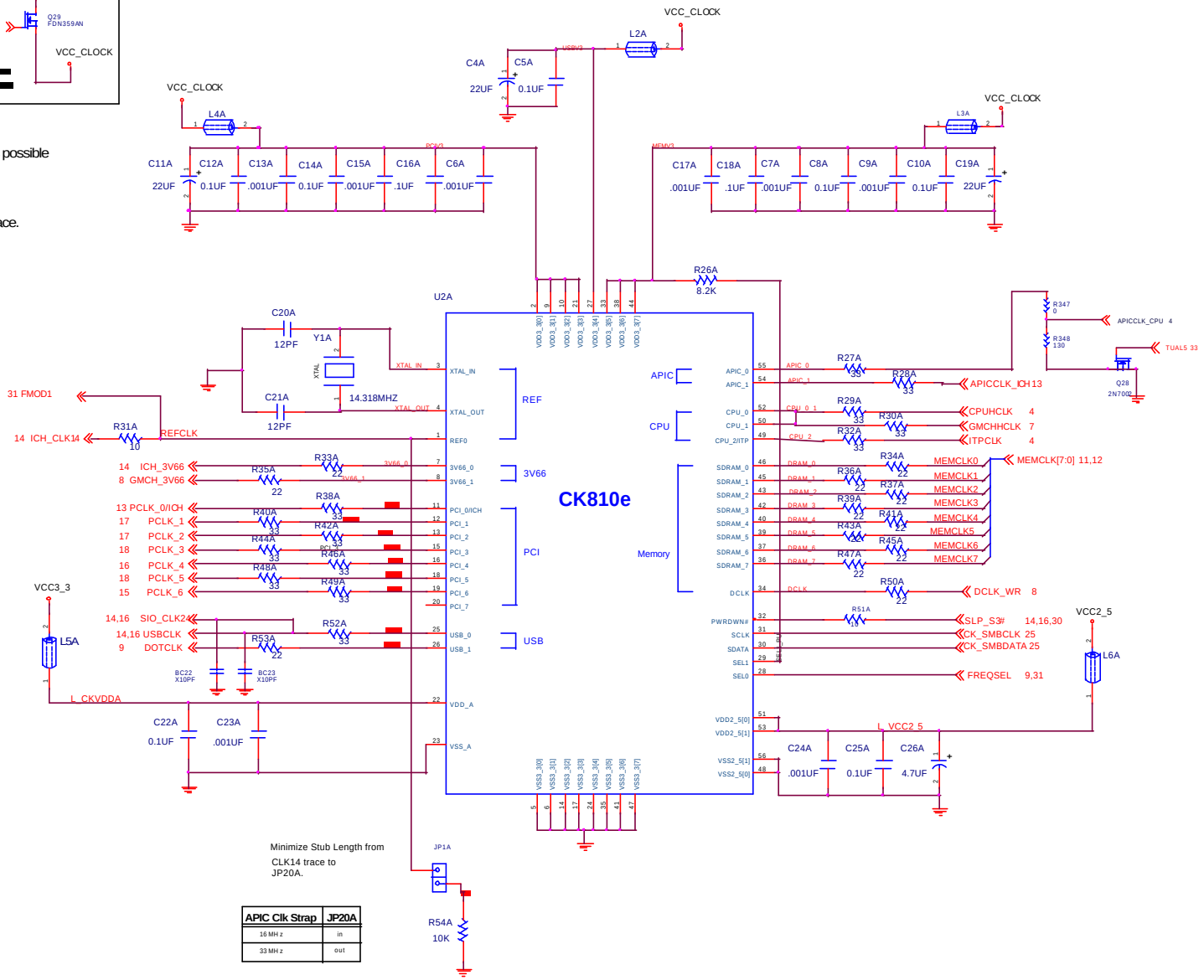
Title: Intel® 810e2 Chipset Universal Socket 370 CRB AGTL TERMINATION		REV. 1.0
intel **IAMG Platform Apps Engineering** 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 5 of 36	

Clock Synthesizer



Notes:

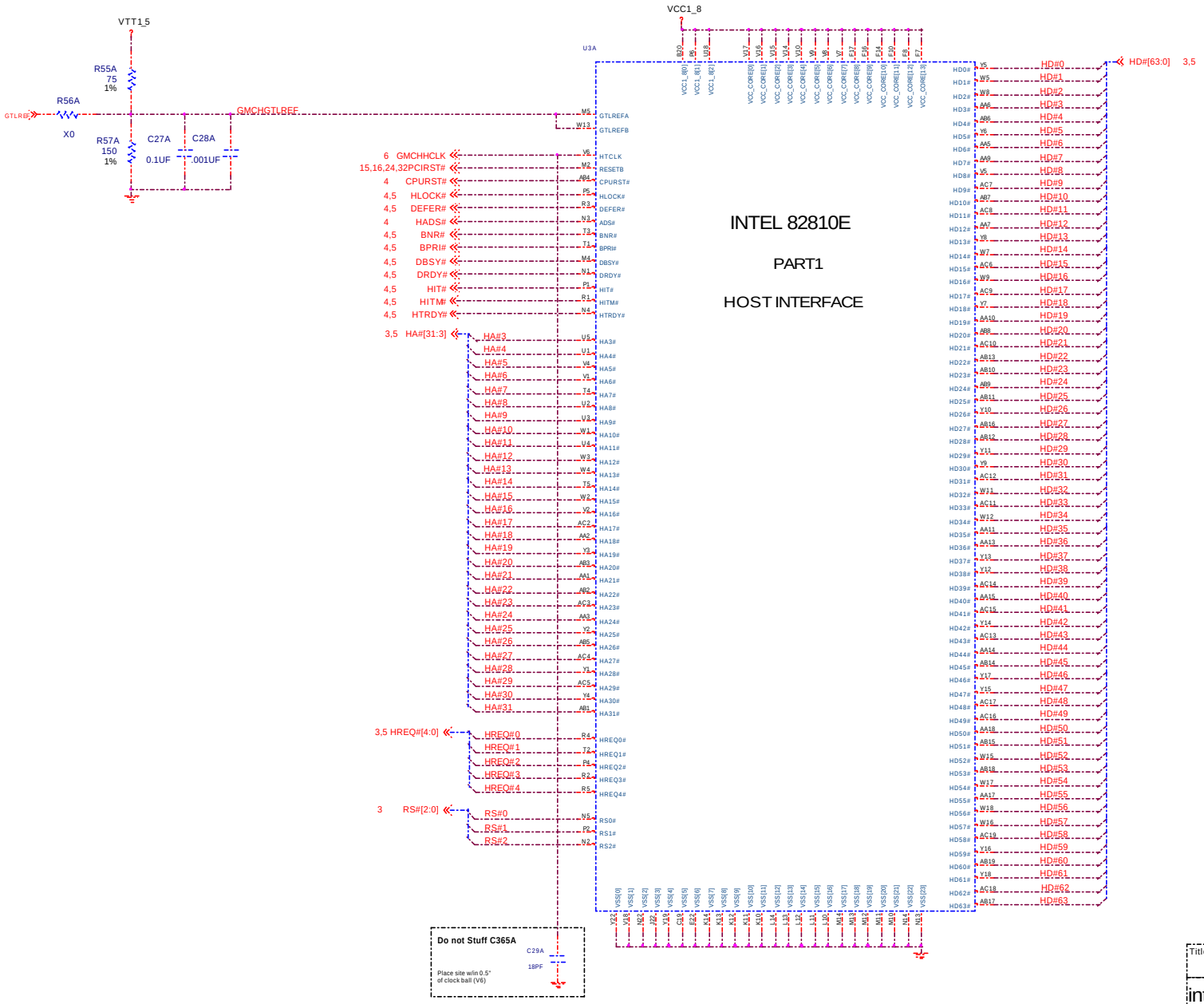
- Place all decoupling caps as close to VCC/GND pins as possible
- PCI_0/ICH pin has to go to the ICH.
  (This clock cannot be turned off through SMBus)
- CPU_ITP pin must go to the ITP. It is the only
  CPU CLK that can be shut off through the SMBUS interface.



APIC Clk Strap	JP20A
16 MHz	in
33 MHz	out

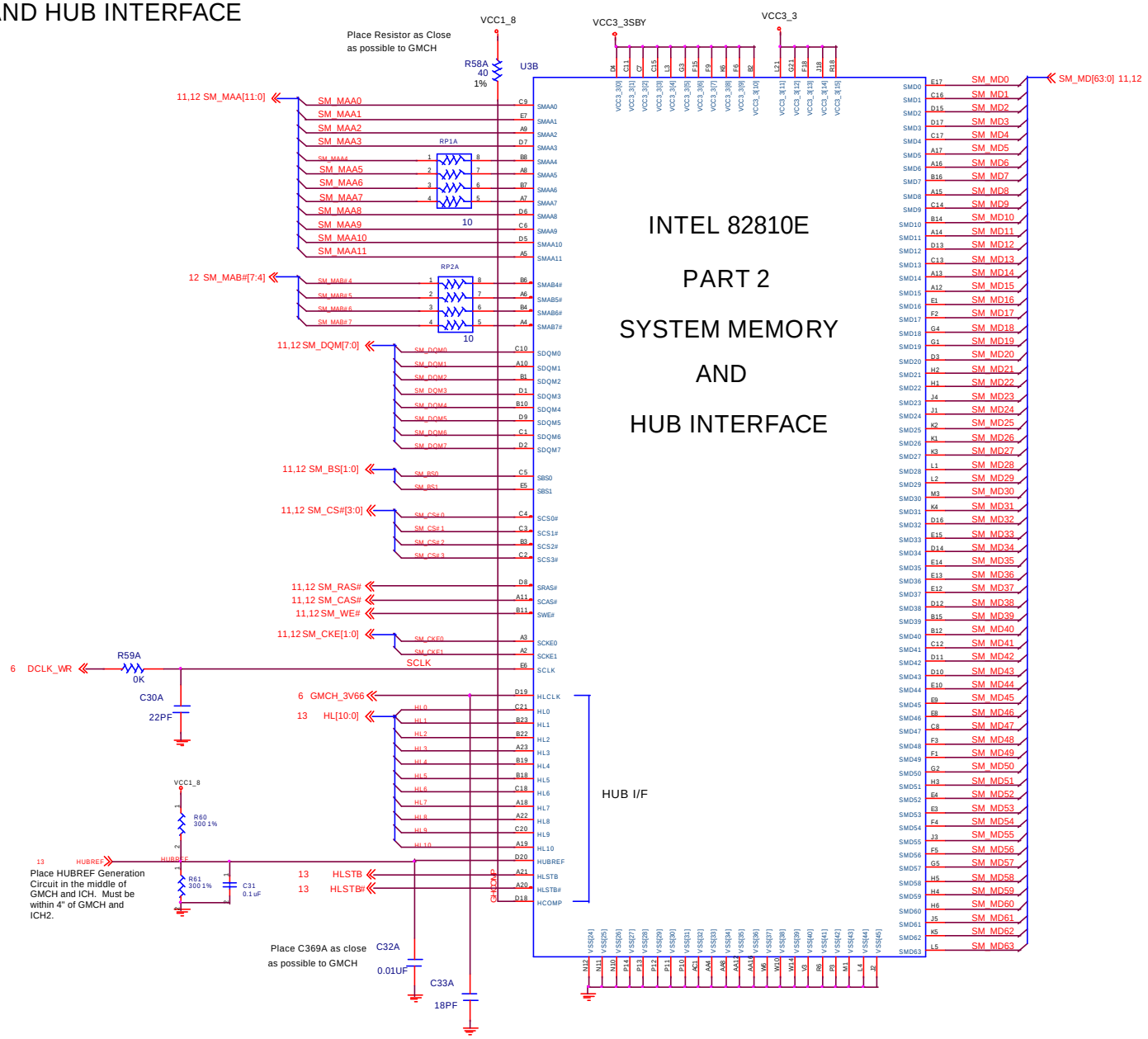
Title:	Intel® 810e2 Chipset Universal Socket370 CRB Clock Synthesizer	REV. 1.0
intel	IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date 9/02/01 Sheet 6 of 36

82810E, PART 1: HOST INTERFACE



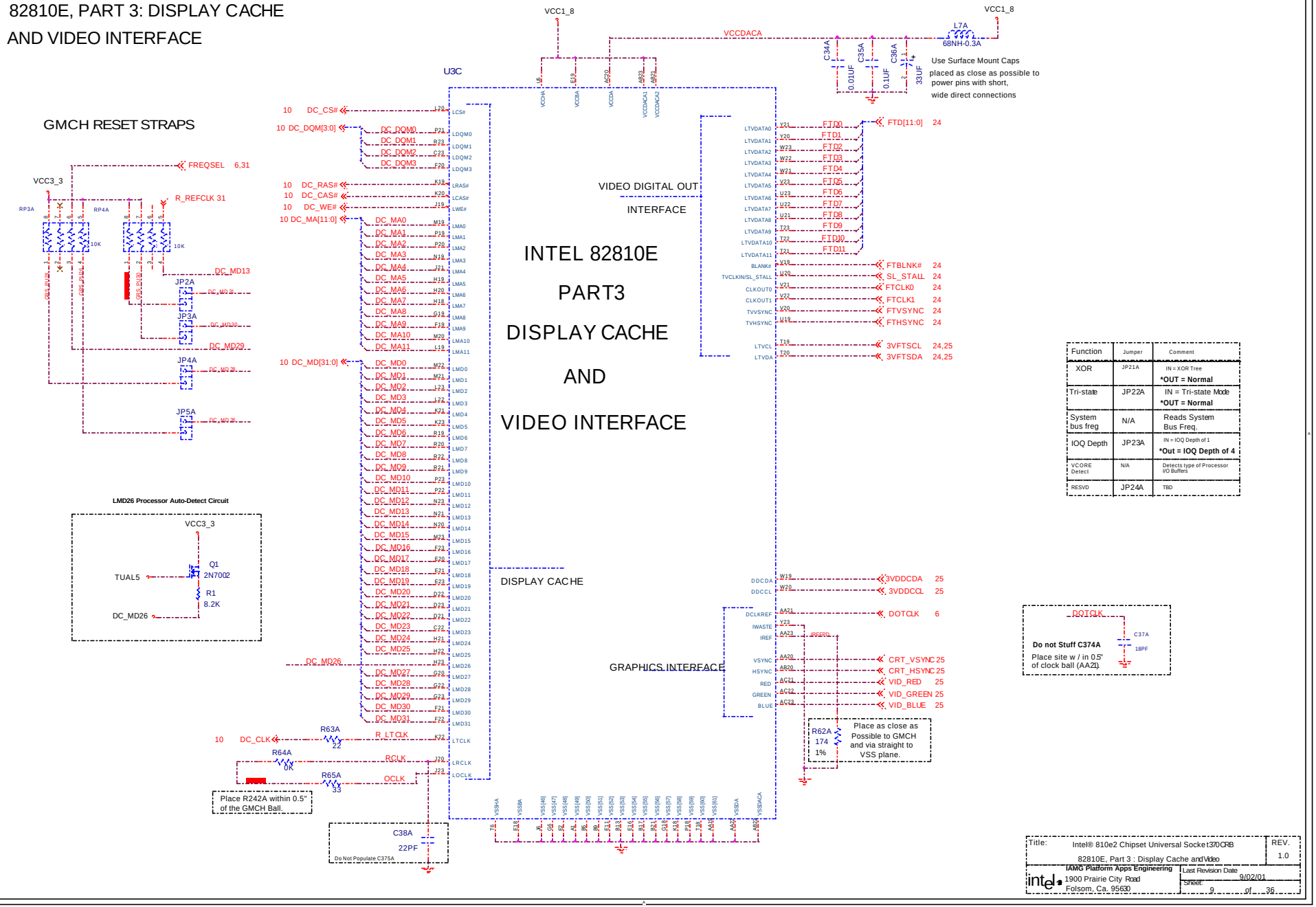
Title: Intel® 810e2 Chipset Universal Socket370CRB 82810E, Part 1: Host Interface		REV. 1.0
IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01	
	Sheet: 7 of 36	

