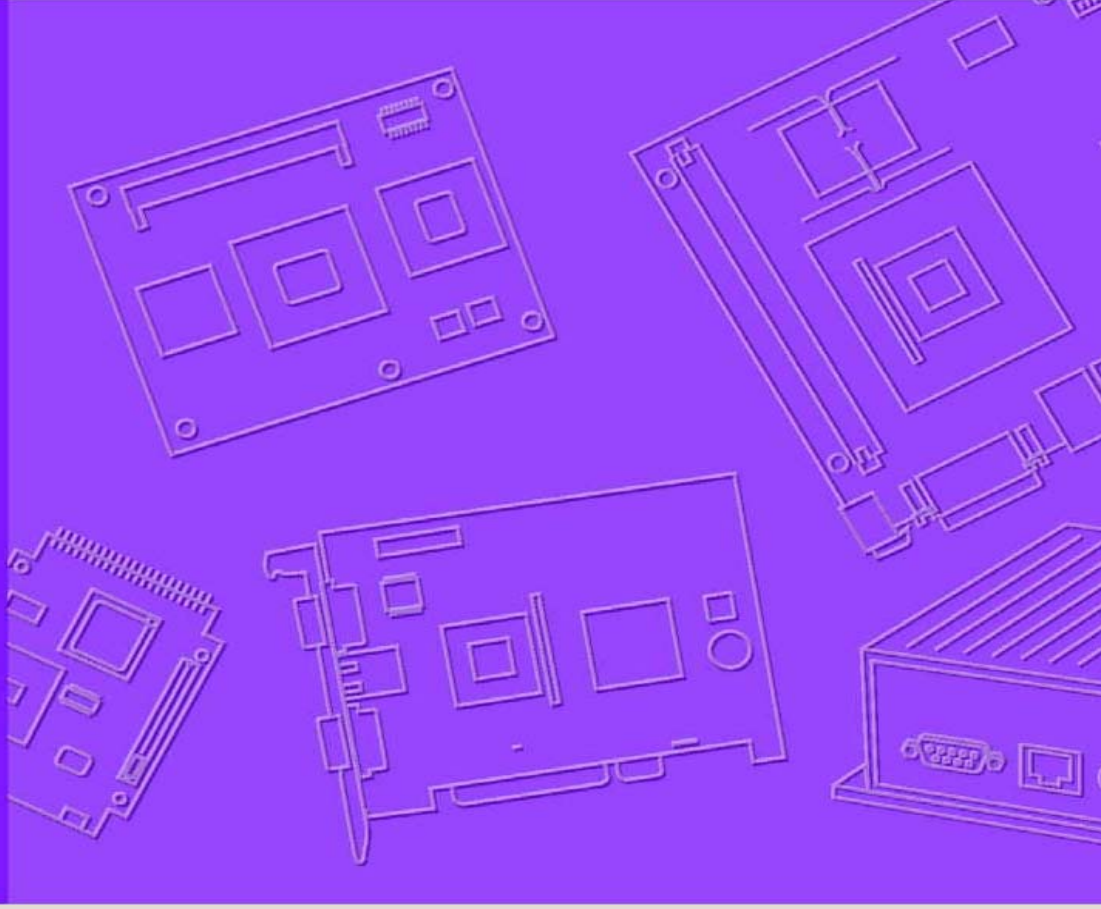
Design Guide



COM-Express

Carrier Board Design Guide

Notices

The product specifications, design and this user's manuals content are subject to change without notice.

If you have any questions, please contact your merchant or our service center for clarification. We are not responsible for any losses resulting from using this product no matter what the reason.

Revision History

Version/Revision date	Date	Description
Version 1.0	**July, 2006**	**Initial release**
Version 1.1	**July, 2007**	
Version 1.2	**April, 2009**	**Change format to Addendum**
Version 2.0	**May, 2010**	
Version 2.1	**August, 2010**	

Part No. 2006001801	Edition 2.1
Printed in Taiwan	August 2010

Table of Contents

Chapter 1 Introduction

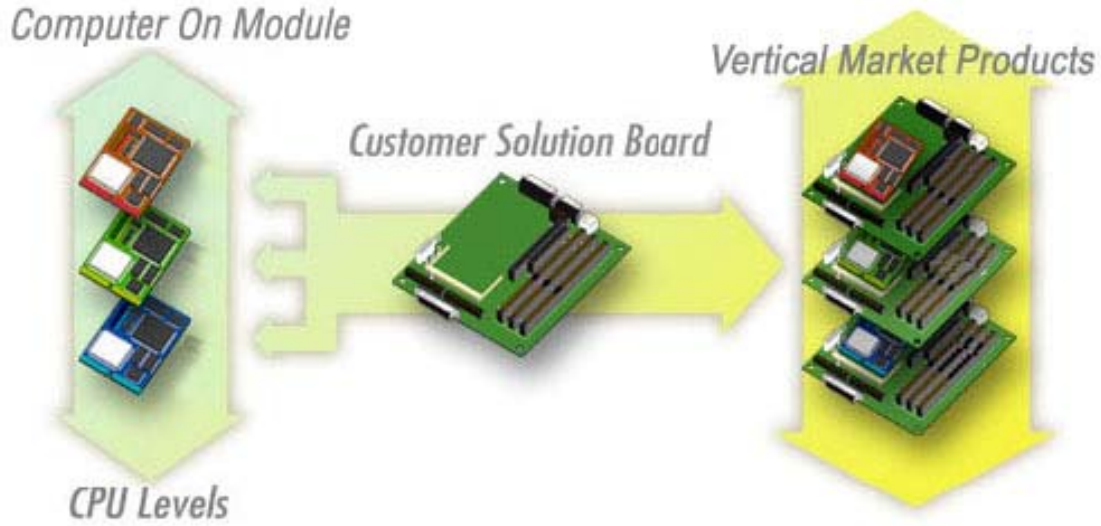
This design guide addendum organizes and provides Advantech's COM carrier board design recommendations for COM-Express Modules. All other Schematic Guidelines for the carrier board are applicable and can be found in the PICMG Design Guide V1.0.

Please contact Advantech sales/application engineer if there are any questions about designing the carrier board, or you plan to use this processor in applications other than mobile or desktop platforms.

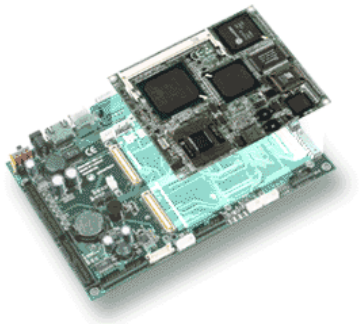
1.1 COM-Express Overview

COM-Express complies with COM Express standard from the PCI Industrial Computer Manufacturers Group (PICMG) which provides next generation performance of the smallest state of the art embedded modules. Advantech's COM Design Support Services (CDSS) help customers develop and integrate their carrier board with Advantech's COM modules. CDSS provides a series of valuable services such as Product, Design Assistance, Software and Thermal Solution services, together they help reduce design risks when designing carrier boards. For more details, please visit "http://com.advantech.com"

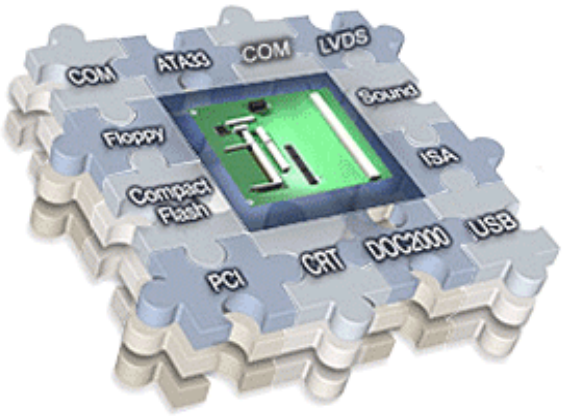
Advantech offers a wide range of COM products to cater to each customer's demands. The modular designs add more flexibility to the system. The COM Express form factor allows the COM-Express Modules to be easily and securely mounted on a customized solution board. The design and multiple processor choices eliminate CPU integration worries and allow fast application support for the most dynamic embedded needs.



COMs are widely used modular CPU boards with high integration features. COM-Express supports a wide range of processor and chipsets. They include technologies like: PCI Express, Serial ATA, USB 2.0, etc. Not only do COMs allow quick design, they also provide the benefits of easy installation, maintenance and upgrade ease.



Though small in size, COMs implement CPU architectures and basic common circuits. Many system integrators find that Advantech COM solutions already cover around 80% of their feature requirements. This makes COM products powerful time and cost savers.



1.2 Terminology

Table 1.1 Conventions and Terminology	
AC'97	Codec 97' Audio interface
COM	A serial port interface on IBM PC-compatible computers running Microsoft Windows or MS-DOS
COM-Express	New generation technology of Computer On Module
CPU	Central processing Unit
CRT	Cathode Ray Tube
DDR2	Double Data Rate second generation SDRAM memory technology
DVI	Digital Visual Interface
DVO	Digital Video Out
EMI	Electromagnetic Interference
ESD	Electrostatic Discharge
ExpressCard	A hardware standard replacing PC cards supported both PCI Express and USB 2.0 connectivity
FSB	Front Side Bus, synonymous with Host or CPU bus
GMCH	Refers to the Graphics Memory Controller Hub chipset component
HD Audio	High Definition Audio

I2C	Inter-IC (a two wire serial bus created by Philips)
IDE (ATA)	Integrated Drive Electronics (Advanced Technology Attachment)
INTx	An interrupt request signal where x stands for interrupts A, B, C, and D.
LAN	A local area network (LAN) is a computer network covering a small physical area
LCD	Liquid Crystal Display
LPC	The Low Pin Count (LPC) Interface Specification for legacy I/O has facilitated the industry's transition toward ISA-less systems.
LVDS LCD	Low Voltage Differential Signaling: A high speed, low power data transmission standard used for display connections to LCD panels.
MCH	Refers to the Memory Controller Hub chipset component
NTSC	National Television Standards Committee
PAL	Phase Alternate Line
PCI	Peripheral Component Interface
PCI-Express	New generation PCI interface with serial interconnection technology
RTC	Real Time Clock
SATA	Serial ATA interface
SDVO	Serial Digital Video Out
SMBus	System Management Bus.
TMDS	Transition Minimized Differential Signaling
TV	Television supports NTSC and PAL
UART	A universal asynchronous receiver/transmitter that translates data between parallel and serial forms.
USB	Universal Serial Bus

1.3 Referenced Documents

Table 1.2 Referenced Documents	
Document	**Location**
ACPI - Advanced Configuration and Power Management Specification	http://www.acpi.info/
AC'97 AUDIO	http://download.intel.com/support/motherboards/desktop/sb/ac97_r23.pdf
APM - Advanced Power Management Specification	http://www.microsoft.com/whdc/archive/amp_12.mspx
COM Express Specification	http://www.picmg.org/
Ethernet(IEEE 802.3)	http://standards.ieee.org/getieee802/802.3.html
I2C Bus Interface	http://www.semiconductors.philips.com/buses/i2c
PCI	http://www.pcisig.com/
RS232	http://www.eia.org/
SMBus	http://www.smbus.org/specs/
USB	http://www.usb.org/home

Chapter 2 Carrier Board Schematic Guidelines

2.1 Gigabit Ethernet (GBE)

COM-Express supports the IEEE802.3 network interface and flexible dynamically loadable EEPROM algorithm. The network interface complies with the IEEE standard for 10BASE-T, 100BASE-T and 1000BASE-T Ethernet interfaces.