82810E, PART 2: SYSTEM MEMORY AND HUB INTERFACE



Title:	Intel® 810e2 Chipset Universal Socket370CRB 82810E, Part 2: System Memory and Hub	REV. 1.0
intel	IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date 9/02/01 Sheet: 8 of 36

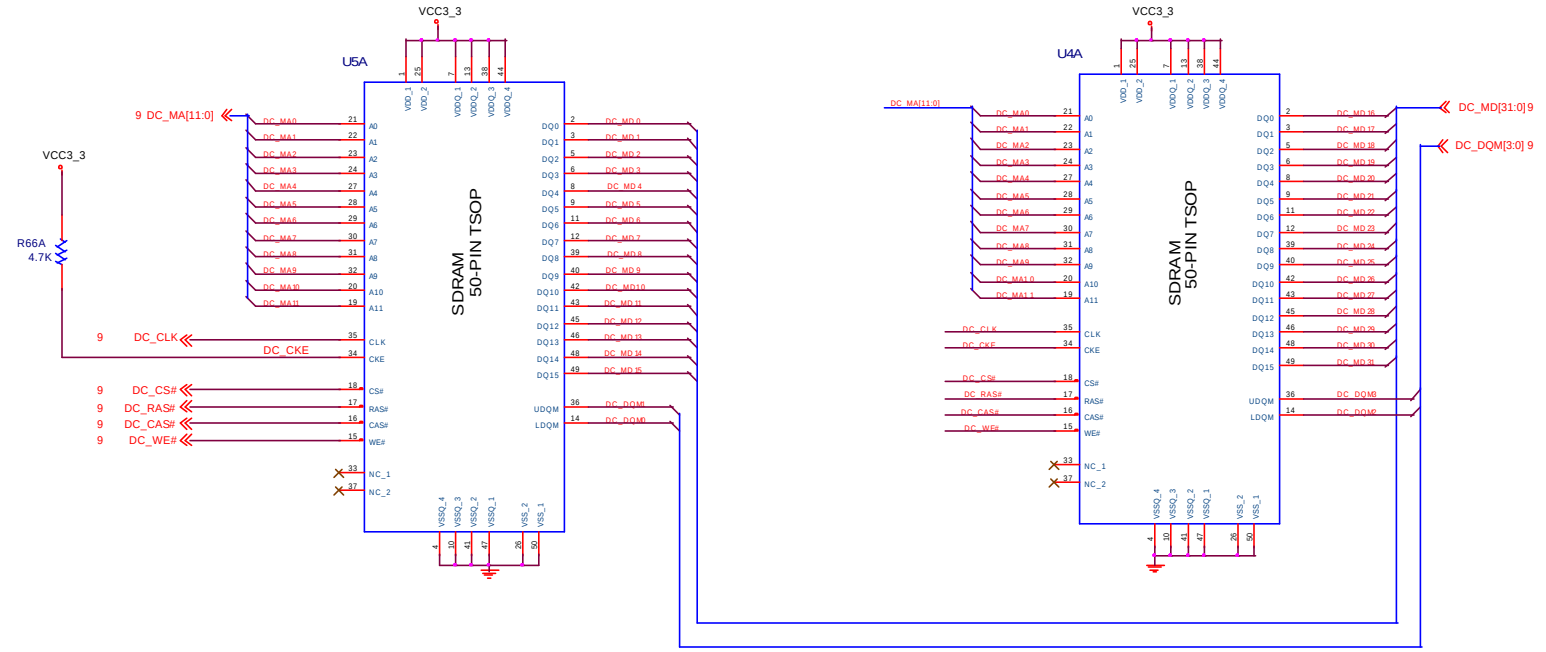
82810E, PART 3: DISPLAY CACHE AND VIDEO INTERFACE



Function	Jumper	Comment
XOR	JP21A	IN = XOR Tree *OUT = Normal
Tri-state	JP22A	IN = Tri-state Mode *OUT = Normal
System bus freq	N/A	Reads System Bus Freq.
IOQ Depth	JP23A	IN = IOQ Depth of 1 *Out = IOQ Depth of 4
VCORE Detect	N/A	Detects type of Processor I/O Buffers
RESVD	JP24A	TBD

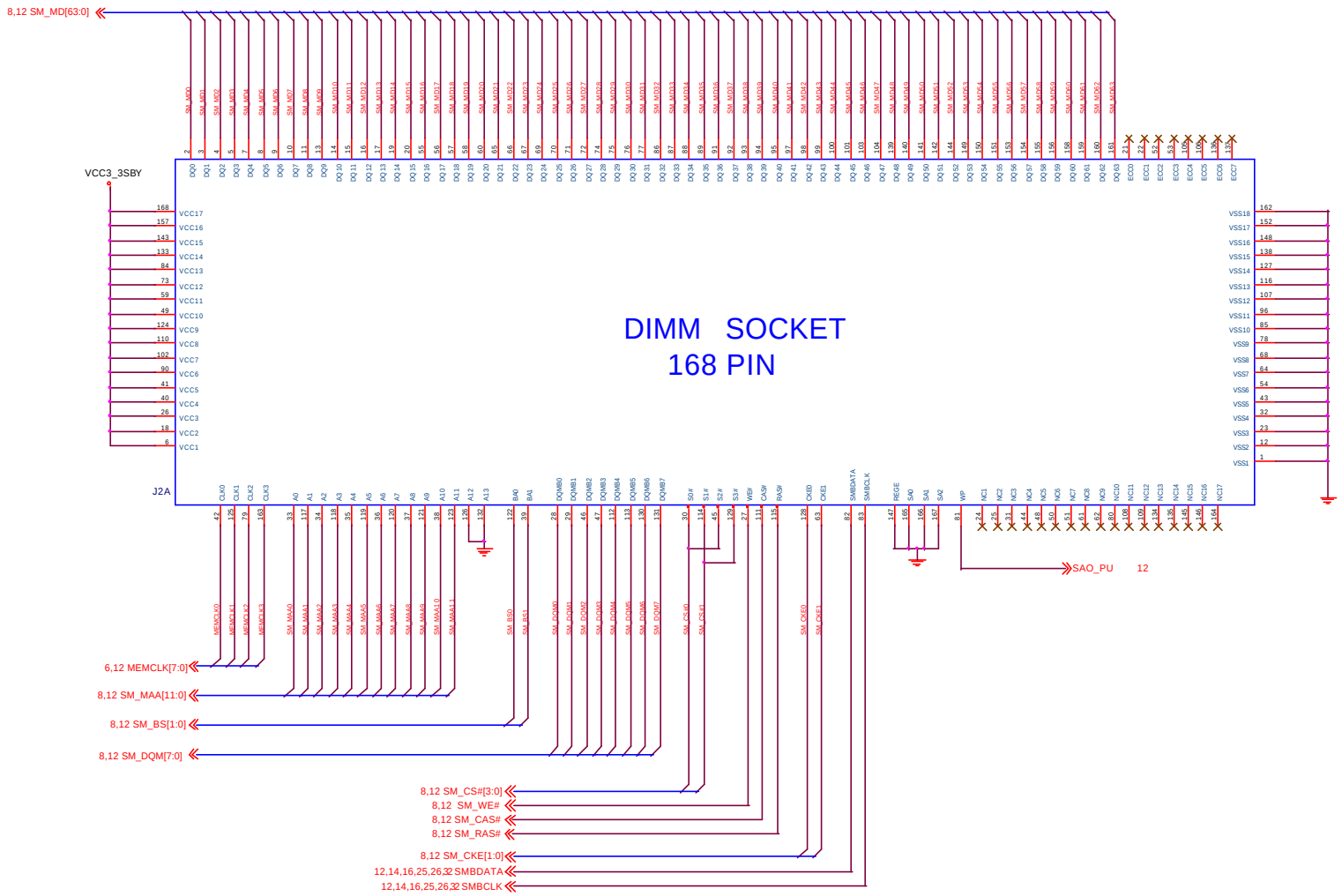
Title:	Intel® 810e2 Chipset Universal Socket 370 CRB 82810E, Part 3 : Display Cache and Video	REV. 1.0
IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet 9 of 36	

4MB Display Cache



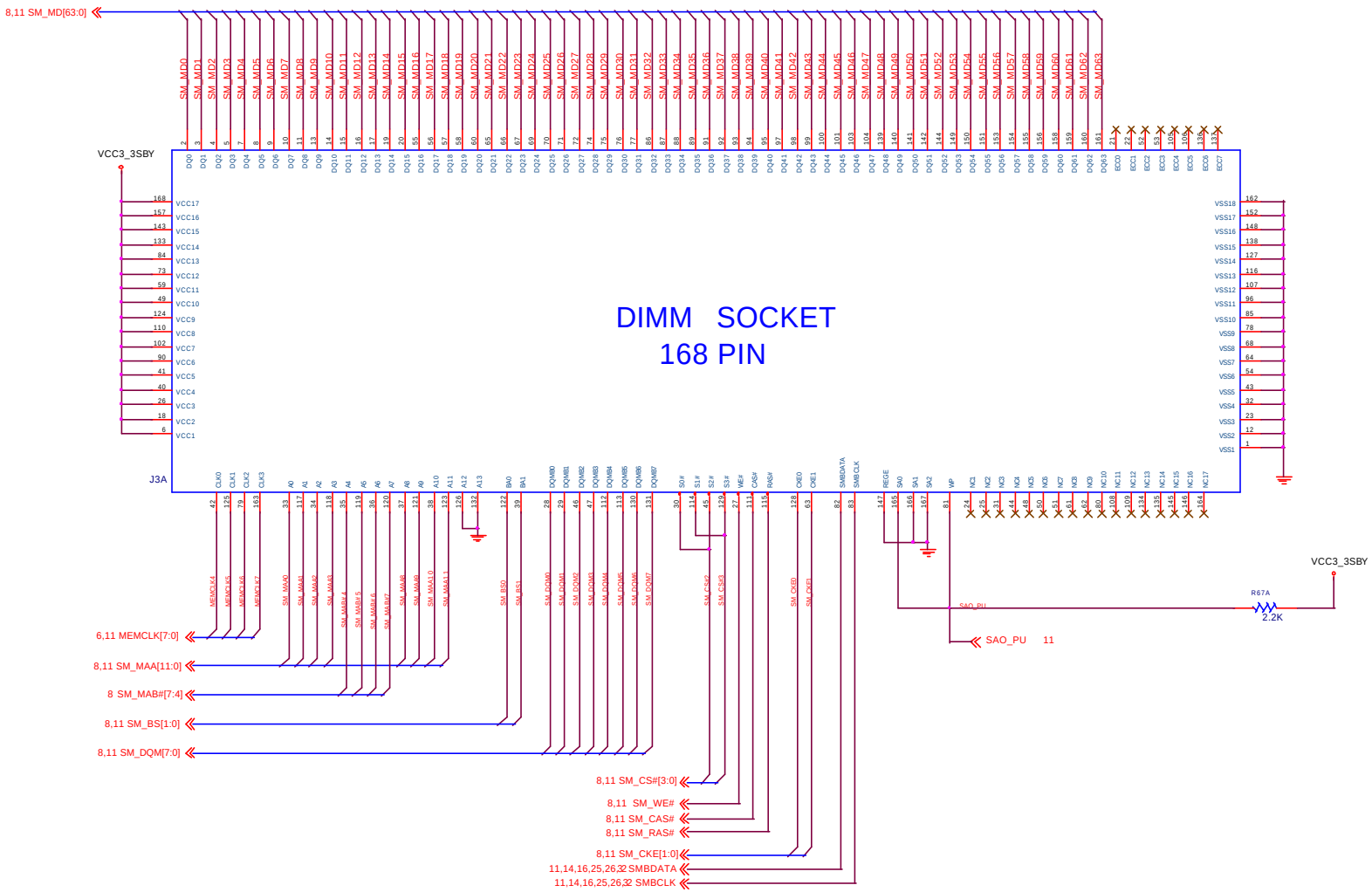
Title:	Intel® 810e2 Chipset Universal Socket370CRB Display Cache	REV. 1.0
intel	IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date 9/02/01 Sheet: 10 of 36

SYSTEM MEMORY

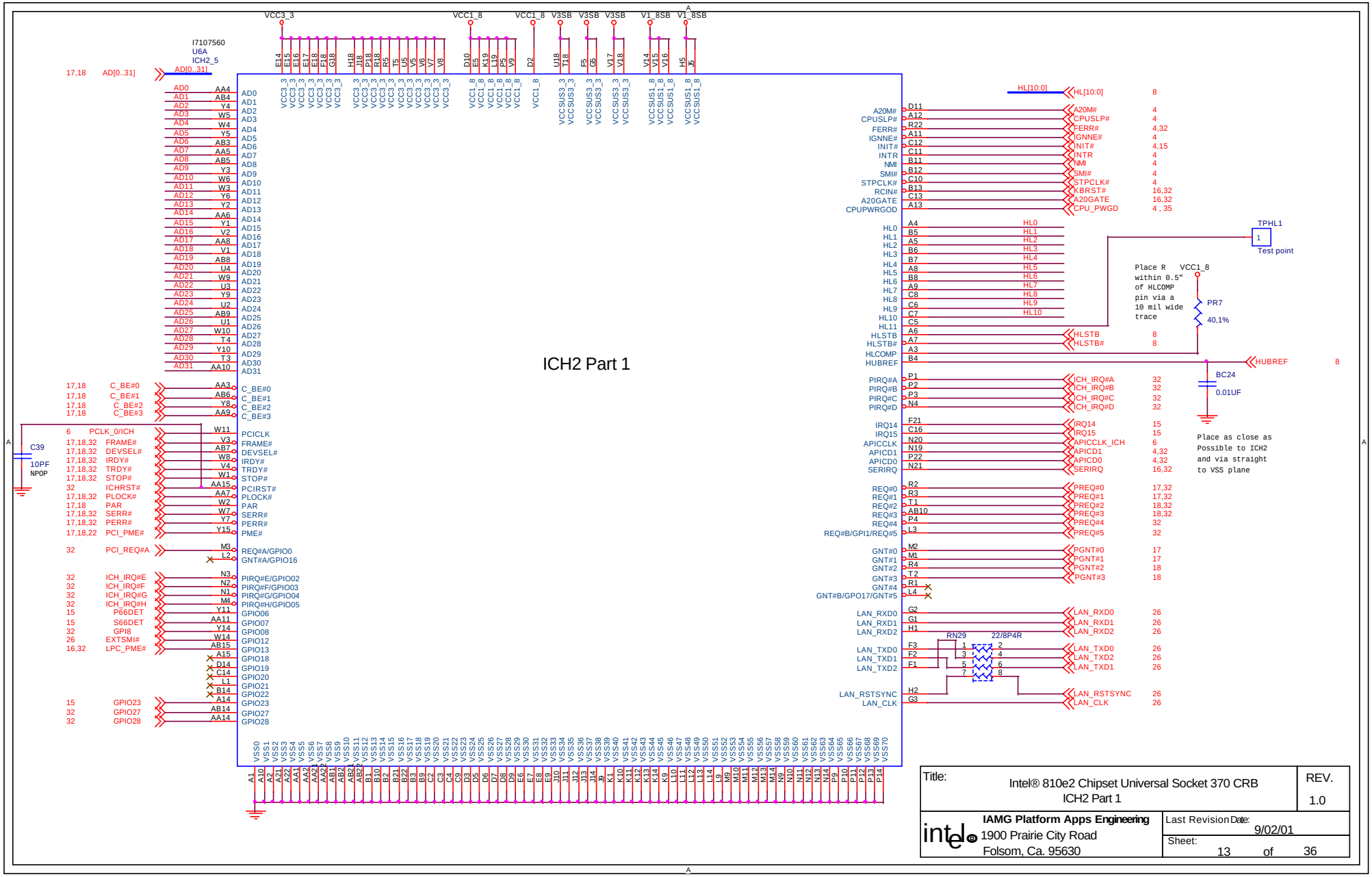


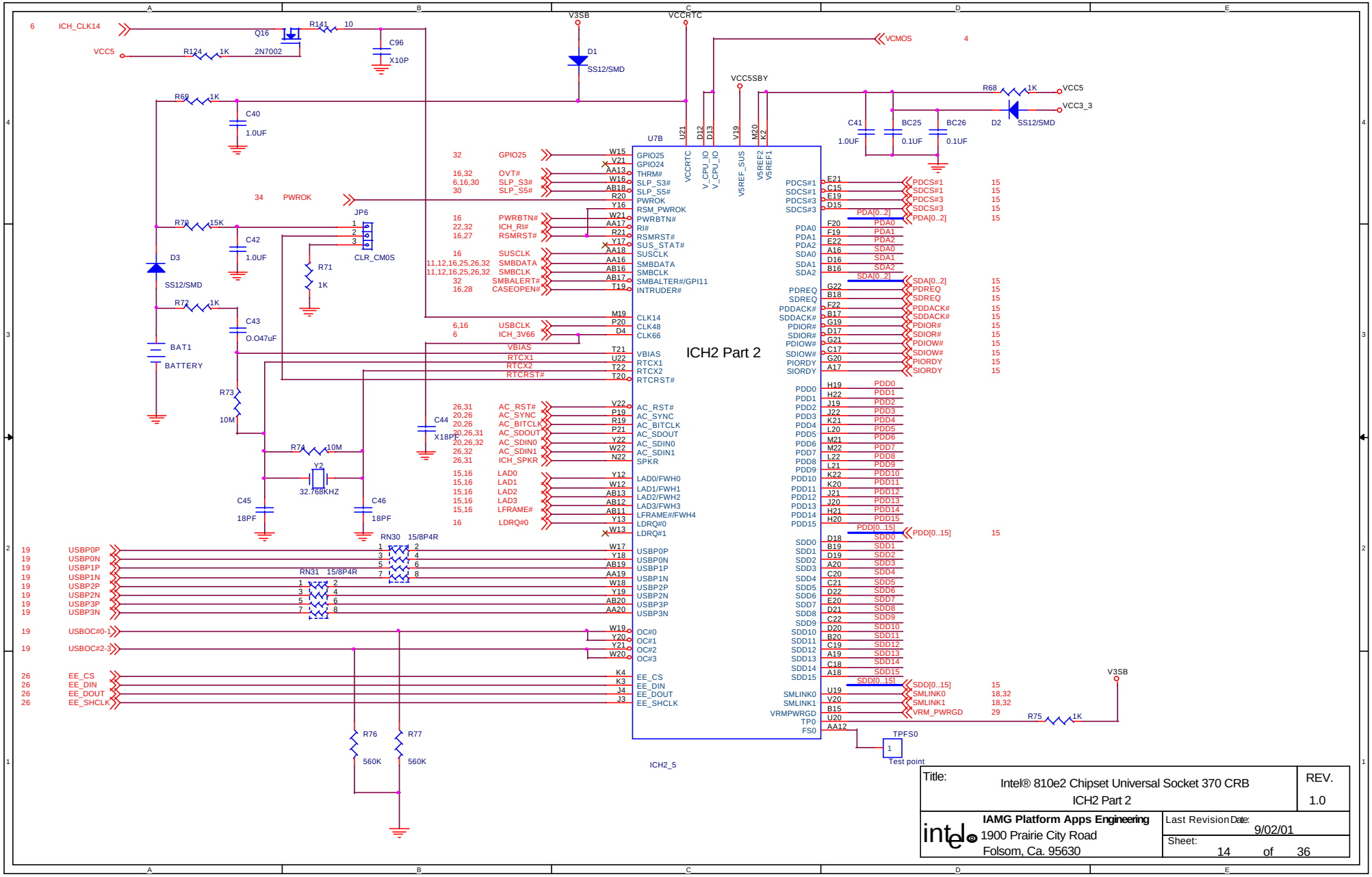
Title:	Intel® 810e2 Chipset Universal Socket370 CRB System Memory : DIMM0	REV. 1.0
intel	IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date 9/02/01 Sheet: 11 of 36

SYSTEM MEMORY

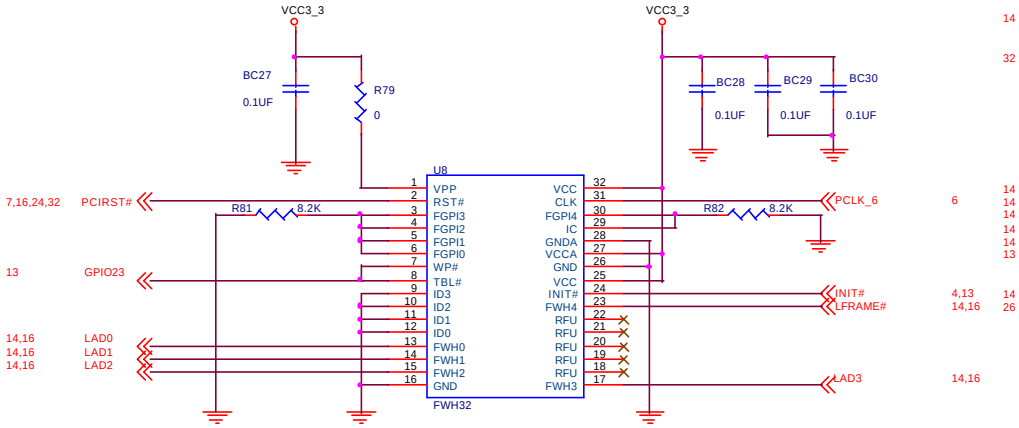


Title:	Intel® 810e2 Chipset Universal Socket370CRB System Memory : DIMM1	REV. 1.0
intel	IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date 9/02/01 Sheet: 12 of 36

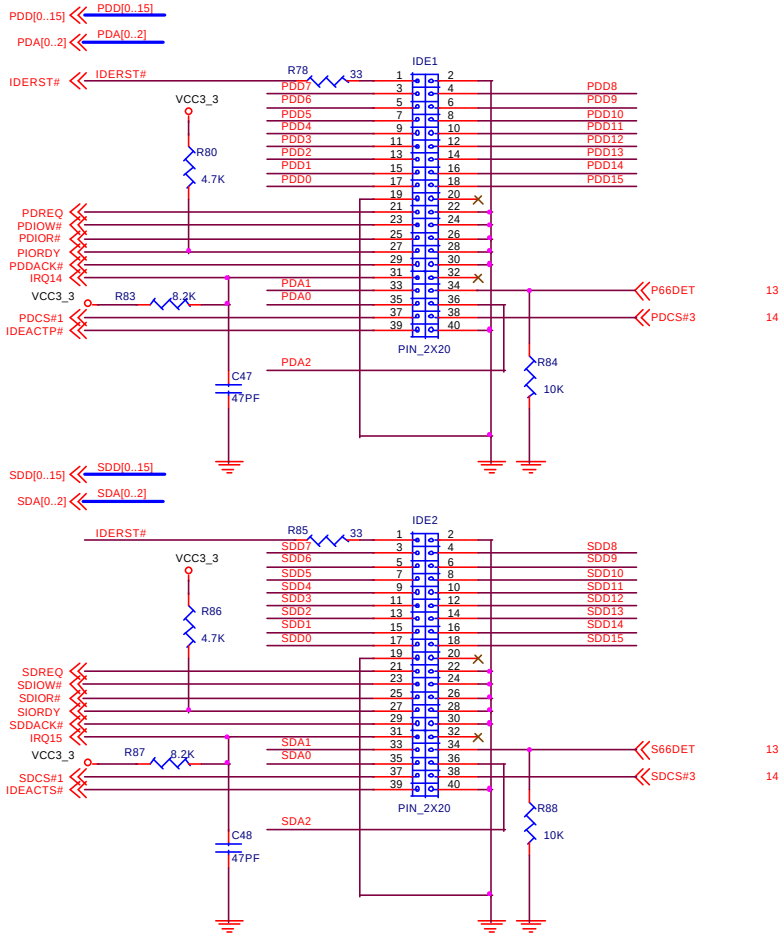

ICH2 Part 1
I7107560
U6A
ICH2_5
AD[0..31]
C39
10PF
NPOP
PCLK_0/ICH
C_BE#0
C_BE#1
C_BE#2
C_BE#3
FRAME#
DEVSEL#
IRDY#
TRDY#
STOP#
ICHRST#
PLOCK#
PAR
SERR#
PERR#
PCI_PME#
PCI_REQ#A
ICH_IRQ#E
ICH_IRQ#F
ICH_IRQ#G
ICH_IRQ#H
P66DET
S66DET
GPI8
EXTSMI#
LPC_PME#
GPIO23
GPIO27
GPIO28
PCICLK
REQ#A/GPIO0
GNT#A/GPIO16
PIRQ#E/GPIO02
PIRQ#F/GPIO03
PIRQ#G/GPIO04
PIRQ#H/GPIO05
GPIO06
GPIO07
GPIO08
GPIO12
GPIO13
GPIO18
GPIO19
GPIO20
GPIO21
GPIO22
A20M#
CPUSLP#
FERR#
IGNNE#
INIT#
INTR
NMI
SMI#
STPCLK#
RCIN#
A20GATE
CPUPWRGOD
HL[10:0]
HLSTB
HLSTB#
HLCOMP
HUBREF
PIRQ#A
PIRQ#B
PIRQ#C
PIRQ#D
IRQ14
IRQ15
APICCLK
APICD1
APICD0
SERIRQ
REQ#0
REQ#1
REQ#2
REQ#3
REQ#4
REQ#B/GPI1/REQ#5
GNT#0
GNT#1
GNT#2
GNT#3
GNT#4
GNT#B/GPO17/GNT#5
LAN_RXD0
LAN_RXD1
LAN_RXD2
LAN_TXD0
LAN_TXD1
LAN_TXD2
LAN_RSTSYNC
LAN_CLK
RN29
22/8P4R
TPHL1
Test point
Place R within 0.5" of HLCOMP pin via a 10 mil wide trace
PR7
40,1%
BC24
0.01UF
Place as close as Possible to ICH2 and via straight to VSS plane
VCC3_3
VCC1_8
V3SB
V1_8SB
VCCSUS3_3
VCCSUS1_8
Title:
Intel® 810e2 Chipset Universal Socket 370 CRB
ICH2 Part 1
REV.
1.0
IAMG Platform Apps Engineering
1900 Prairie City Road
Folsom, Ca. 95630
Last Revision Date:
9/02/01
Sheet:
13
of
36


ICH2 Part 2
U7B
ICH2_5
V3SB
VCCRTC
VCMOS
VCC5SBY
VCC5
VCC3_3
SS12/SMD
BATTERY
CLR_CM0S
TPFS0
Test point
Title:
Intel® 810e2 Chipset Universal Socket 370 CRB
ICH2 Part 2
REV.
1.0
IAMG Platform Apps Engineering
1900 Prairie City Road
Folsom, Ca. 95630
Last Revision Date:
9/02/01
Sheet:
14 of 36