2.1.1 Signal Descriptions

Table 2-1 shows COM-Express Ethernet signals, including pin number, signal naming, I/O, and descriptions.

Table 2-1 GBE signals

Pin	Signal	I/O	Description
A13,A9,A7,A3 A12,A10,A6,A2	GBE0_MDI[0:3]+ GBE0_MDI[0:3]-	I/O	Gigabit Ethernet Controller 0: Media Dependent Interface Differential Pairs 0,1,2,3. The MDI can operate in 1000, 100 and 10 Mbit / sec modes. Some pairs are unused in some mode, per the following: 1000BASE-T 100BASE-TX 10BASE-T MDI[0]+/- B1_DA+/- TX+/- TX+/- MDI[1]+/- B1_DB+/- RX+/- RX+/- MDI[2]+/- B1_DC+/- MDI[3]+/- B1_DD+/-
B2	GBE0_ACT#	OD	Gigabit Ethernet Controller 0 activity indicator, active low.
A8	GBE0_LINK#	OD	Gigabit Ethernet Controller 0 link indicator, active low.
A4	GBE0_LINK100#	OD	Gigabit Ethernet Controller 0 100 Mbit / sec link indictor, active low.
A5	GBE0_LINK1000#	OD	Gigabit Ethernet Controller 0 1000 Mbit / sec link indictor, active low.
A14	GBE0_CTREF	REF	Reference voltage for Carrier board Ethernet channel 0 magnetics center tap. The reference voltage is determined by the requirements of the module PHY and may be as low as 0 V and as high as 3.3 V. The reference voltage output shall be current limited on the module. In the case in which the reference is shorted to ground, the current shall be limited to 250 mA or less.

2.1.2 DC Specifications

Table 2-2 GBE I/O Voltage

Symbol	Parameter	Min	Max	Unit	Note
V_{IL}	Input Low Voltage	--	0.9	V	*1
V_{IH}	Input High Voltage	1.7	--	V	
V_{OL}	Output Low Voltage	0	0.5	V	
V_{OH}	Output High Voltage	2.1	3.6	V	

Note *1. Vcc is 3.3V (from 3.0V min. to 3.6V max.)

2.1.3 Schematic Guidelines

2.1.3.1 Differential Pairs

Designing for Gigabit Ethernet operation is very similar to the designing for 10/100 Mbps. 10/100Mbps Ethernet has two differential pairs, and Gigabit Ethernet has four differential pairs. Figure 2-1 and Figure 2-2 show the 10/100M Ethernet and Gigabit Ethernet Connections.

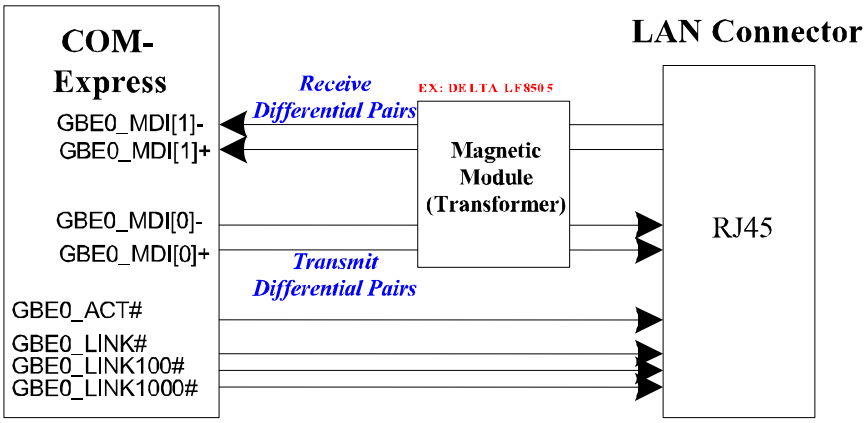


Figure 2-1 10/100M Ethernet Connections

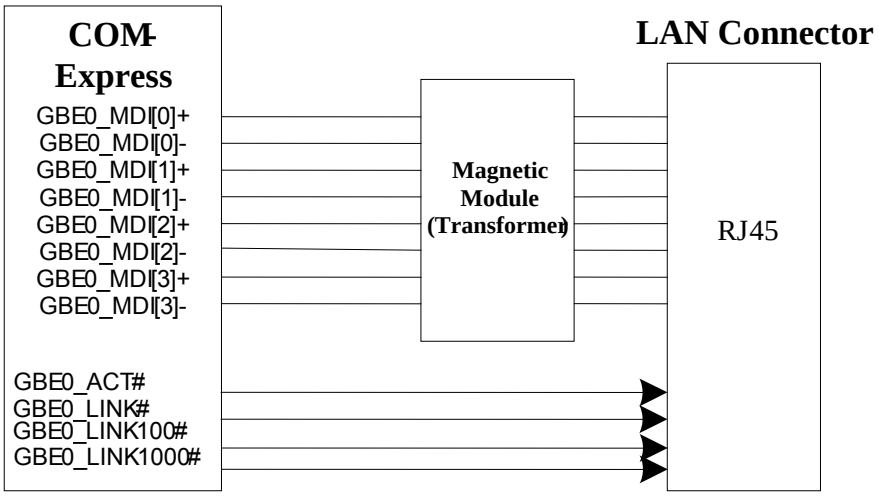


Figure 2-2 Gigabit Ethernet Connections

2.1.3.2 Center Tap connection of Transformer

The COM-E signal GBE0_CTREF pin A14 should be connected directly to the center tap of the transformer. The center tap voltage is the output from the COM-Express

Module to the carrier board usually 1.8V, 2.5V, or another voltage, depending on the LAN chip design.

2.1.3.3 LAN Connector with Integrated Magnetic

For simplifying the schematic and layout considerations of LAN connector, it is strongly recommended to use the RJ45 LAN connector. Figure 2-3 shows the integrated magnetic schematic.

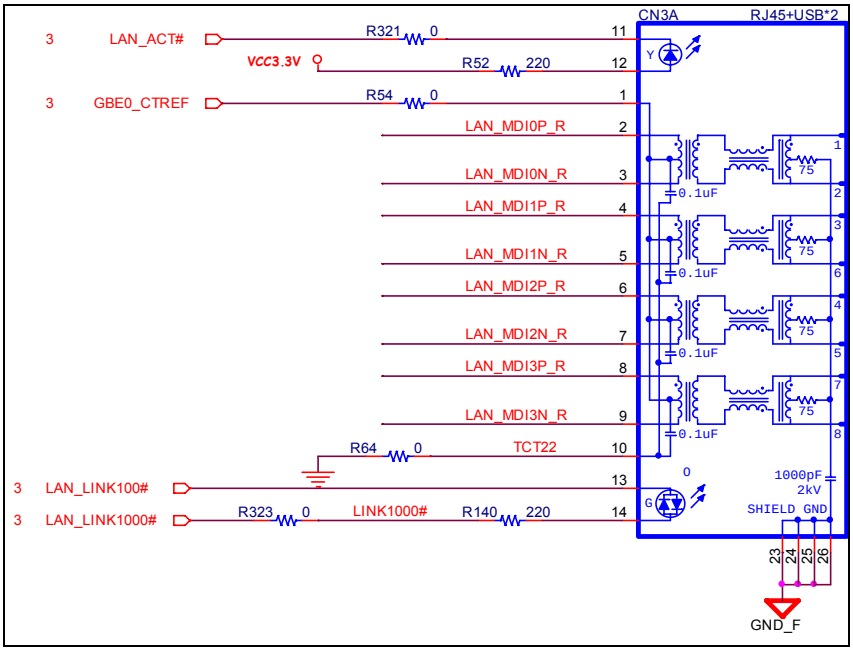


Figure 2-3 Gigabit Ethernet Connections with Integrated Magnetic

2.1.3.4 Implementation of Ethernet LED indicators

RJ-45 connector with LED indicators needs 3.3V to drive the LEDs. The Link and activity LEDs can be implemented by using the COM-Express Module's GBE0_ACT#, GBE0_LINK#, GBE0_LINK100#, and GBE0_LINK1000# pins. The sink current is connected to the cathode of the LED, and the anode of the LED should be pulled to 3.3V through a resistor as 220 Ω.

2.1.4 Layout Guidelines

Route the transmit and receive lines on the carrier board as differential pairs, with a differential impedance of 100 Ω. PCB layout software allows determination of the proper trace width and spacing to achieve the impedance after the PCB stack-up configuration.

The TX+/TX- signal pair should be well separated from the RX+/RX- signal pair. Both pairs should be well separated from any other signals on the PCB. The total routing length of these pairs from the COM-Express Module to the Ethernet RJ45 connector should be made as short as practical.

For Ethernet connector placement, place it as close as possible to the COM-Express Module pins to shorten the routing lengths of all Ethernet signals. Differential signal traces should be kept as short as possible to decrease the possibility of being

affected by high frequency noise from other signals and power planes, and capacitive loading is also reduced.

Please refer to the Advantech layout checklist for detail.

2.1.4.1 Differential pairs design considerations

- 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. Do not use serpentines to try to match trace lengths in the differential pair. Serpentines cause impedance variations causing signal reflections, which can be a source of signal distortion. Try to keep the length difference of the differential pair less than 5 mil.

- The total length of each differential pair should be less than 4 inches. Keep the length of each differential pair under 4 inches. Figure 2-4 shows an example. Please refer to the Advantech layout checklist for detailed length matching.

- Do not route the transmit differential traces closer than 50 mils to the receive differential traces for 10/100 Mbps.

- Do not route any other signal traces (including other differential pairs) parallel to the differential traces or closer than 50 mils. to the differential traces. Figure 2-4 shows an example. It's recommended to keep length L3 longer than 50 mils.

- Keep separate traces within a differential pair as small as possible down to 5 to 8 Mils, depending on the impedance control. Close separation of the traces allow the traces to couple well to each other.

- For high-speed signals, they should minimize the number of corners and vias. If a 90° bend is required, it is recommended to use two 45° bends instead. Figure 2-5 shows the example.

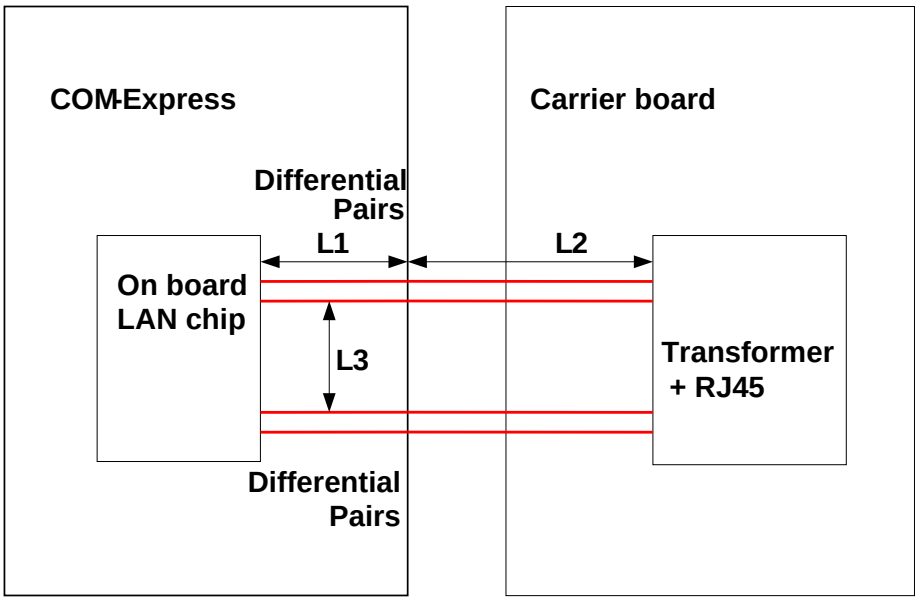


Figure 2-4 Differential signals route example

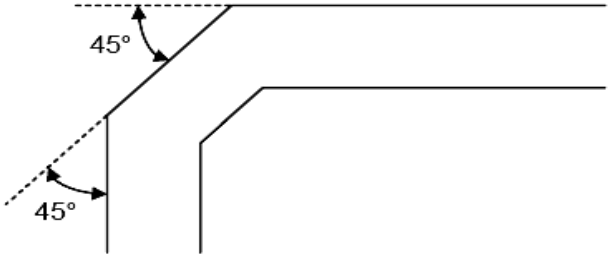


Figure 2-5 Bend layout example

2.1.4.2 Transformer

It's recommended to use the integrated Magnetic Modules/RJ-45 LAN connectors. If using the discrete Magnetic Modules and RJ-45 connector, the transformer should be placed close to the RJ-45 LAN connector to reduce EMI emissions. Each differential pair of data signals is required to be parallel to each other with the same trace length on the component (top) layer and to be parallel to a respective ground plane.The connector with integrated magnetic is much simplified for layout. The more complex layout as Figure 2-6 and Figure 2-7 shows the 10/100M and Gigabit Ethernet layout with discrete magnetic.

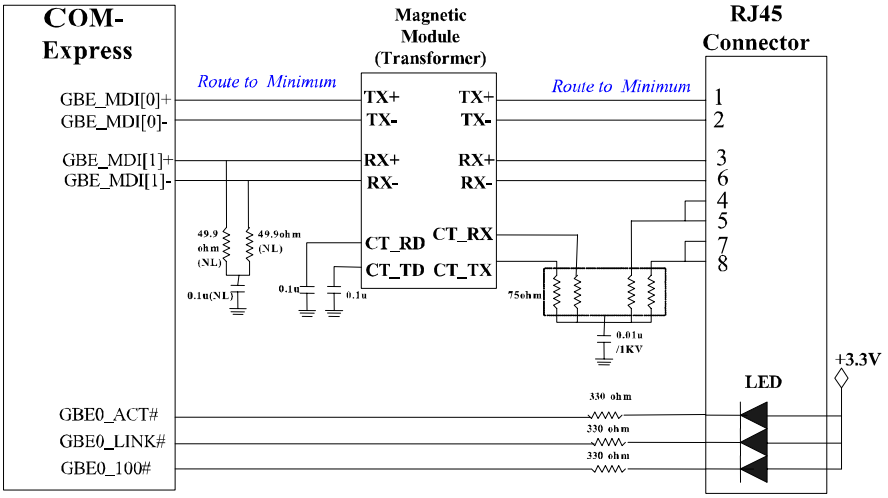


Figure 2-6 10/100M Ethernet Interconnection

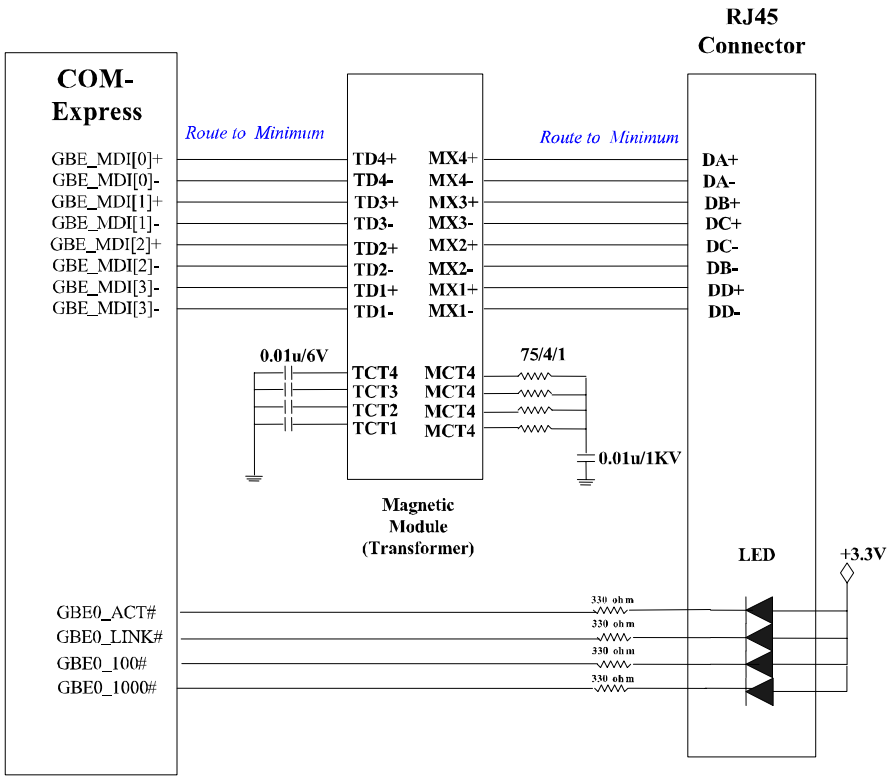


Figure 2-7 Gigabit Ethernet Interconnection

2.1.4.3 Power Considerations

In general, any section of traces that are intended for use with high-speed signals should observe proper termination practices. Many board layouts remove the ground plane underneath the transformer and the RJ-45 connector to minimize capacitive coupling of noise between the plane and the external Ethernet cable.

2.1.4.4 Critical Dimensions

There are two critical dimensions that must be considered during the layout phase of an Ethernet controller. These dimensions are identified in Figure 2-8 as distance A and B.

Distance A: Transformer to RJ-45 LAN Connector (Priority 1). The distance labeled A should be given the highest priority in the backplane layout. The distance between the transformer module and the RJ-45 connector should be kept to less than 1 inch of separation. The following trace characteristics are important and should be observed:

1. 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.

2. Trace Symmetry: Differential pairs should be routed with consistent separation and with exactly the same lengths and physical dimensions (for example, width and spacing).

Distance B: From COM-Express Module to Transformer (Priority 2). Distance B should also be designed to be as short as possible. Be sure not to route Distance B

over 2.5 inches for Intel's layout guide. The high-speed signals propagating through these traces require the shortest distances between these components.

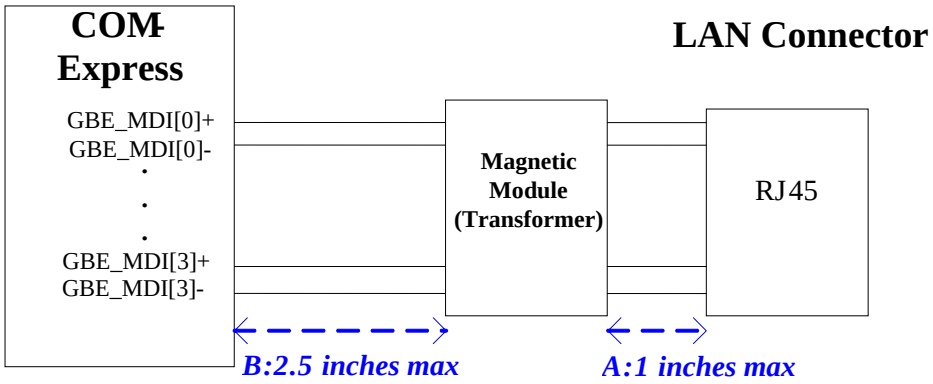


Figure 2-8 Critical Dimensions

2.2 AC'97 Audio / HD Audio

COM-Express provides an AC'97 Audio / HD Audio interface which is compliant to AC'97 AUDIO Rev. 2.3 Specification and the HD Audio Specification. Please establish the CODEC of AC'97 Audio / HD Audio on the carrier board for your application.

2.2.1 Signal Description

Table 2-3 shows COM-Express AC'97 Audio / HD Audio interface signals, including pin number, signals, I/0 and descriptions.