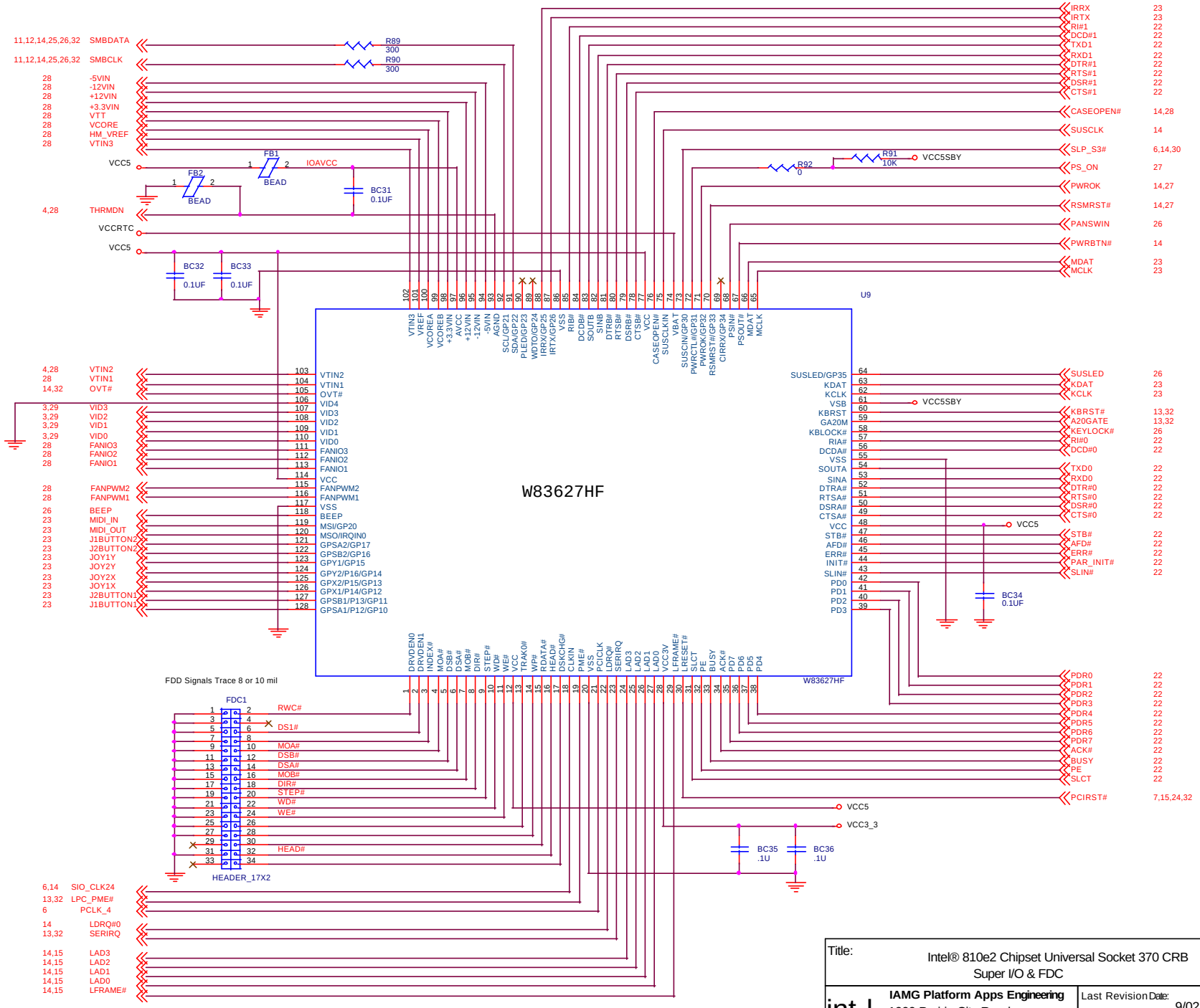
FWH



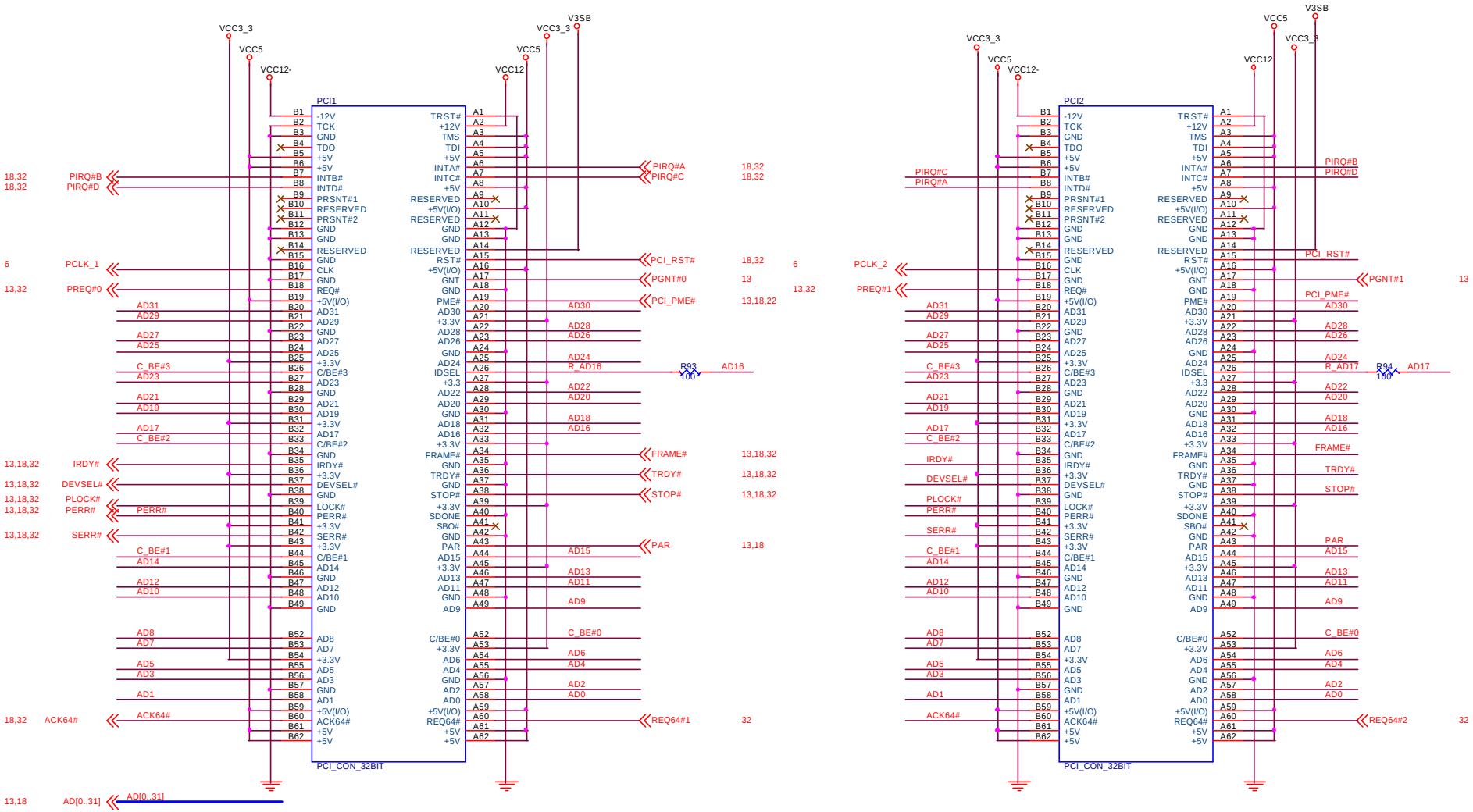
IDE

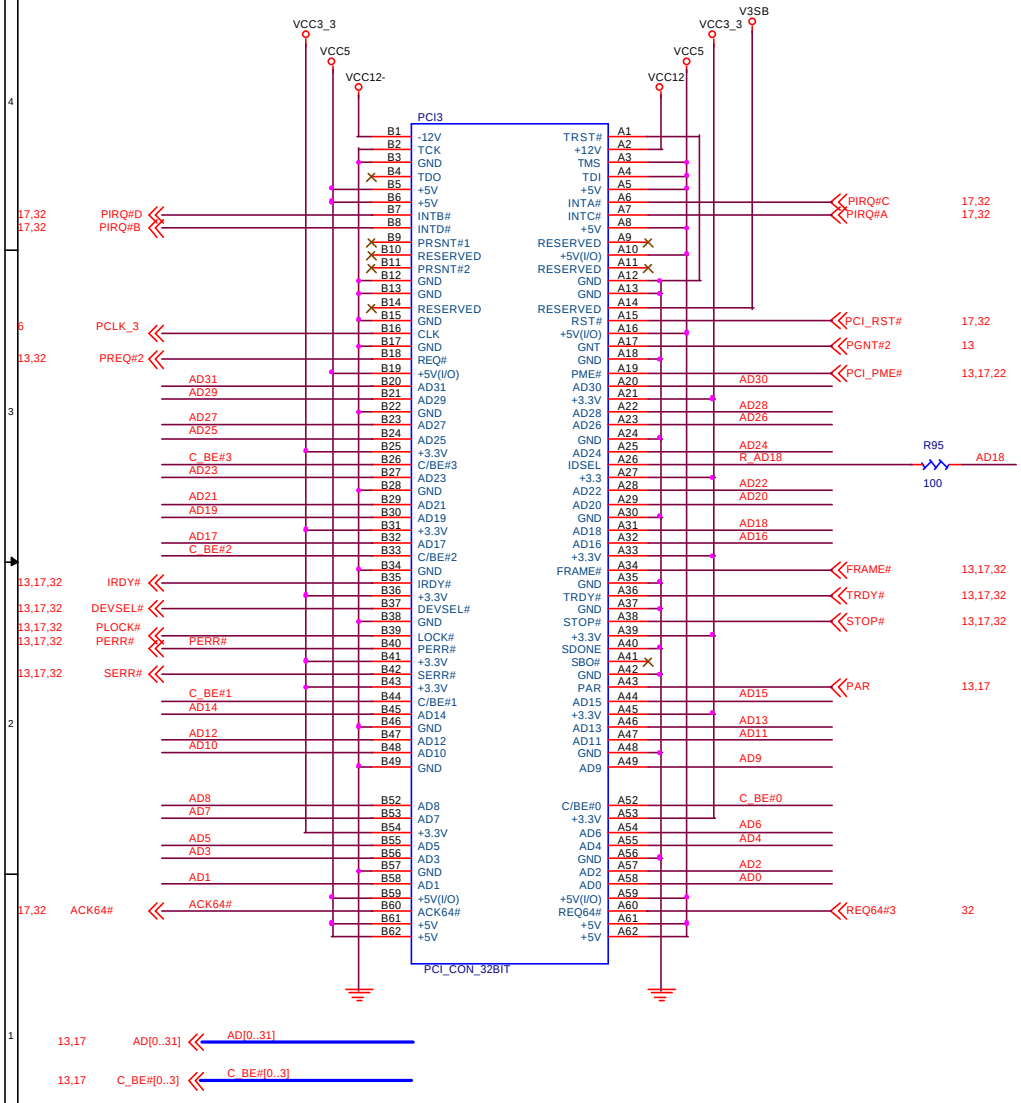


Title:	Intel® 810e2 Chipset Universal Socket 370 CRB FWH & ULTRA-ATA100 IDE connectors	REV. 1.0
intel **IAMG Platform Apps Engineering** 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 15 of 36	

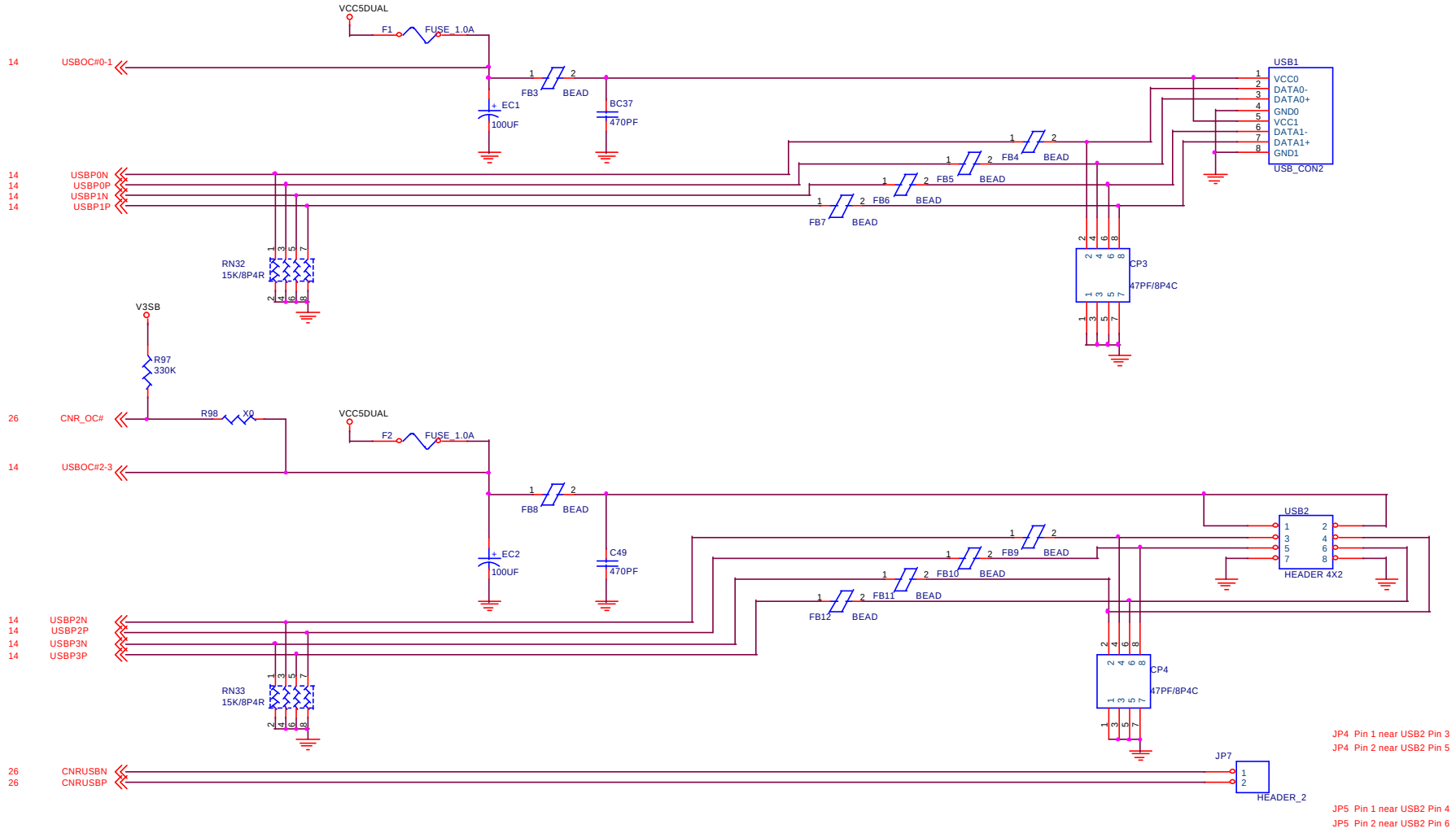


Title:	Intel® 810e2 Chipset Universal Socket 370 CRB Super I/O & FDC	REV. 1.0
intel **IAMG Platform Apps Engineering** 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 16 of 36	

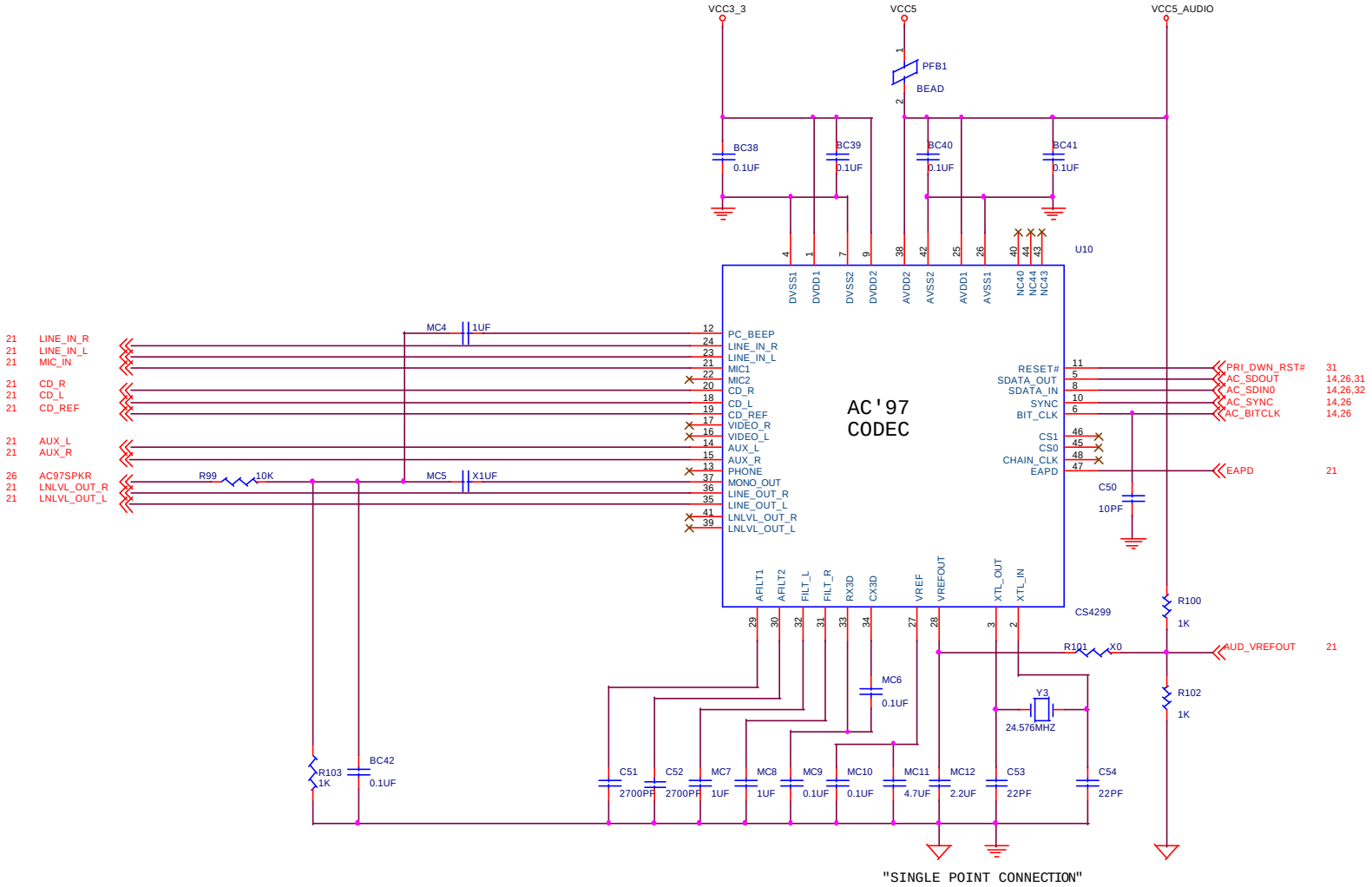

PCI1
PCI_CON_32BIT
PCI2
PCI_CON_32BIT
VCC3_3
VCC5
VCC12-
VCC12
V3SB
PIRQ#B
PIRQ#D
PIRQ#A
PIRQ#C
PCLK_1
PCLK_2
PREQ#0
PREQ#1
PCI_RST#
PGNT#0
PGNT#1
PCI_PME#
AD31
AD29
AD27
AD25
C_BE#3
AD23
AD21
AD19
AD17
C_BE#2
IRDY#
DEVSEL#
PLOCK#
PERR#
SERR#
C_BE#1
AD14
AD12
AD10
AD8
AD7
AD5
AD3
AD1
ACK64#
REQ64#1
REQ64#2
FRAME#
TRDY#
STOP#
PAR
C_BE#0
AD6
AD4
AD2
AD0
AD9
AD15
AD13
AD11
R_AD16
R_AD17
R93
R94
100
150
AD16
AD17
AD[0..31]
C_BE#[0..3]
Title:
Intel® 810e2 Chipset Universal Socket 370 CRB
PCI 1 & 2
REV.
1.0
IAMG Platform Apps Engineering
1900 Prairie City Road
Folsom, Ca. 95630
Last Revision Date:
9/02/01
Sheet:
17
of
36