Table 2-3 Audio signals description

Pin	Signal	I/O	Description
A30	AC_RST#	O	**AC'97 Audio / HD Audio :** Reset output to AC97 CODEC, active low
A29	AC_SYNC	O	**AC'97 Audio / HD Audio :** 48 kHz fixed-rate, sample-synchronization signal to the CODEC(s)
A32	AC_BITCLK	I/O	**AC'97 Audio:** Bit Clock Input: This signal is a 12.288 MHz serial data clock generated by the external codec(s). This signal has an integrated pull-down resistor **HD Audio:** Bit Clock Output: This signal is a 24 MHz serial data clock generated by COM-Express. This signal has an integrated pull-down resistor so that AC_BITCLK does not float when an HD Audio codec (or no codec) is connected but the signals are temporarily configured as AC'97 AUDIO.
A33	AC_SDOUT	O	**AC'97 Audio / HD Audio :** Serial TDM data output to the CODEC
B30,B29, B28	AC_SDIN[0:2]	I	**AC'97 Audio / HD Audio :** Serial TDM data inputs from up to 3 CODECs

2.2.2 DC Specifications

Table 2-4 AC'97 AUDIO CODEC DC specification

Symbol	Parameter	Min	Max	Unit	Note
Dvdd	Digital supply voltage	Dvdd+5%	Dvdd+5%	V	
Avdd	Analog supply voltage	4.75	5.25	V	
Vil	Input Low Voltage	-	0.35Vdd	V	
Vih	Input High Voltage	0.65Vdd	-	V	

*1. Dvdd=5V or 3.3V

Table 2-5 AC'97 AUDIO CODEC analog I/O DC specification

Symbol	Parameter	Min	Typ	Max	Unit	Note
AUXAL/R	Full scale input voltage	-	1.0	-	Vrms	
MIC	Full scale input voltage	-	0.1	-	Vrms	
SNDL/R	Full scale output voltage	-	1.0	-	Vrms	

2.2.3 AC'97 Audio Spec

Refer to "Audio Codec '97 Revision 2.1 May 22, 1998" Chapter 9 for digital signals AC spec. and Chapter 10 for analog performance spec.

2.2.4 Schematic Guidelines

AC97 and HD Audio are the popular architectures for implementing audio, modem, and communications functionality in the IPC market. The architecture of the COM-Express HD Audio allows a maximum of three CODECs to be connected.

2.2.4.1 Connection of AC'97 Audio and HD Audio

2.2.4.2 AC'97 Audio:

The Figure 2-9 shows the connections for COM-Express AC'97 Audio signals with three Codecs.

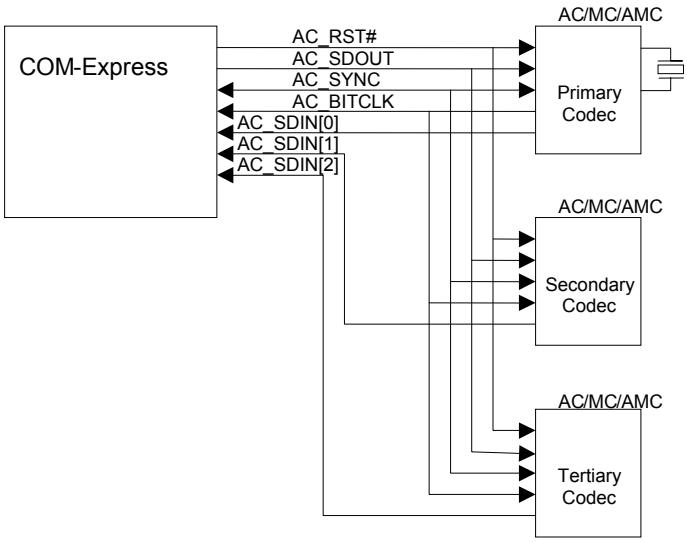


Figure 2-9 AC'97 Audio Connections

The clock is provided from the primary codec on the link via AC_BITCLK, and it is derived from a 24.576 MHz crystal or oscillator. Refer to the primary codec vendor for crystal or oscillator requirement. AC_BITCLK is a 12.288 MHz clock driven by the primary codec to the COM-Express digital controller on board and to the other codec(s) on the carrier board.

2.2.4.3 HD Audio:

Figure 2-10 shows the connections for COM-Express HD Audio signals with three HD Codecs. The clock of is provided from the COM-Express module via AC_BITCLK signal. The AC_BITCLK is a 24 MHz clock driven by the COM-Express module as an output to the codec present on the carrier board.

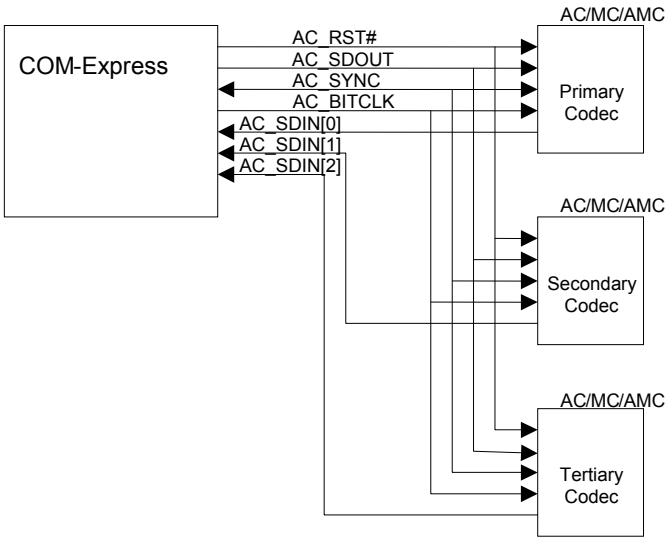


Figure 2-10 HD Audio Connections

Note: AC'97 Audio and HD Audio are mutually exclusive and cannot be used at the same time on a platform.

2.2.4.4 HDA Audio Jack Detection Function

The HDA audio jack connectors should be designed to detect and inform the operating system which jack is plugged or un-plugged. It can be done by the sensing resistors and the programmable GPIO pins. Figure 2-11 shows the example of jack detection function with sensing resistors. Please refer to the codec specification to get more information.

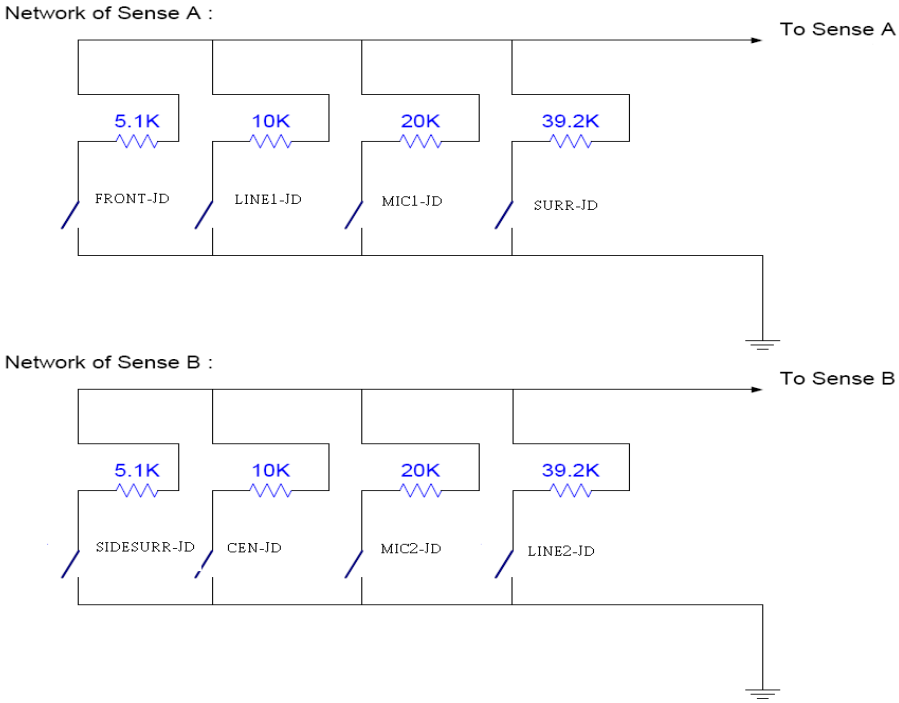


Figure 2-11 Jack Detection with sensing resistors

The schematic of the evaluation board is shown in Figure 2-12.