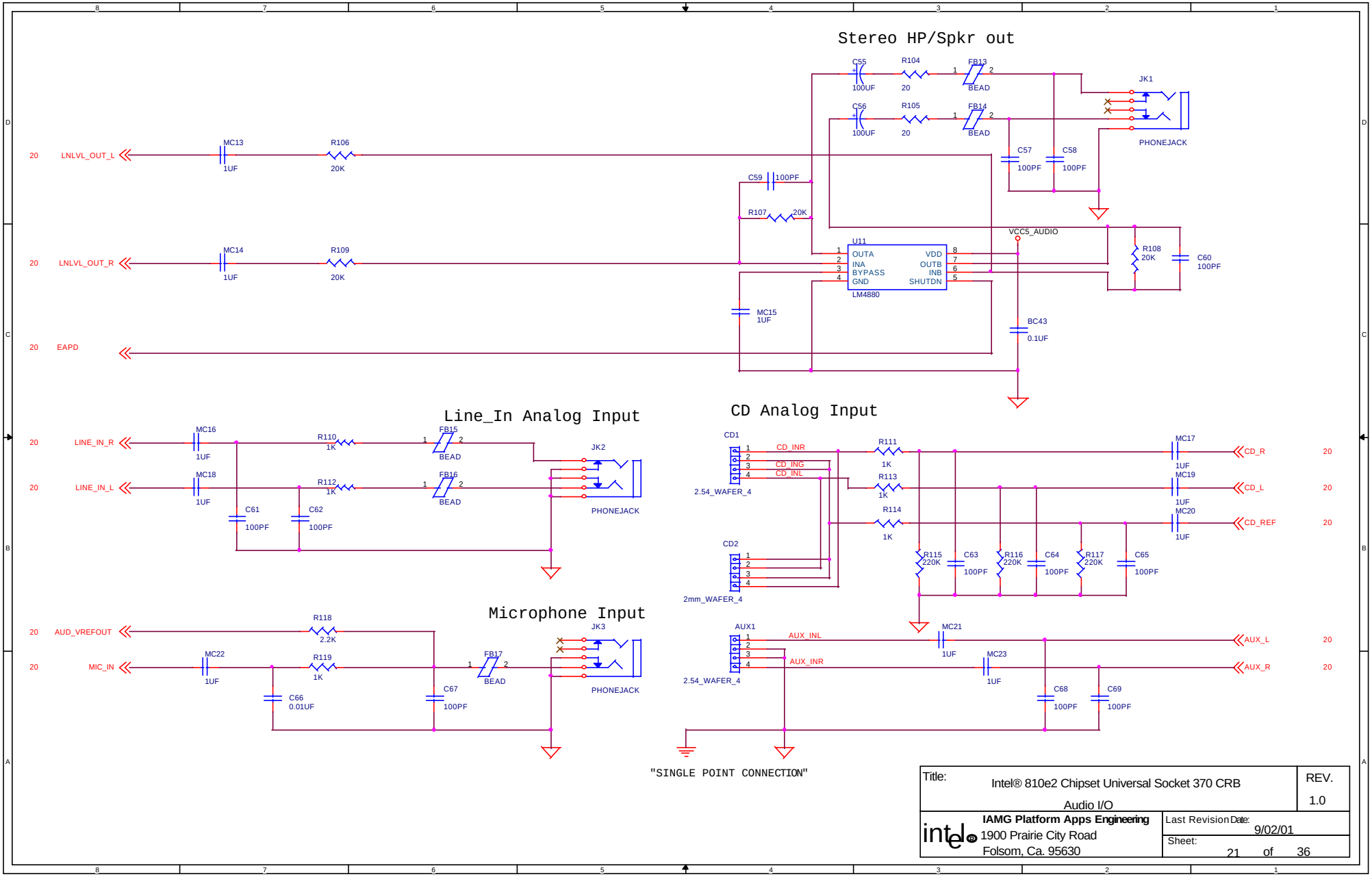
Title: Intel® 810e2 Chipset Universal Socket 370 CRB PCI 3		REV. 1.0
IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01	
	Sheet: 18 of 36	



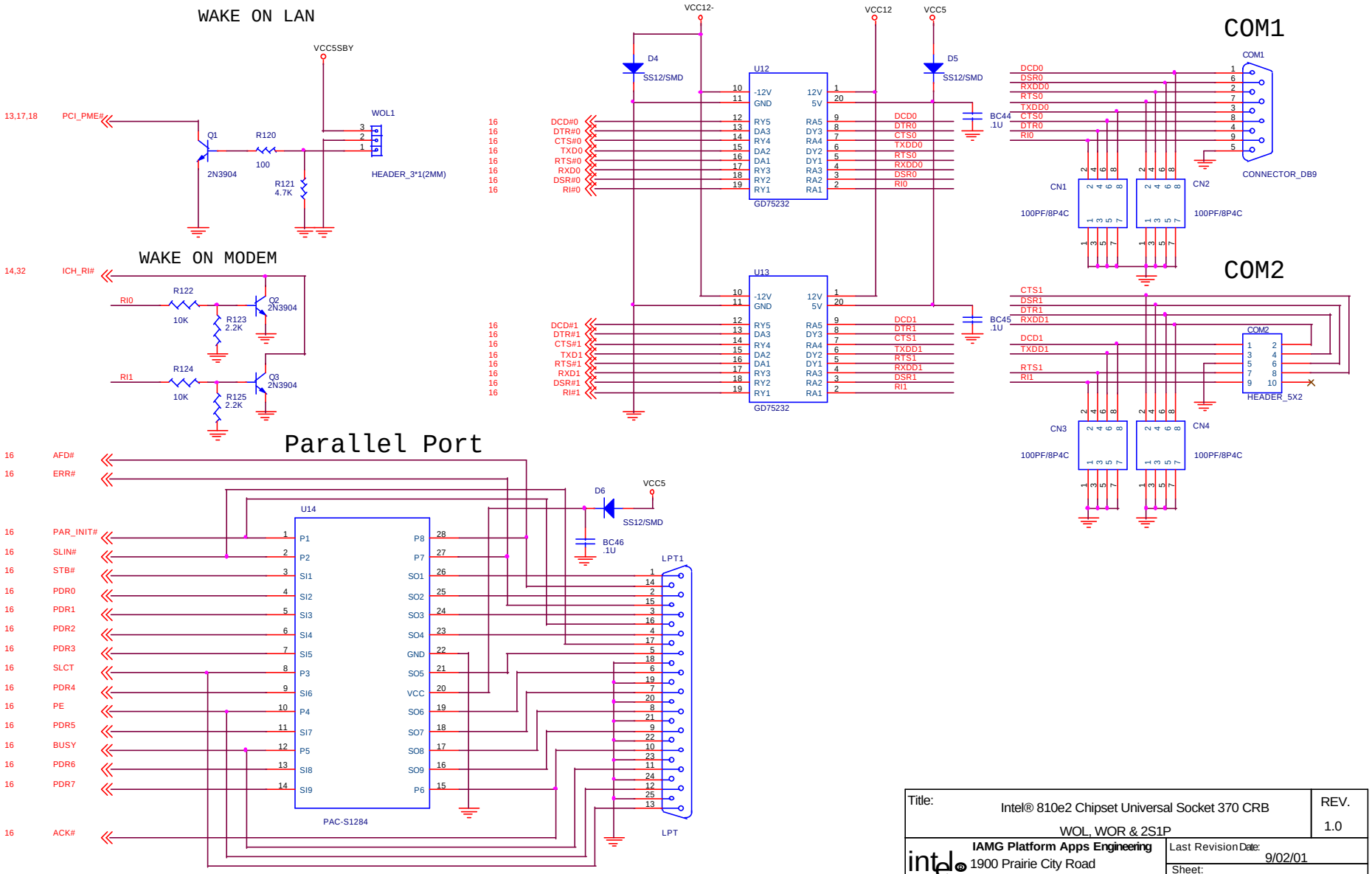
Title: Intel® 810e2 Chipset Universal Socket 370 CRB USB 0-3		REV. 1.0
intel® IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01	
	Sheet: 19 of 36	



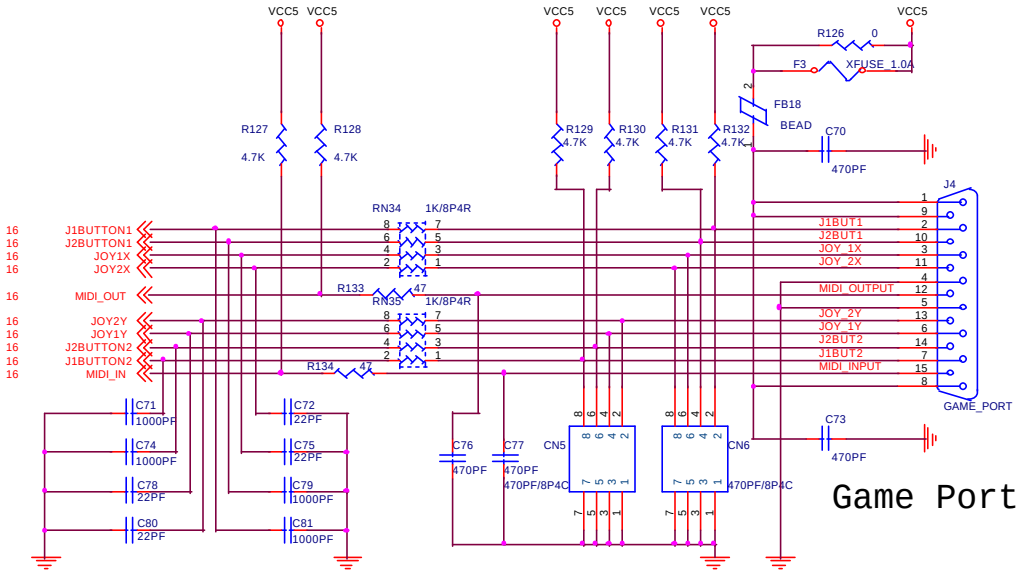
Title:	Intel® 810e2 Chipset Universal Socket 370 CRB AC97 CODEC	REV. 1.0
intel IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 20 of 36	



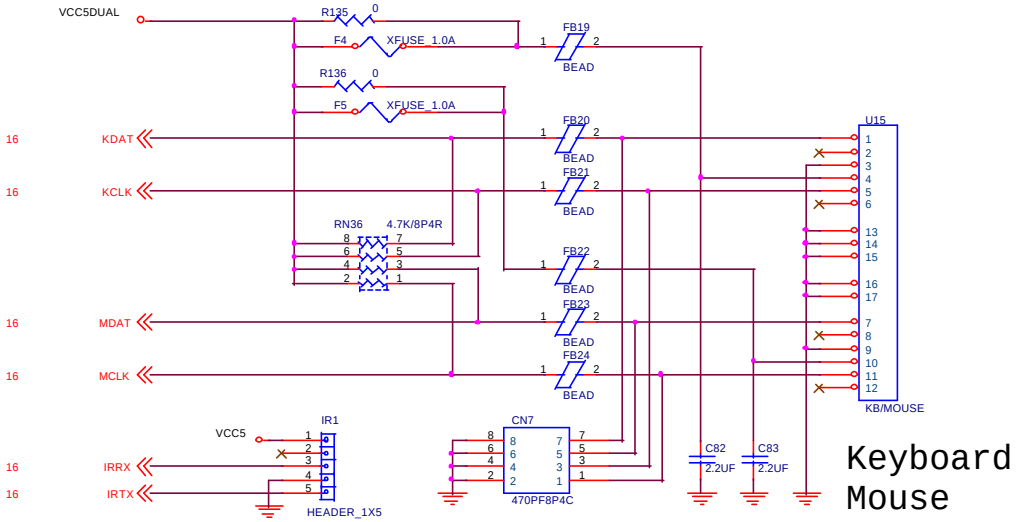
Title:	Intel® 810e2 Chipset Universal Socket 370 CRB Audio I/O	REV. 1.0
intel	**IAMG Platform Apps Engineering** 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 21 of 36



Title:	Intel® 810e2 Chipset Universal Socket 370 CRB WOL, WOR & 2S1P	REV. 1.0
intel IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 22 of 36	



Game Port

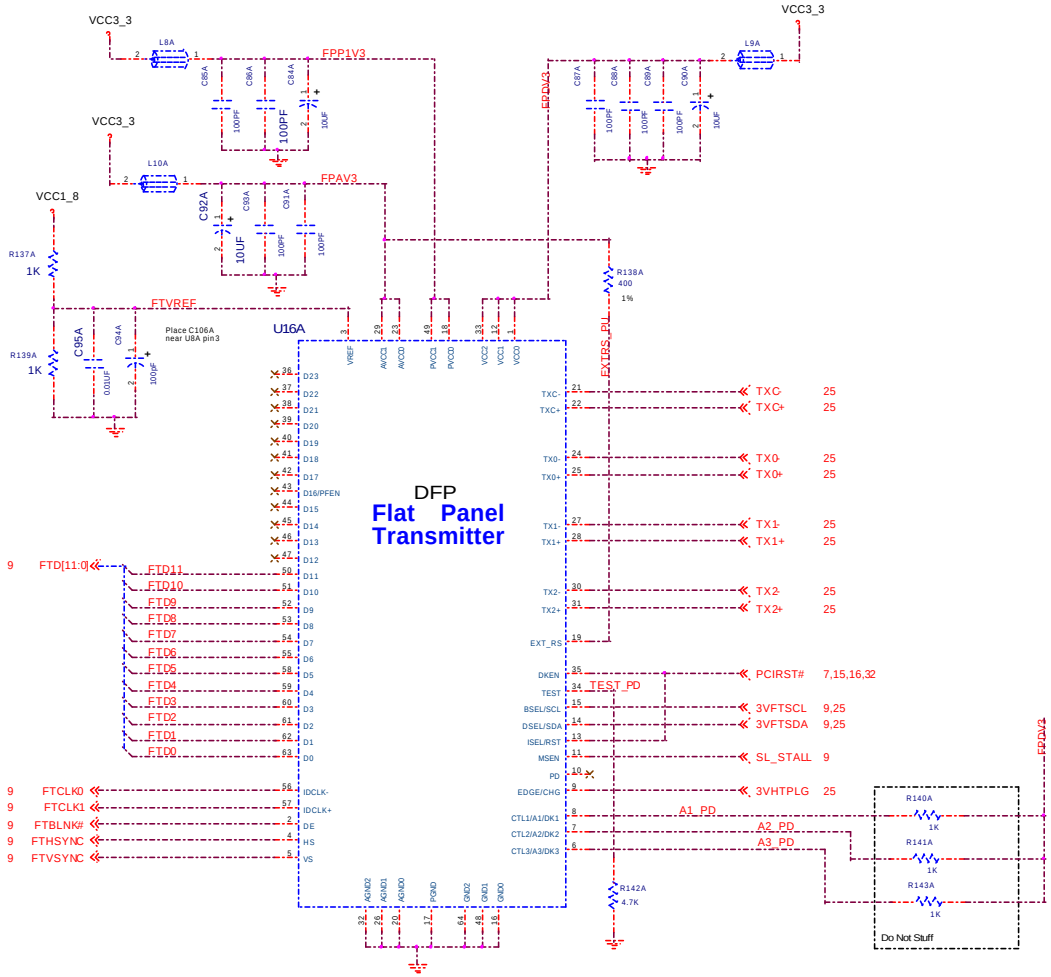


Keyboard
Mouse

IR

Title: Intel® 810e2 Chipset Universal Socket 370 CRB Kybrd / Mse / F. Disk / Gme Connectors		REV. 1.0
IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01	
	Sheet: 23 of 36	

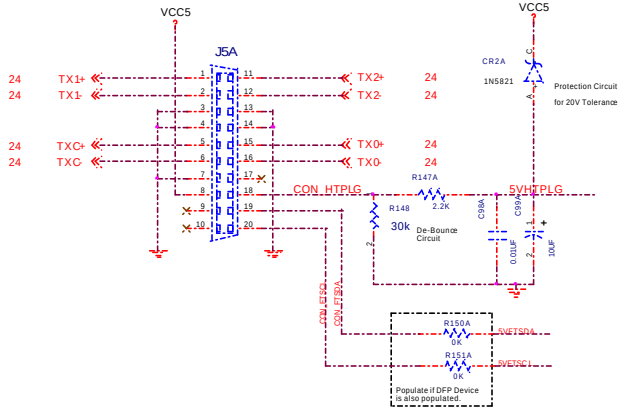
Digital Video Out



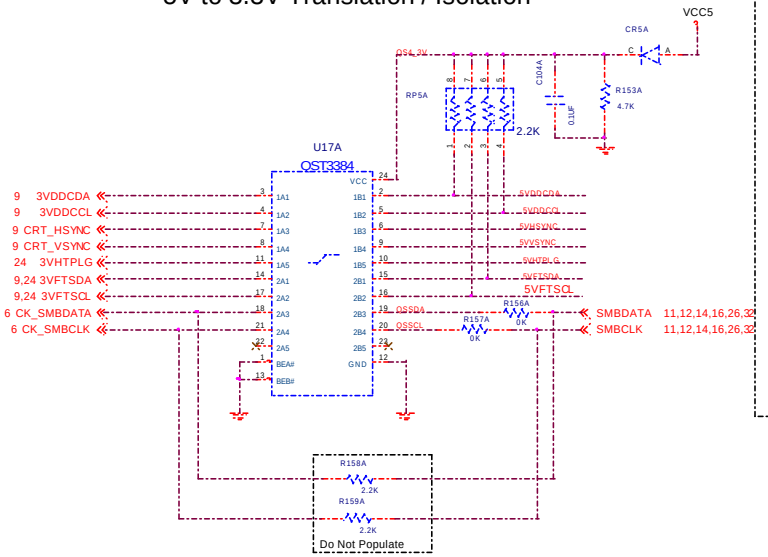
Title:	Intel® 810e2 Chipset Universal Socket370 CRB Digital Video Out	REV. 1.0
intel	IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date 9/02/01 Sheet 24 of 36

Video Connectors

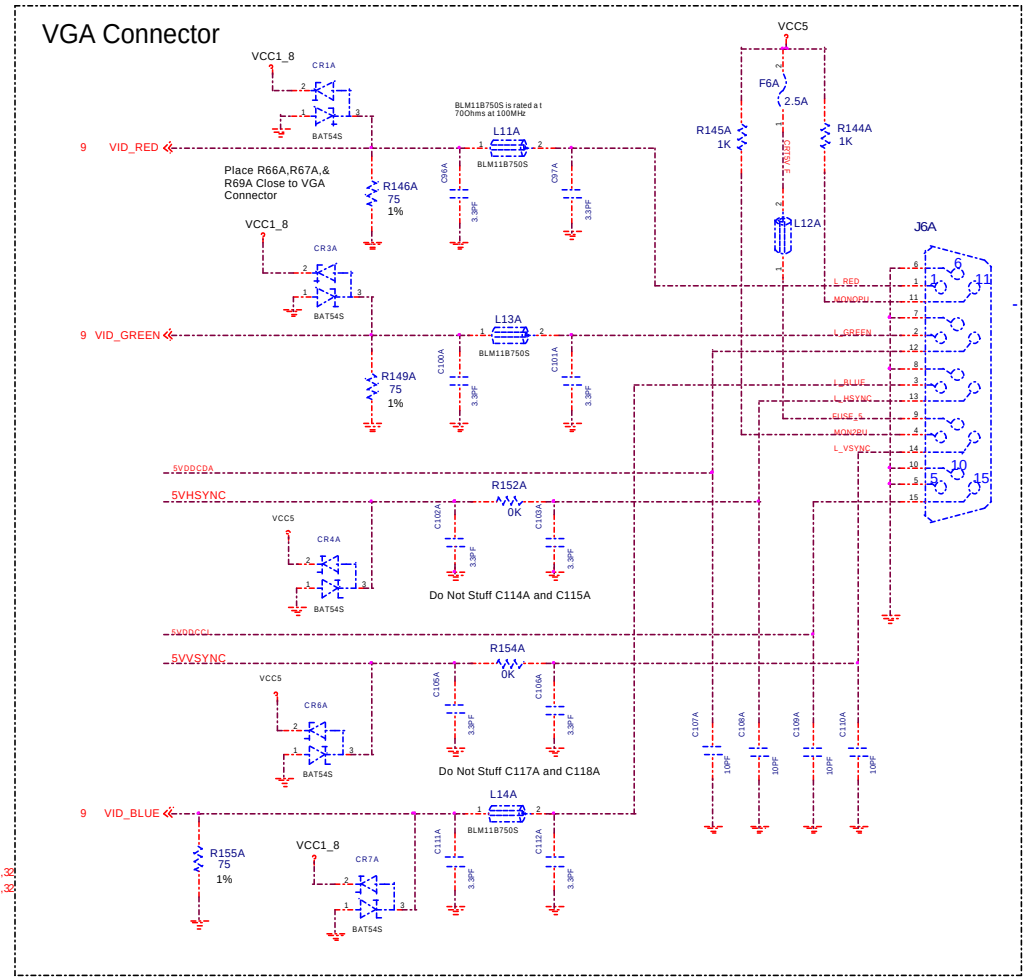
20 Pin Flat Panel Connector



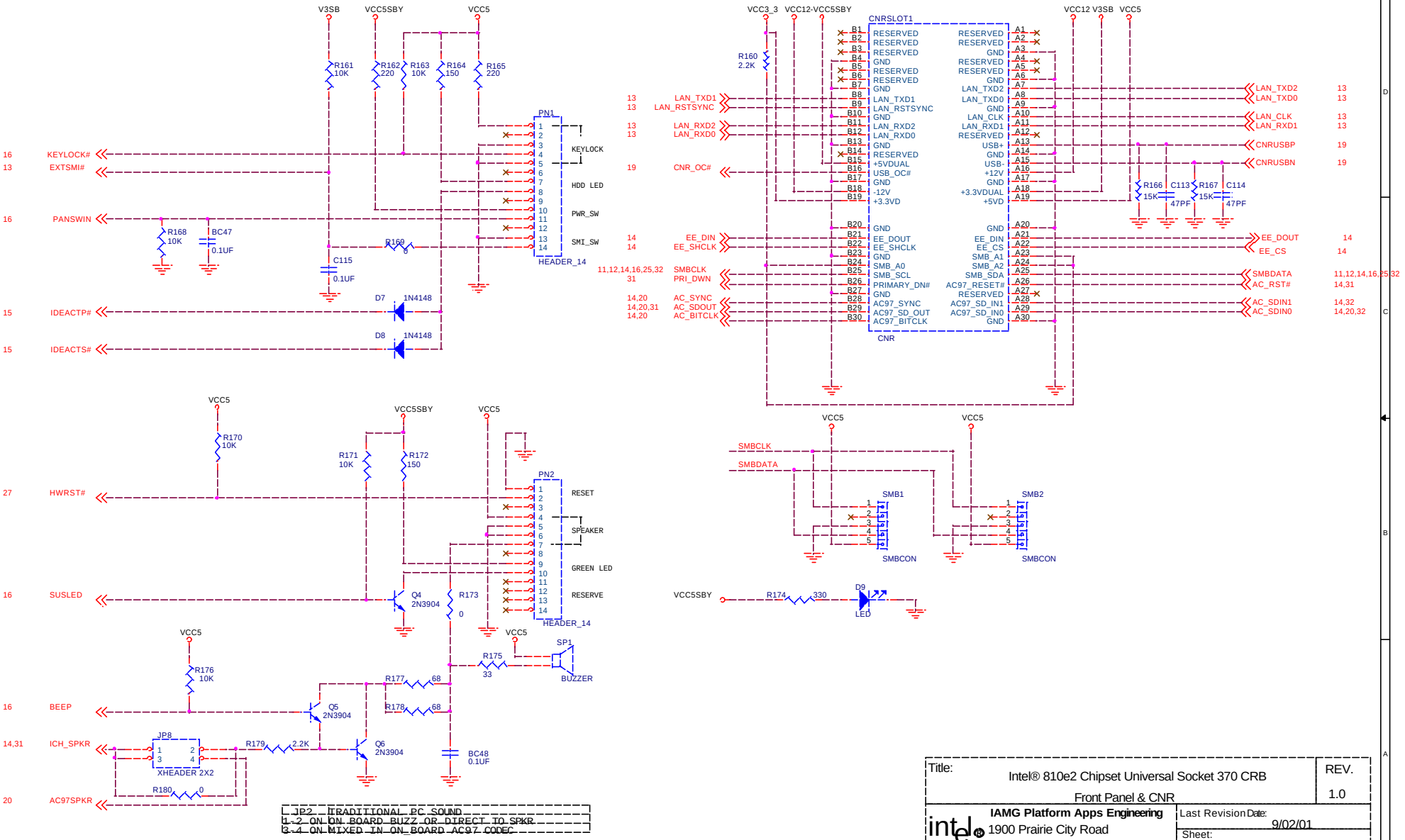
5V to 3.3V Translation / Isolation

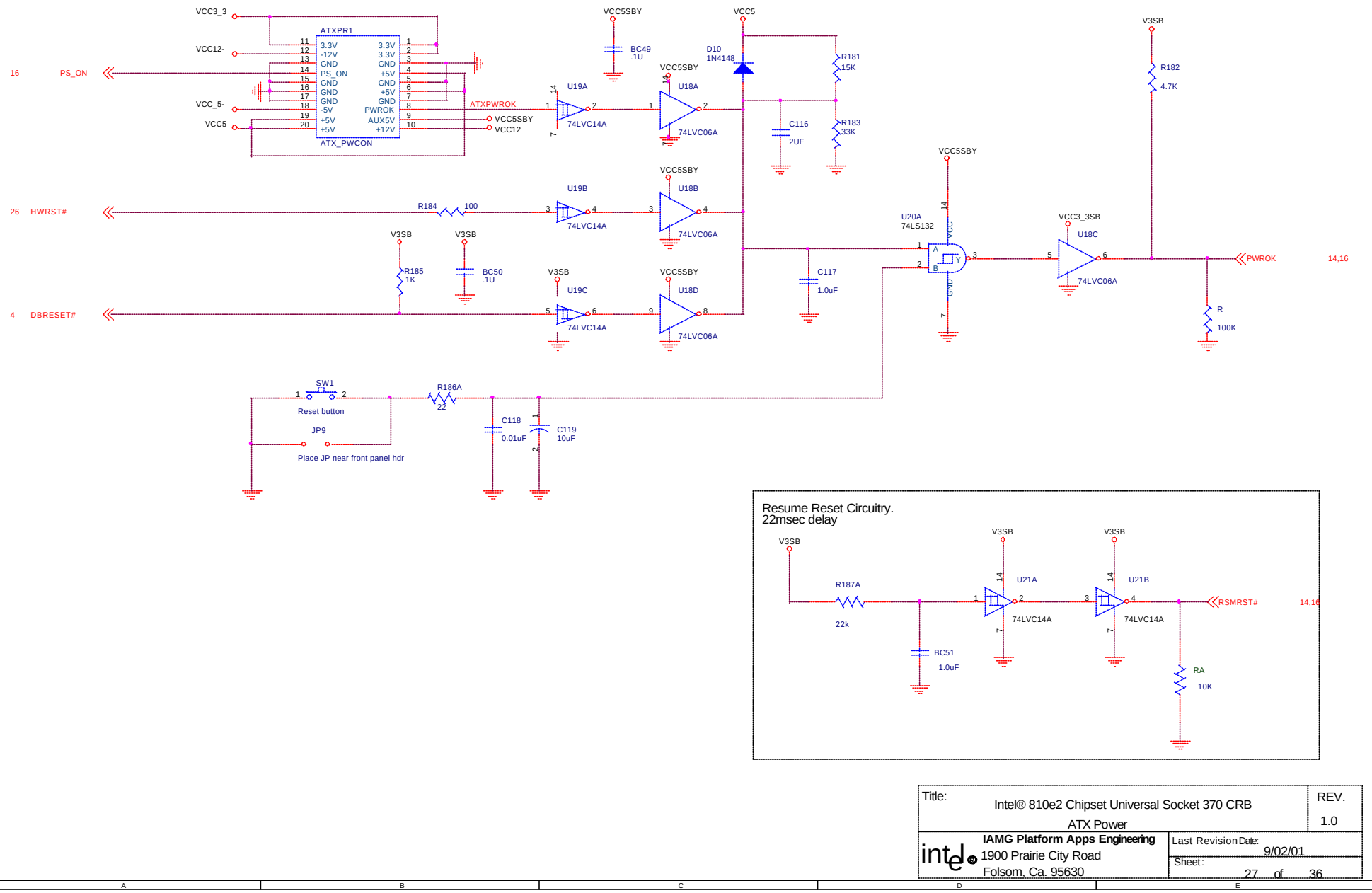


VGA Connector

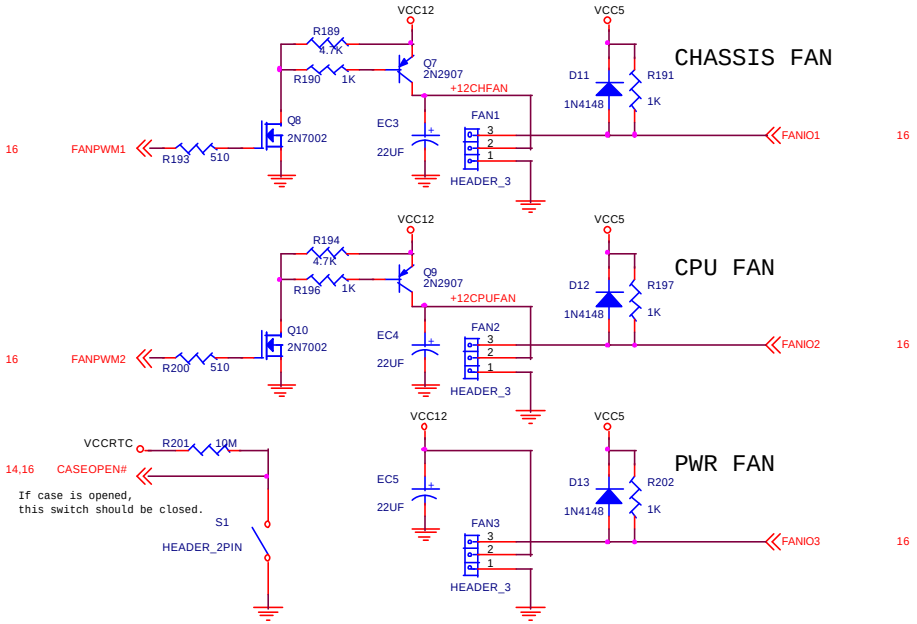


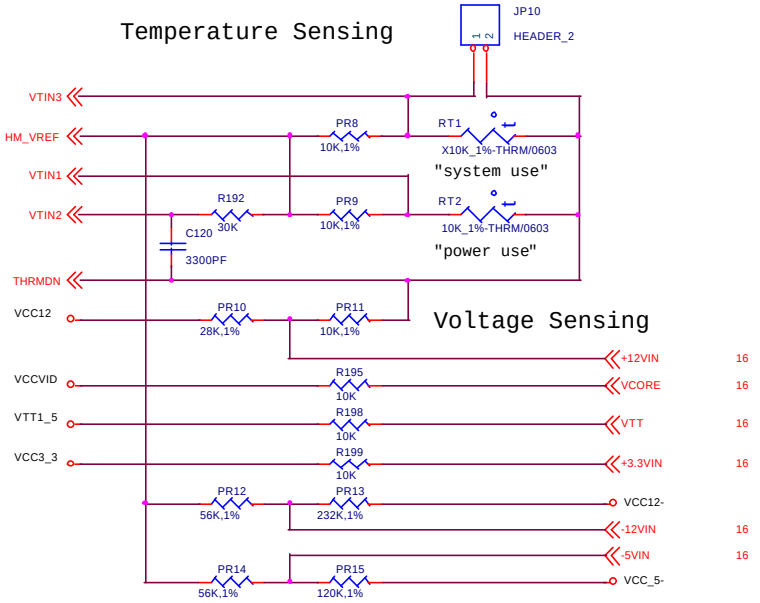
Title:	Intel® 810e2 Chipset Universal Socket370CRB Video Connectors	REV. 1.0
intel	IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date 9/02/01 Sheet 25 of 36