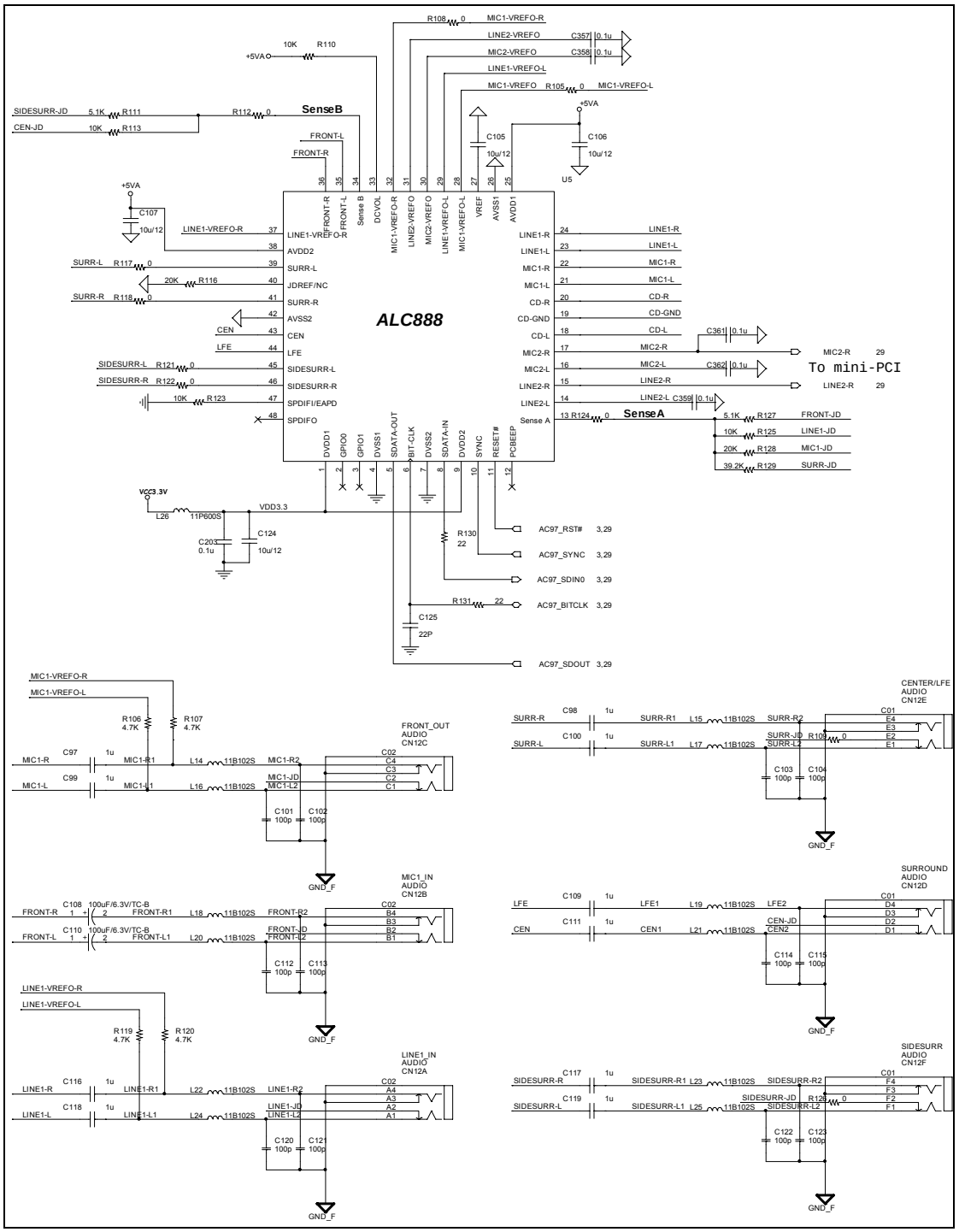


Figure 2-12 Reference Audio Schematic

2.2.5 Layout Guidelines

2.2.5.1 General Board Routing Recommendations

- The ground return paths for the analog signals should be considered.
- Digital signals routed in the vicinity of the analog audio signals must not cross the power plane split. Place the analog and digital signals as far as possible from each other.
- Partition the board with all analog components grouped together in one area and all digital components in another.
- Keep digital signal traces, especially the clock, as far away as possible from the analog inputs and voltage reference pins.
- All resistors in the signal path or on the voltage reference should be metal film. Carbon resistors can be used for DC voltages and the power supply path, where the voltage coefficient, temperature coefficient, and noise are not factors.
- Locate the crystal or oscillation closed to the codec.
- The AC'97 Audio / HD Audio trace impedance from codec to COM-Express Module should be 55 Ω ± 15%.

2.2.5.2 EMI Considerations

The signals entering or leaving the analog area must cross the ground split through the beads between digital ground and analog ground. No signal can cross the split/gap between the ground planes, which will cause a ground loop and greatly increase EMI emissions and degrade the analog and digital signal quality.

2.2.5.3 HD Audio Layout Guidelines

Figure 2-13 and Table 2-6 show the AC_SDIN layout topology.

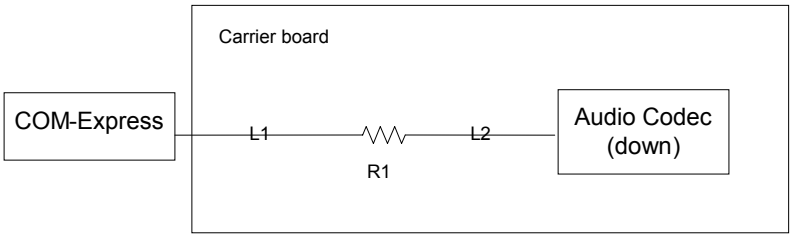


Figure 2-13 HD Audio – AC_SDIN Topology

Table 2-6 HD Audio – AC_SDIN Routing

Trace Impedance	HD Audio Requirements	Trace length	Series Termination Resistance	Signal Length Matching
55 Ω +/- 15%	5 on 7 (stripline) 5 on 7 (microstrip)	L1= 1" – 11" L2= 0.5"	R1= 33 Ω	N/A

Figure 2-14 and Table 2-7 show the layout topology #1 of AC_SDOUT, AC_SYNC,AC_BITCLK, and AC_RST# signals.

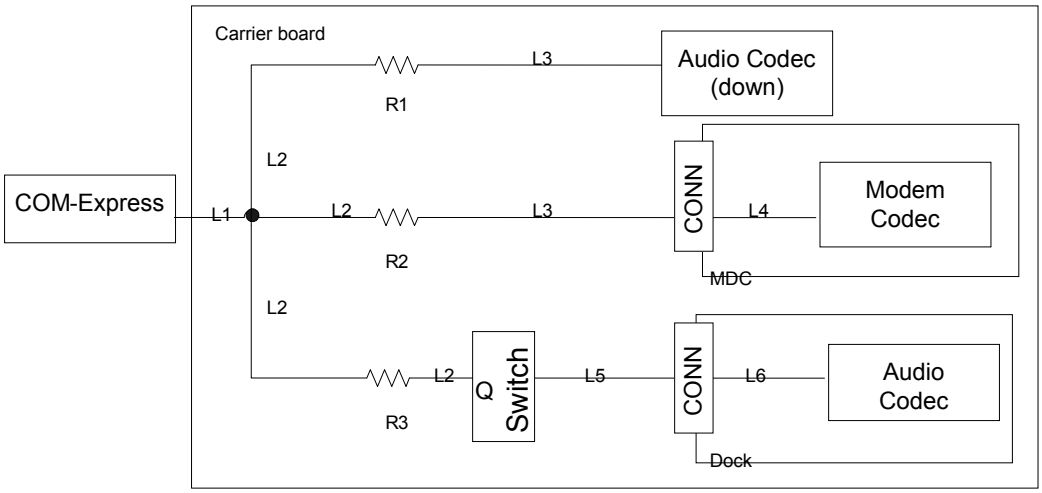


Figure 2-14 HD Audio – AC_SDOUT/AC_SYNC/AC_BITCLK/AC_RST# Topology #1

Table 2-7 HD Audio – AC_SDOUT/AC_SYNC/AC_ BITCLK/AC_RST# Topology #1				
Trace Impedance	**HD Audio Requirements**	**Trace length**	**Series Termination Resistance**	**Signal Length Matching**
55 Ω +/- 15%	5 on 7 (stripline) 5 on 7 (microstrip)	L1= 1“ – 11“ L2= 0.5“ L3= 1“ – 15“ L4= 1.5“ L5≤0.5“ L6= 5“	R1= 33 Ω+/- 5% R2= 39 Ω+/- 5% R3= 39 Ω+/- 5%	N/A

Figure 2-15 and Table 2-8 show the layout topology #2 of AC_SDOUT, AC_SYNC,AC_BITCLK, and AC_RST# signals.

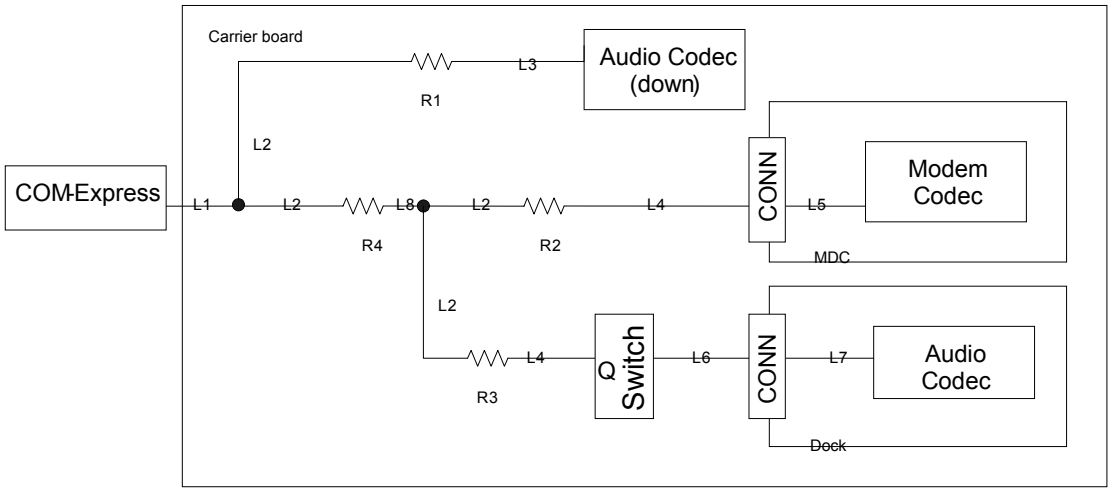


Figure 2-15 HD Audio – AC_SDOUT/AC_SYNC/AC_BITCLK/AC_RST# Topology #2

Table 2-8 HD Audio – AC_SDOUT/AC_SYNC/AC_ BITCLK/AC_RST# Topology #2

Trace Impedance	HD Audio Requirements	Trace length	Series Termination Resistance	Signal Length Matching
55 Ω +/- 15%	5 on 7 (stripline) 5 on 7 (microstrip)	L1= 0.5“ L2≤ 0.1“ L3= 1“ – 7“ L4= 1 - 5“ L5= 1.5“ L6≤ 0.5“ L7= 5“ L8= 0.1“ – 6“	R1= 39 Ω R2= 39 Ω R3= 39 Ω R4= 0 Ω	N/A

Generally the trace length on the COM-Express Module is around 5 inches. The total trace length on the carrier board should be less than 15 inches. Please refer to the Advantech layout checklist for details.

2.2.5.4 Grounding Techniques

Take care the grounding of the audio jacks, especially the line-in and microphone jacks. Avoid grounding the audio jacks to the ground plane directly under the connectors. Otherwise, the potential of audio noise voltage will be induced into the inputs due to the ground potential difference between the audio jacks ground and the codec’s ground. Figure 2-16 shows the grounding example for AC‘ 97.

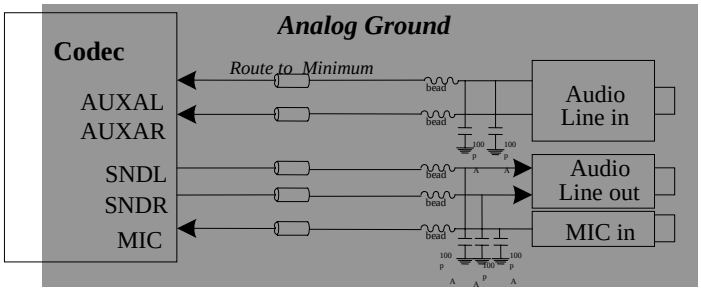


Figure 2-16 AC’97 Audio Ground Guidelines

2.2.5.5 AC’97 Audio Mic / Line-In / Aux-In Considerations

The back panel audio inputs (Mic, Line-in, Aux-in) should be independently routed. The ground return paths should be isolated from the carrier board ground plane. Use a capacitor to filter noise from the inputs bias net which may feed to all jacks. Route the input traces as far as possible from other traces.