V3SB
VCC5SBY
VCC5
R161 10K
R162 220
R163 10K
R164 150
R165 220
PN1
KEYLOCK
HDD LED
PWR_SW
SMI_SW
HEADER_14
16 KEYLOCK#
13 EXTSMI#
16 PANSWIN
R168 10K
BC47 0.1UF
R169 0
C115 0.1UF
15 IDEACTP#
15 IDEACTS#
D7 1N4148
D8 1N4148
VCC3_3
VCC12
CNRSLOT1
R160 2.2K
13 LAN_TXD1
13 LAN_RSTSYNC
13 LAN_RXD2
13 LAN_RXD0
19 CNR_OC#
14 EE_DIN
14 EE_SHCLK
11,12,14,16,25,32 SMBCLK
31 PRI_DWN
14,20 AC_SYNC
14,20,31 AC_SDOUT
14,20 AC_BITCLK
CNR
LAN_TXD2 13
LAN_TXD0 13
LAN_CLK 13
LAN_RXD1 13
CNRUSBP 19
CNRUSBN 19
R166 15K
C113 47PF
R167 15K
C114 47PF
EE_DOUT 14
EE_CS 14
SMBDATA 11,12,14,16,25,32
AC_RST# 14,31
AC_SDIN1 14,32
AC_SDIN0 14,20,32
SMBCLK
SMBDATA
SMB1
SMB2
SMBCON
R170 10K
R171 10K
R172 150
PN2
RESET
SPEAKER
GREEN LED
RESERVE
27 HWRST#
16 SUSLED
Q4 2N3904
R173 0
R174 330
D9 LED
R175 33
SP1 BUZZER
R176 10K
R177 68
R178 68
16 BEEP
Q5 2N3904
14,31 ICH_SPKR
JP8
XHEADER 2X2
R179 2.2K
Q6 2N3904
BC48 0.1UF
R180 0
20 AC97SPKR
JP2 TRADITIONAL PC SOUND
1-2 ON ON BOARD BUZZ OR DIRECT TO SPKR
3-4 ON MIXED IN ON BOARD AC97 CODEC
Title:
Intel® 810e2 Chipset Universal Socket 370 CRB
Front Panel & CNR
REV.
1.0
IAMG Platform Apps Engineering
1900 Prairie City Road
Folsom, Ca. 95630
Last Revision Date:
9/02/01
Sheet:
26 of 36

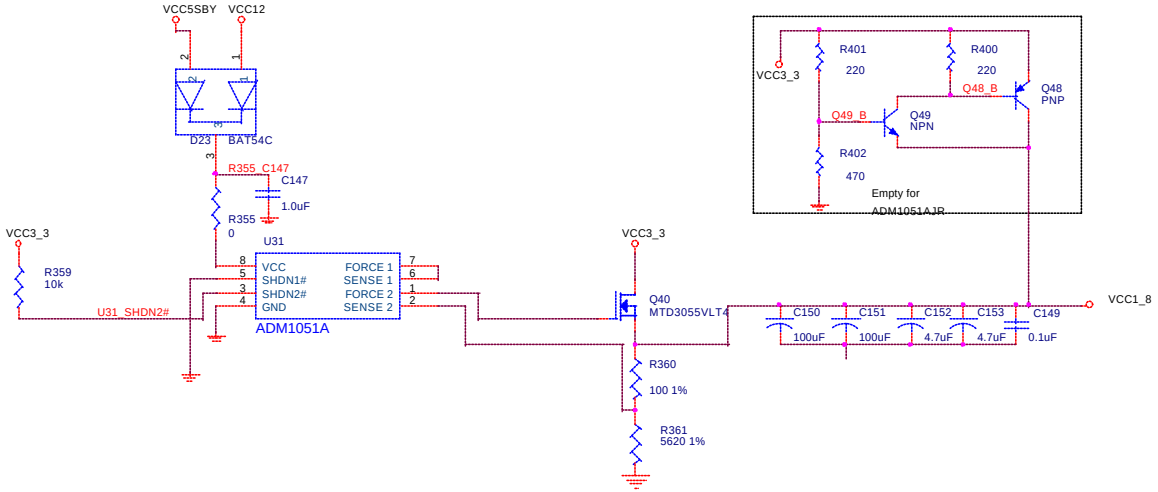


Title:	Intel® 810e2 Chipset Universal Socket 370 CRB ATX Power	REV. 1.0
intel **IAMG Platform Apps Engineering** 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 27 of 36	

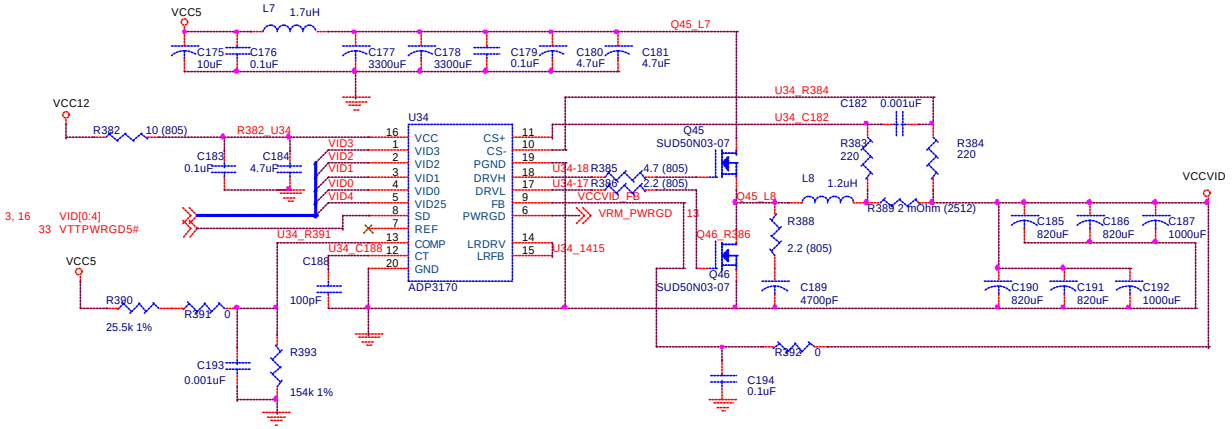


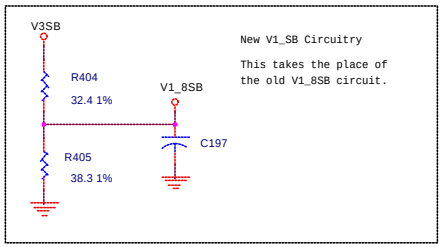


Title:	Intel® 810e2 Chipset Universal Socket 370 CRB H/W Monitor	REV. 1.0
intel	**IAMG Platform Apps Engineering** 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 28 of 36

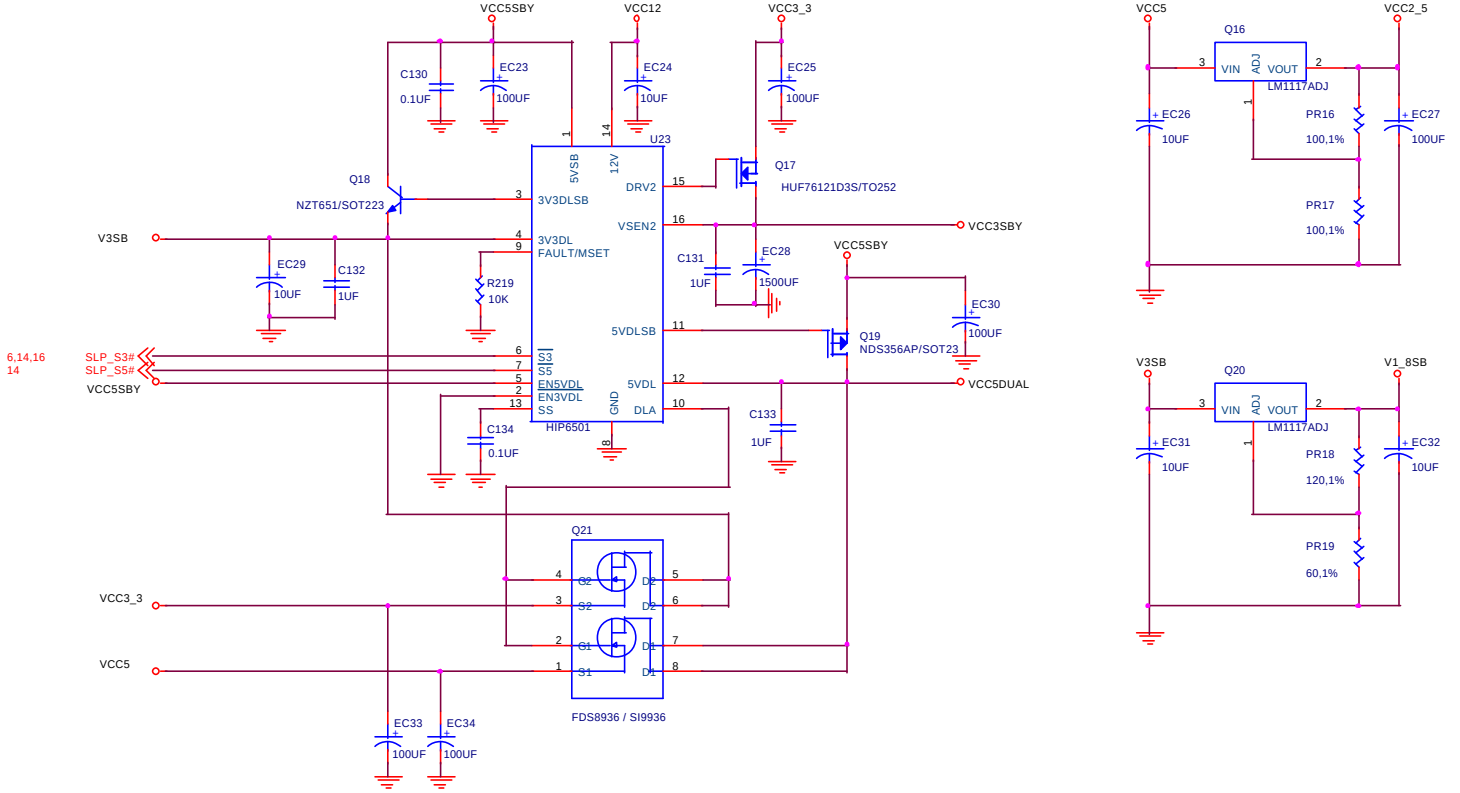


PLEASE SEE PAGE 33 for VTT GENERATOR

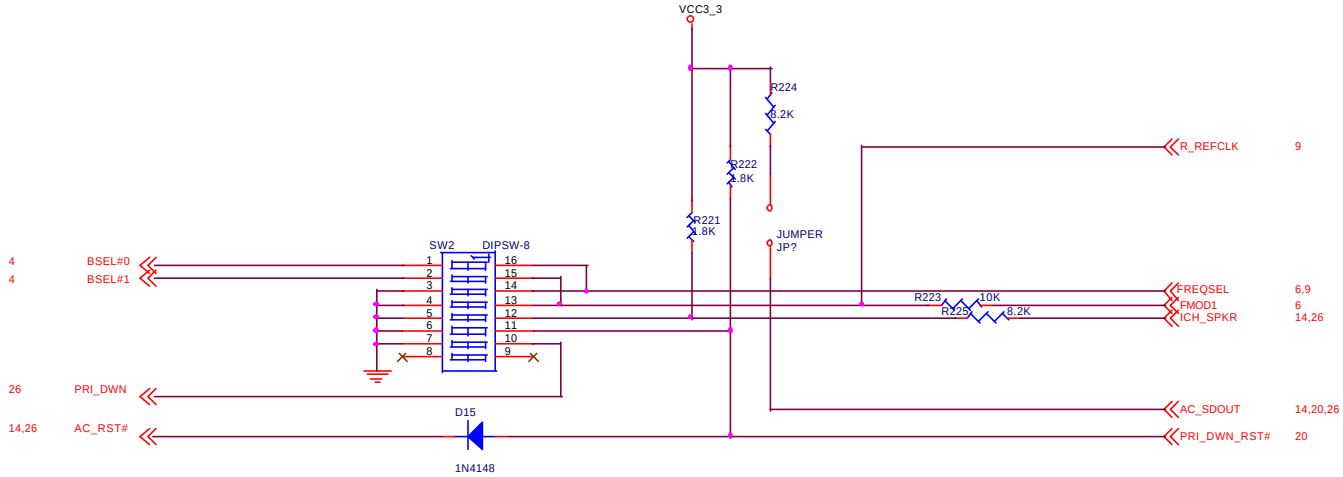




Title:	Intel® 810e2 Chipset Universal Socket 370 CRB Voltage Regulators Part 1	REV. 1.0
intel **IAMG Platform Apps Engineering** 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 29 of 36	



Title: Intel® 810e2 Chipset Universal Socket 370 CRB Voltage Regulators Part 2		REV. 1.0
IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01	
	Sheet: 30 of 36	