2.3 Serial ATA

COM-Express Module provides up to four Serial ATA (SATA) interface, depending on the chipset specs of the module.

2.3.1 Schematic Guidelines

2.3.1.1 Serial ATA AC Coupling Requirements

Both the TX and RX SATA differential pairs require AC coupling capacitors. Figure 5-51 shows the connection for COM-Express SATA signals. All AC coupling capacitors on the transmitter (TX) and receiver (RX) are placed on the COM-Express Module. Do not place the AC coupling capacitors on the carrier board. Figure 2-17 and Figure 2-18 show the connections.

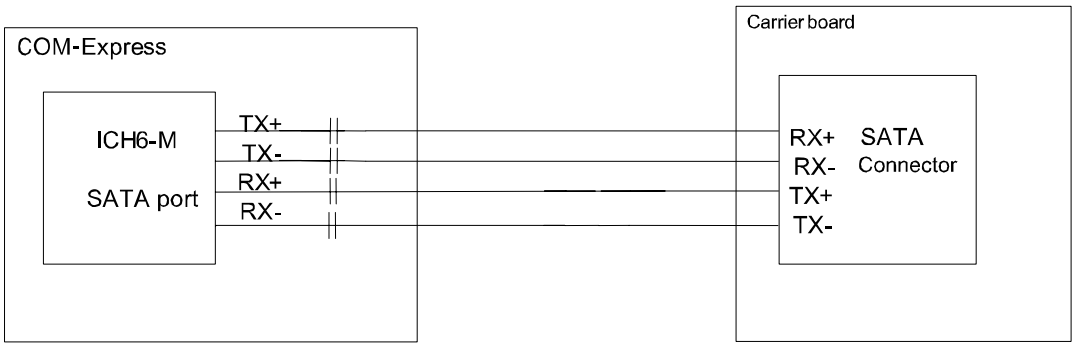


Figure 2-17 SATA interconnection example

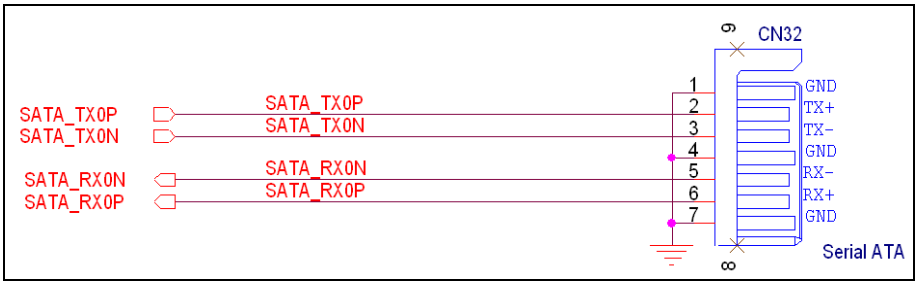


Figure 2-18 SATA Schematic Reference

2.3.1.2 Indicated LED Implementation

COM-Express Module provides a signal (ATA_ACT#) to indicate the activity of the SATA devices. It can be designed in conjunction with the LED signal of IDE PATA hard drives as DASP-S#0 signal. The example is shown in Figure 2-19. While the ATA_ACT# is active low, it indicates the SATA device is active, and then HD_LED# is changed to low.

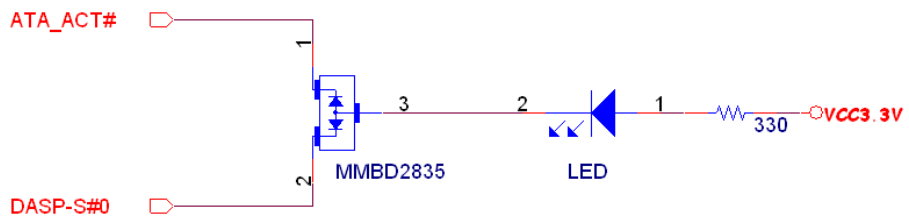


Figure 2-19 SATA LED Schematic Reference

2.3.2 Layout Guidelines

2.3.2.1 General routing and placement

- Place the SATA connectors as closed as possible to the COM-Express Module. The routing length is recommended to be not more than 3 inches. The Intra-pair trace length distance matching should be less than 5 mils.
- SATA signals must be ground referenced.
- Route all traces over continuous GND planes, with no interruptions. Avoid crossing over anti-etched areas if at all possible. Any discontinuity or split in the ground plane can cause signal reflections and should be avoid.
- Minimize layer changes. Use as few vias per SATA trace as possible (via count should include through hole connectors as an effective via). If a layer change is necessary, ensure that trace matching for either the TX or RX pair occurs within the same layer.
- Do not route SATA traces under crystals, oscillators, clock synthesizers, magnetic devices or ICs that use and/or duplicate clocks.
- Avoid stubs whenever possible. Utilize vias and connector pads as test points instead.
- The SATA differential trace impendence target is 100 Ω ± 20%. Use an impedance calculator to determine the trace width and spacing required for the specific board stack-up being used, keeping in mind that the target is a 100 Ω ± 20%.

2.3.2.2 Serial ATA Trace length

- The length of the SATA differential pairs should be designed as short as possible. For direct-connected topology where the SATA differential signal pair is routed directly to a mobile SATA connector, It's recommended the trace length of SATA signals should be within 3 inches for better signal integrity.
- The SATA differential pair traces should be length matched. The difference between two line traces of TX / RX differential pairs should be restricted to less than 20 mils, and less trace mismatch is recommended.

Figure 2-20 shows an example of SATA trace length pair matching. L_A must equal to L_A', L_B must equal to L_B', …and so on. It's recommended to avoid the vias for layer change, ensuring that the differential pairs are equal if necessary.

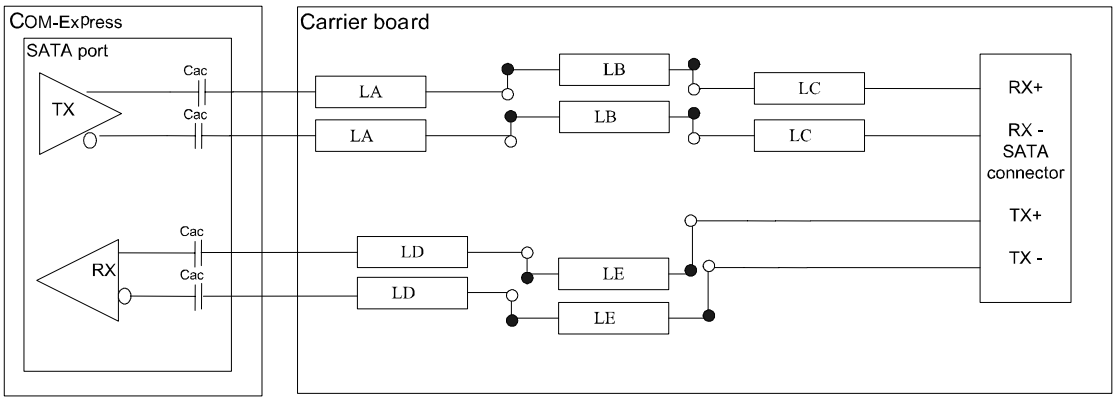


Figure 2-20 SATA traces length matching

2.4 Primary IDE

2.4.1 Signal Description

Table 2-9 shows the COM-Express IDE signals.

Table 2-9 IDE signals description			
Pin	**Signal**	**I/O**	**Description**
-	IDE_D[0..15]	I/O	Bidirectional data to/from IDE device
D13,14,15	IDE_A[0..2]	O	Address lines to IDE device
D16	IDE_CS1#	O	IDE Device Chip Select for 1F0h to 1FF0h range
D17	IDE_CS3#	O	IDE Device Chip Select for 3F0h to 3FF0h range
D8	IDE_REQ	I	IDE DMA Request for IDE Master. This is the input pin from the IDE DMA request to do the IDE Master Transfer. It will active high in DMA or Ultra-33 mode and always be inactive low in PIO mode.
D10	IDE_ACK#	O	IDE device DMA Acknowledge
C13	IDE_IORDY	I	IDE device I/O ready input Pull low by the IDE device, active low
C14	IDE_IOR#	O	I/O ready line to IDE device
D9	IDE_IOW#	O	I/O write line to IDE device Data latched on trailing (rising) edge
D12	IDE_IRQ	I	Interrupt request from IDE device
D18	IDE_RESET#	O	Low active hardware reset (RSTISA inverted).
D77	IDE_CBLID#	O	Input from off-module hardware indicating the type of IDE cable being used. High indicates a 40-pin cable used for legacy IDE modes. Low indicates that an 80 pin cable with interleaved grounds is used. Such a cable is required for Ultra-DMA 66, 100 and 133 modes.

2.4.2 DC Specifications

Table 2-10 Ultra DMA modes 1-4 (5V)					
Symbol	**Parameter**	**Min**	**Max**	**Unit**	**Note**
V_{IH}	Input High Voltage	-	5.5	V	
V_{OL}	Output Low Voltage	-	0	V	
V_{OH}	Output High Voltage	2	-	V	

2.4.3 IDE Spec

Please refer to “Information Technology - AT Attachment with Packet Interface – 7 Volume 2 (ATA/ATAPI-7 V2)” Annex B.5 for the details

2.4.4 Schematic Guidelines

2.4.4.1 Design Considerations

The IDE port can support two hard drives or other ATAPI devices. The two devices on the port are wired in parallel, which is accomplished by plugging both drives into a single flat ribbon cable equipped with two socket connectors. A jumper can be manually set on each IDE device for selecting master or slave mode.

If two devices are used in the master/slave mode, the IDE_CBLID# of both devices must be connected together as in Fig 2-21. These pairs of pins negotiate between the master and slave devices. The devices may not function correctly unless these pins are interconnected. If two devices are plugged into one standard IDE cable, the cable will interconnect the pins properly by itself.

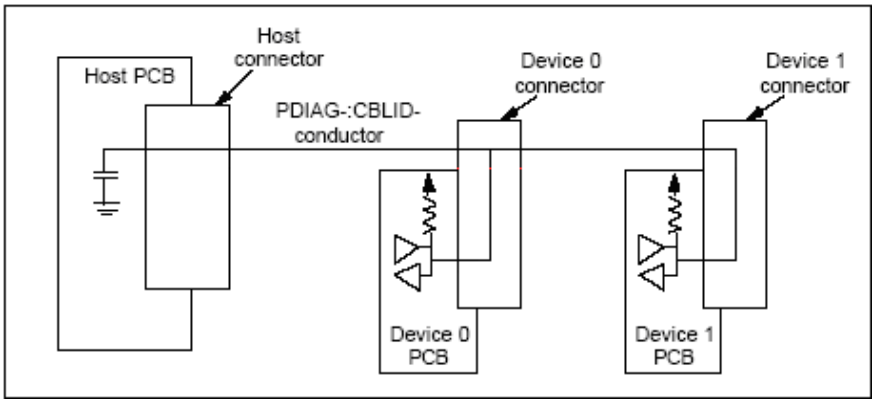


Figure 2-21 IDE Master/Slave Handshake Signals Connection

The DASP-S# (pin 39 on IDE connector or pin 45 on CF connector) should be also connected between master and slave devices. The reference schematic is as Figure 2-22.

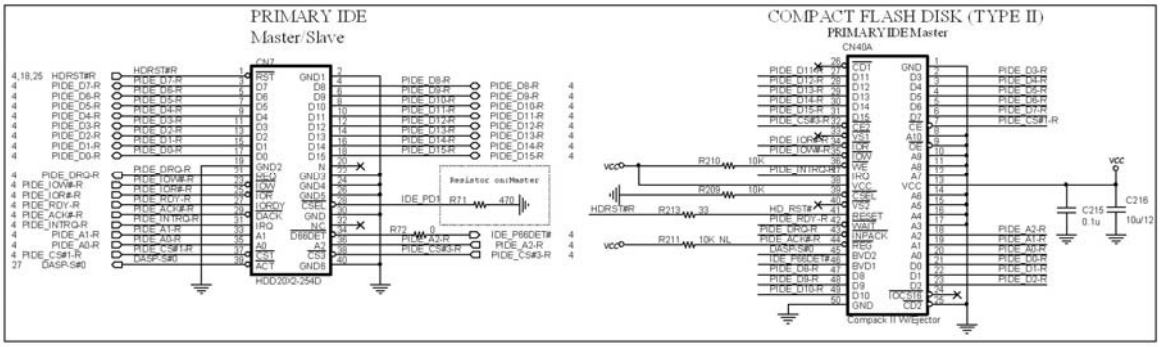


Figure 2-22 IDE Master/Slave DASP-S# Signals Connection

2.4.4.2 UDMA Support

COM-Express Modules support UDMA ATA 33 / 66 / 100 data transfer modes. If an advanced IDE data transfer mode such as UDMA 66 / 100 is required, the 80-pin type IDE connector and cable are needed for signal integrity.

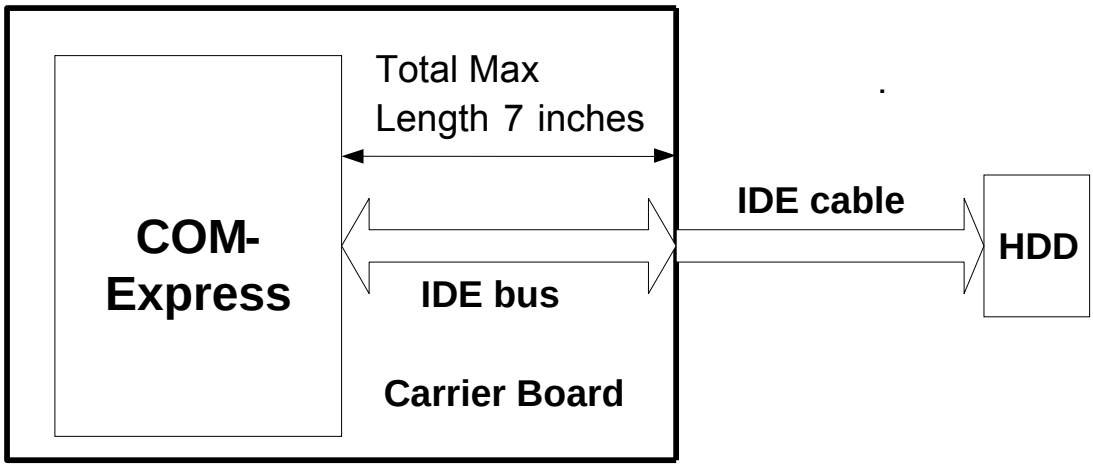


Figure 2-23 IDE Bus Trace Length on Carrier Board

2.4.4.3 IDE interface connections

All necessary pulled-up/down resistors are implemented on the COM-Express Module. Do not implement these resistors on the carrier board.

If there is no IDE device, all IDE pins should be left as NC (Not Connected).

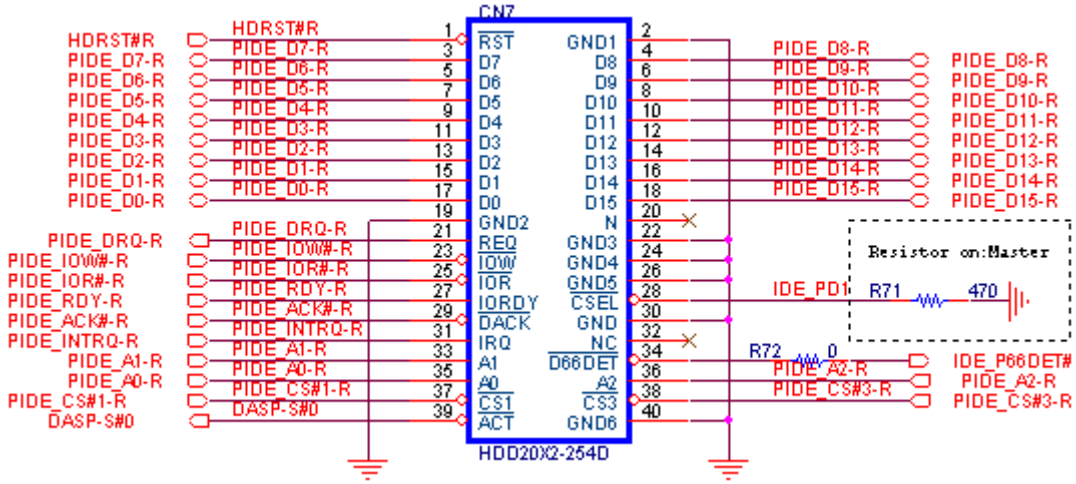


Figure 2-24 IDE Connections

2.4.4.4 Compact Flash Socket Implementation Application Notes

For the IDE application, the Compact Flash (CF) card cannot be hot-plugged. If hot-plug support is necessary, the PCI-based Card-Bus controller chip can be integrated

onto the carrier board and used to control the CF hot-plug function. The Figure 2-25 shows the CF schematics.

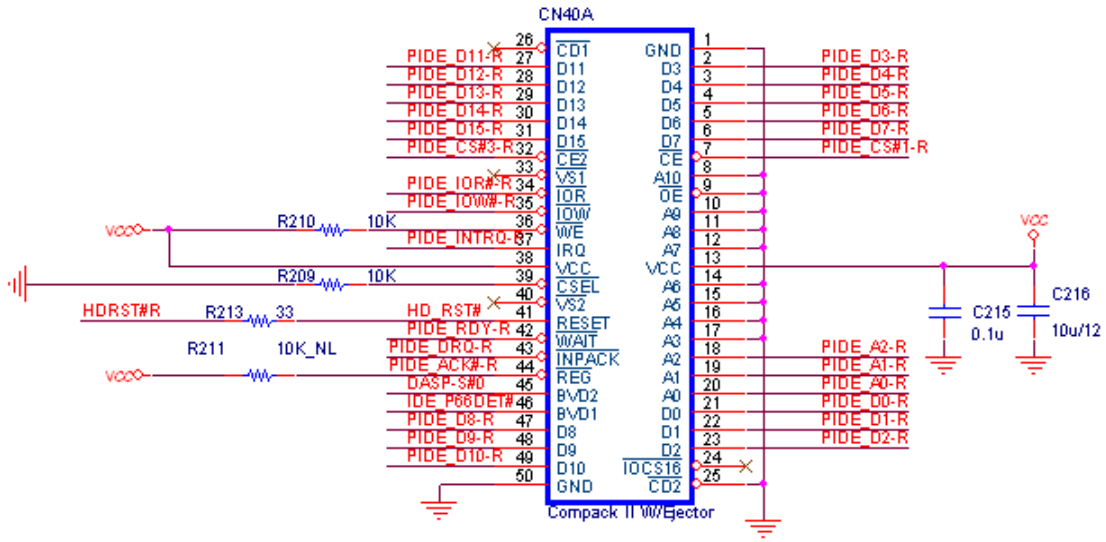


Figure 2-25 CF Connections

The CF card can be configured as a slave device when the CSEL signal is set as non-connection. If two CF cards (or one CF card and one hard drive) are used in the master/slave mode on the same IDE port, the IDE_CBLID# and DASP-S#0 pins on both devices must be connected. The signal negotiates the communication between the master and slave devices.

2.4.5 Layout Guidelines

2.4.5.1 IDE data and strobe routing guideline

The IDE interface can be routed with 6-mil traces on 6-mil spaces (dependent upon stack-up parameters), and must be less than 7 inches in length (from COM-Express connector to carrier board IDE connector). The maximum length difference between the data signals and the strobe signal (IDE_IOR# and IDE_IOW#) should be less than 100 mils. Refer to Advantech layout checklist for the detail of each platform. Use Daisy chain not Y type routing if two connectors as both IDE and CF are needed.

Table 2-11 IDE Routing Summary

Trace Impedance	IDE Routing requirements	Trace length	IDE Signal length matching
55 Ω ± 15%	5 on 5 (Based on stack-up in chap 4)	1 ~ 7 inches	The two strobe signals must be matched within 100 mils of each other. The data lines must be within ± 450 mils of the average length of the two strobe signals

2.5 PCI Express Bus

COM-Express provides a PCI Express Bus interface that is compliant with the PCI Express Base Specification, Revision 1.0a. It supports several general purpose PCI Express port (x1) and external graphics using PCI Express architecture (x16) as PEG interface.

2.5.1 Signal Description

Table 2-12 shows COM-Express PCI Express bus signals for general purpose.
Table 2-13 shows PCI Express bus signal for external graphics.
Table 2-14 shows ExpressCard signals.