SW1:7-8	ON BOARD AC97 CODEC
ON 7	PRIMARY CODEC
ON 8	DISABLE

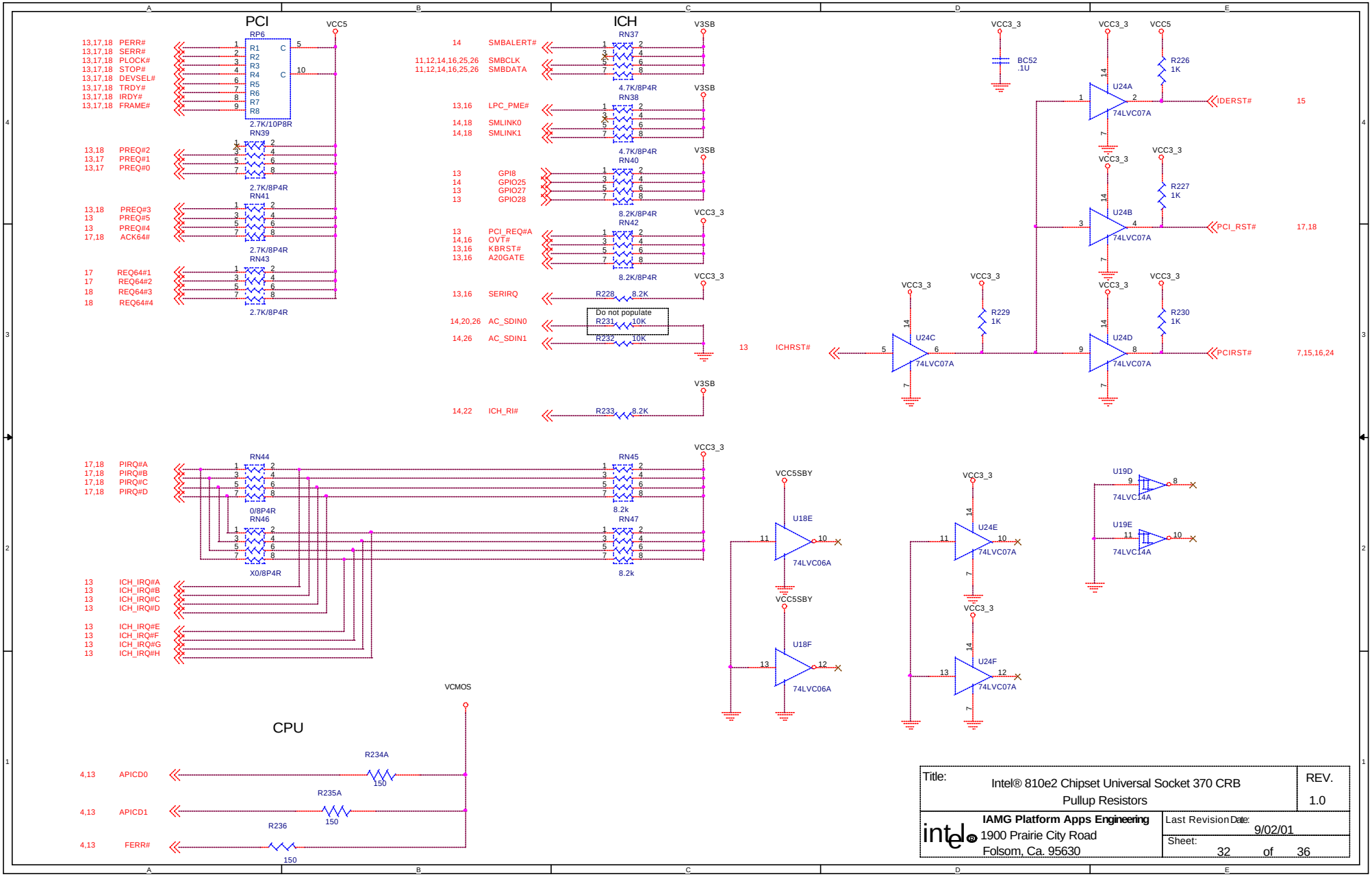
SW1:1-2	ESB / SYSTEM MEMORY
ON1-2	CPU DEFAULT
OFF 1-2	BY SW2:3-4

SW1:3-4	ESB / SYSTEM MEMORY
ON ON	66M/PC100
OFF ON	100M/PC100
OFF OFF	133M/PC100 OR PC133

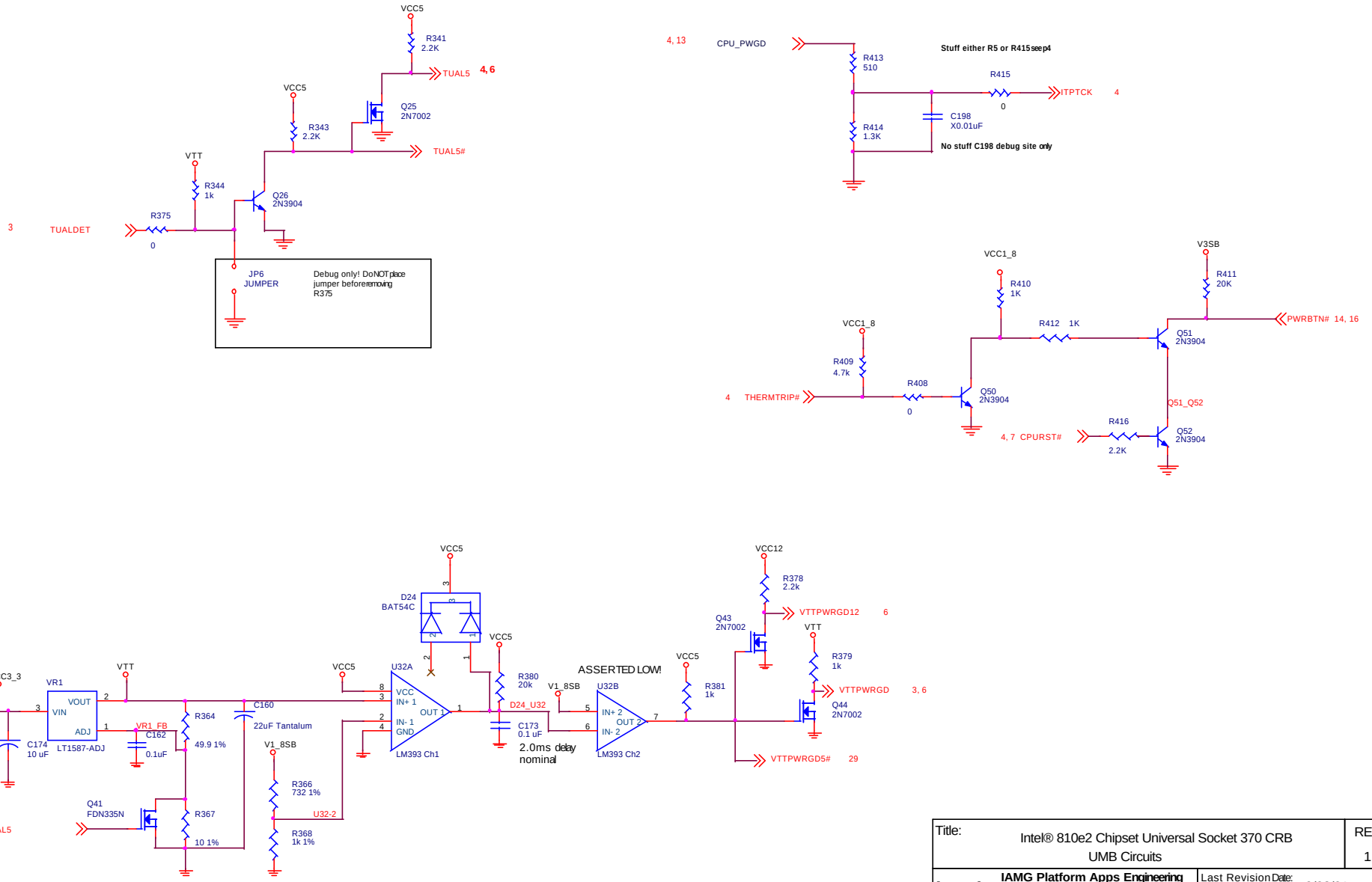
SW1:5	AC_SDOUT
ON	USE CPU FREQ STRAP IN ICH REGISTER
OFF	FORCE CPU FREQ STRAP TO SAFE MODE(1111)

SW1:6	STRAP(SPKR)
ON	NO REBOOT ON 2ND WATCHDOG TIMEOUT
OFF	REBOOT ON 2ND WATCHDOG TIMEOUT

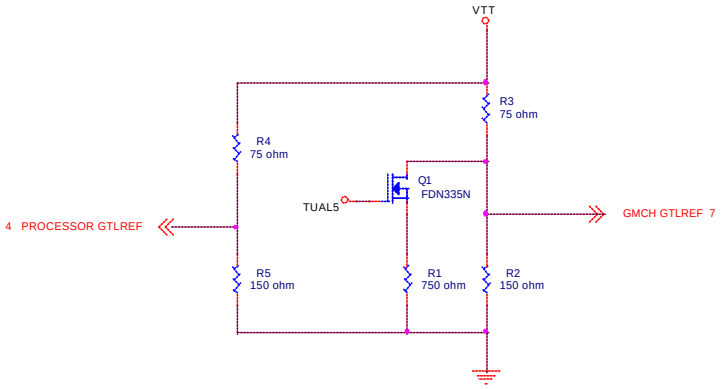
Title:	Intel® 810e2 Chipset Universal Socket 370 CRB System Configuration	REV. 1.0
intel	**IAMG Platform Apps Engineering** 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 31 of 36

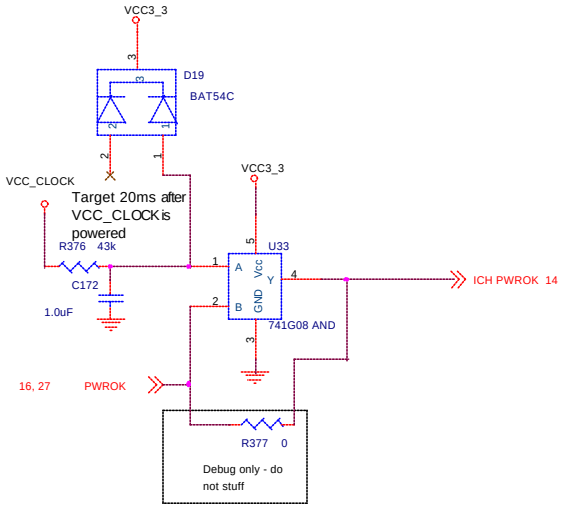


Title:	Intel® 810e2 Chipset Universal Socket 370 CRB Pullup Resistors
REV.	1.0
IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 32 of 36



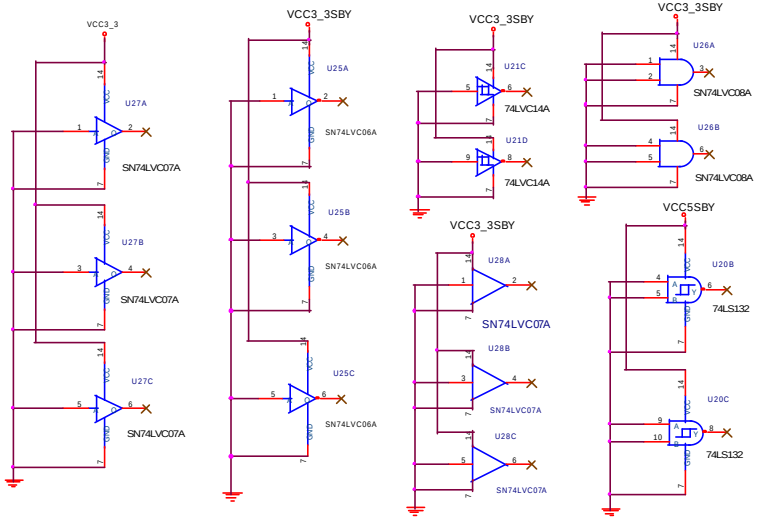
Title:	Intel® 810e2 Chipset Universal Socket 370 CRB UMB Circuits	REV. 1.0
intel **IAMG Platform Apps Engineering** 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01 Sheet: 33 of 36	



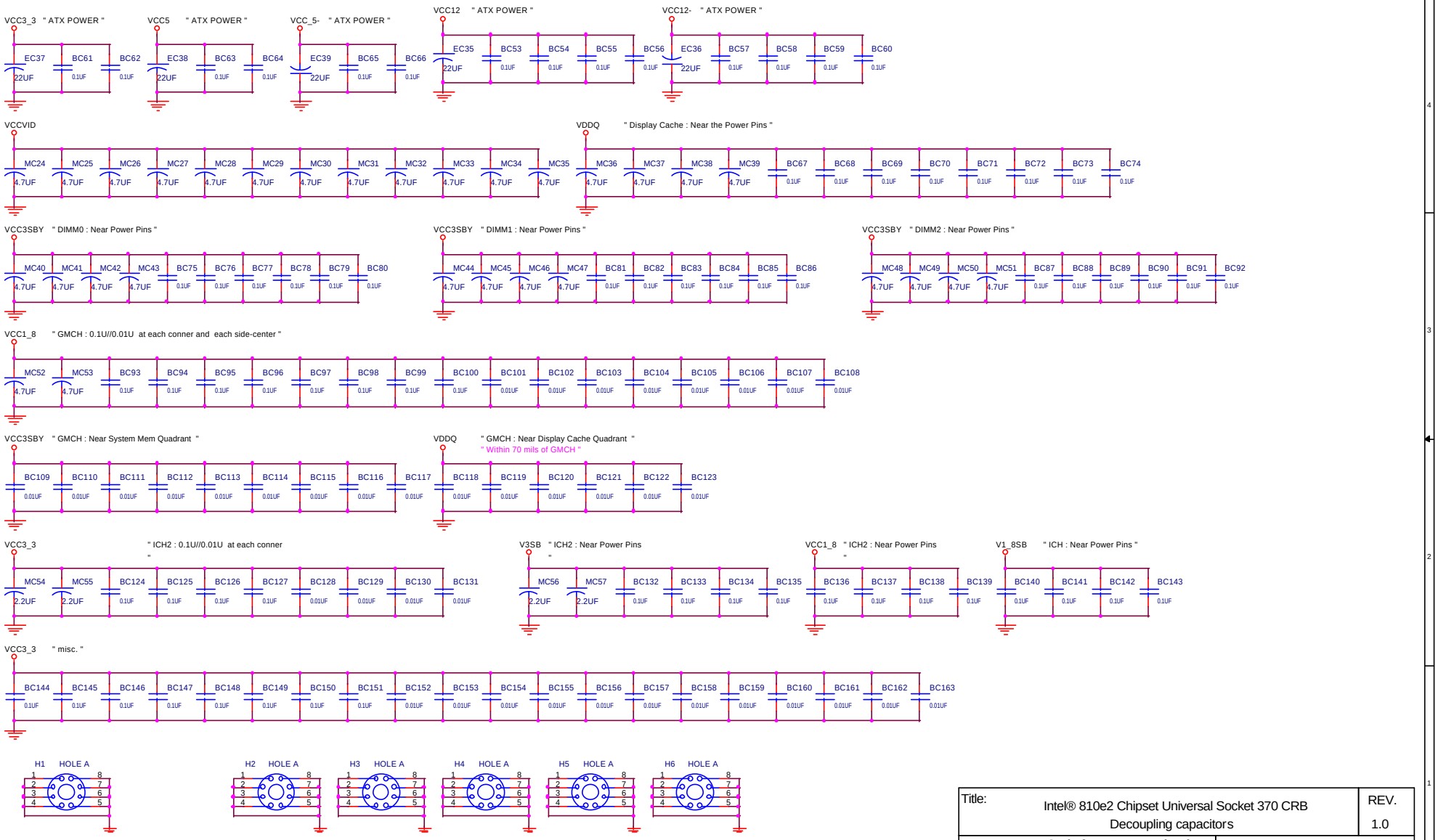


Title: Intel® 810e2 Chipset Universal Socket 370 CRB UMB Circuits		REV. 1.0
IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01	
	Sheet: 34 of 36	

UNUSED GATES



Title:	Intel® 810e2 Chipset Universal Socket370 CRB Unused Gates	REV. 1.0
intel	IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date 9/02/01 Sheet: 35 of 36



Title: Intel® 810e2 Chipset Universal Socket 370 CRB Decoupling capacitors		REV. 1.0
intel® IAMG Platform Apps Engineering 1900 Prairie City Road Folsom, Ca. 95630	Last Revision Date: 9/02/01	
	Sheet: 36 of 36	