Table 2-12 PCIE Signal Description(General purpose)

Pin	Signal	I/O	Description
A68,64,61,58,55,52 A69,65,62,59,56,53	PCIE_TX[0:5]+ PCIE_TX[0:5]-	O	PCI Express Differential Transmit Pairs 0 through 5
B68,64,61,58,55,52 B69,65,62,59,56,53	PCIE_RX[0:5]+ PCIE_RX[0:5]-	I	PCI Express Differential Receive Pairs 0 through 5
-	PCIE_TX[16:31]+ PCIE_TX[16:31]-	O	PCI Express Differential Transmit Pairs 16 through 31 These are same line as PEG_TX[0:15]+ and -
-	PCIE_RX[16:31]+ PCIE_RX[16:31]-	I	PCI Express Differential Receive Pairs 16 through 31 These are same line as PEG_TX[0:15]+ and -
A88 A89	PCIE_CLK_REF+ PCIE_CLK_REF-	O	Reference clock output for all PCI Express Graphics lanes.
B66	WAKE0#	I	PCI Express wakeup signal.

Table 2-13 PEG Signal Description(x16 Graphics)

Pin	Signal	I/O	Description
-	PEG_TX[0:15]+ PEG_TX[0:15]-	O	PCI Express Graphics Transmit Differential Pairs 0 through 15 Some of these are multiplexed with SDVO lines.
-	PEG_RX[0:15]+ PEG_RX[0:15]-	I	PCI Express Graphics Receive Differential Pairs 0 through 15 Some of these are multiplexed with SDVO lines.
D54	PEG_LANE_RV#	I	PCI Express Graphics lane reversal input strap. Pull low on the carrier board to reverse lane order. Be aware that the SDVO lines that share this interface do not necessarily reverse order if this s strap is low.
D97	PEG_ENABLE#-	I	Strap to enable PCI Express x16 external graphics interface. Pull low to disable internal graphics and enable the x16 interface.

Table 2-14 Express Card Support

Pin	Signal	I/O	Description
A49, B48	EXCD[0:1]_CPPE#	I	PCI ExpressCard: PCI Express capable card request, active low, one per card
A48, B47	EXCD[0:1]_RST#	O	PCI ExpressCard: reset, active low, one per card

2.5.2 Schematic Guidelines

2.5.2.1 PCI Express AC Coupling Capacitor

Each PCI Express lane is AC coupled between its corresponding transmitter (TX) and receiver (RX). Figure 2-26 and Figure 2-27 shows the connection for COM-Express signals and PCI Express connector. The AC coupling capacitors of TX+/- is present on COM-Express Module. The AC coupling capacitors of RX+/- should be placed on the carrier board and closely to the transmitter pins of the PCI Express devices.

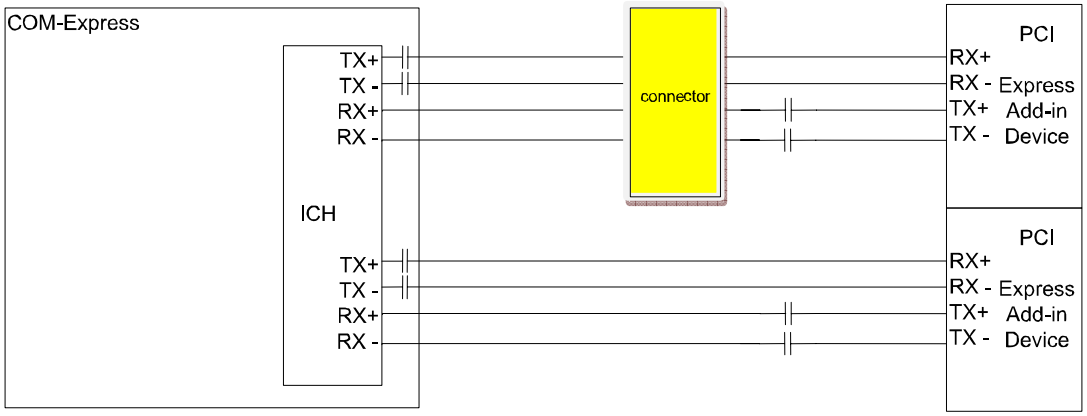


Figure 2-26 PCI Express Interconnect Example

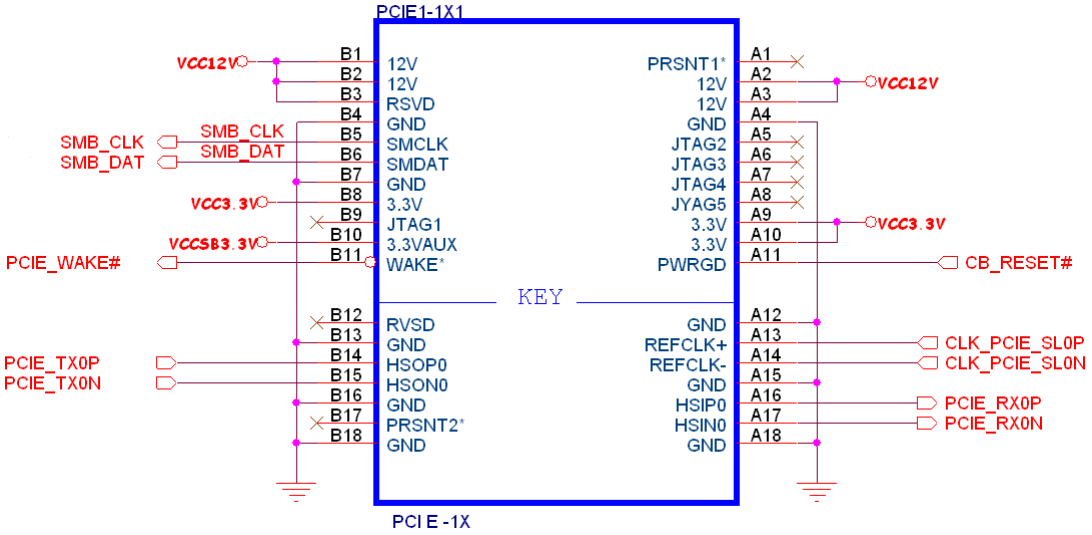


Figure 2-27 PCI Express Connector Schematic Reference

Use the exact same package size for the capacitor on each signal in a differential pair. Table 2-15 shows the PCI Express capacitor reference.

Table 2-15 PCI Express Capacitor Reference

Type	Value	Tolerance	Placement	Length Matching Between Differential Pair
AC Capacitor	75 nF-200 nF	20%	Recommended to place close to the transmit side	As close as possible between the differential pairs

2.5.2.2 Bowtie Topology Considerations: Untangling Nets

It is possible that when interconnecting PCI Express devices, the “bowtie” or signal-crossing scenarios might occur when the link is routed on the PCB. There are three main types of these scenarios:

(1) TX+, TX- crisscrossing within a pair.
(2) Crossing of transmitter and receiver pairs within a lane.
(3) Crossing of lanes within a link.

The PCI Express specification provides two different features: Polarity Inversion and Lane Reversal. Both features help to overcome the layout difficulties encountered in scenarios #1 and #3. The specification does not include any provisions to address scenario #2

- **Polarity Inversion**

The PCI Express spec requires polarity inversion to be supported independently by all receivers across the link, i.e. the positive signal from the transmitter (TX+) can connect to the negative signal of the receiver (RX-) in the same lane. Of course that means the negative signal from the transmitter (TX-) must now also connect to the positive signal of the receiver (RX+) in such a scenario. Figure 2-28 shows an example. It is important that polarity inversion does not mean the direction inversion, i.e. the TX differential pair from one device must still connect to the RX differential pair on the receiver device.

Figure 2-28 Polarity Inversion on a TX to RX Interconnect

- **Lane Reversal**

Lane reversal allows the lane number to be switched from high to low and low to high. For example the lane 15 from the COM-Express would connect to lane 0 of the device, and lane 0 from the COM-Express would connect to lane 15 of the device. The Lane reversal feature needs to be supported by one of the two devices. The feature does not imply direction reversal, i.e. the TX differential pair from an upstream device must still connect to the RX differential pair on the downstream device. Figure 2-29 shows an example for both Lane Reversal and Polarity Inversion.

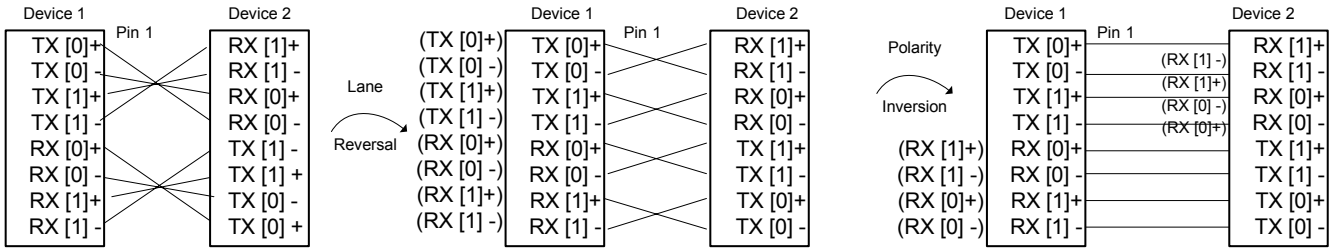


Figure 2-29 Lane Reversal and Polarity Inversion - TX to RX Interconnect

2.5.2.3 Terminating Unused PCI Express Ports

If PCI Express ports are not implemented on the carrier board design, the PCIE _TX+/-[n] and PCIE_RX+/-[n] signals should be left as “Not Connected”.

2.5.3 Layout Guidelines

This section shows the summary of the layout routing guidelines.

2.5.3.1 Differential pairs

The PCI Express signals should be routed as differential pairs. The following is a summary of general routing guidelines for the differential pair traces.

In COM-Express platforms the PCI Express differential trace impendence target is 100 Ω ± 20%.

It is important to equalize the total length of the traces in the pair throughout the trace; each segment of trace length should be equal along the entire length of the pair. Figure 2-30 shows an example. L_A must equal to L_A', L_B must equal to L_B' …, and so on.

It is preferable to route TX and RX differential pairs alternately on the same layer (TX pair next to RX pair rather than another TX pair).

Tight coupling within the differential pair and increased spacing to other differential pairs helps to minimize EMI and crosstalk.

It is important to maintain routing symmetry between the two signals of a differential pair.

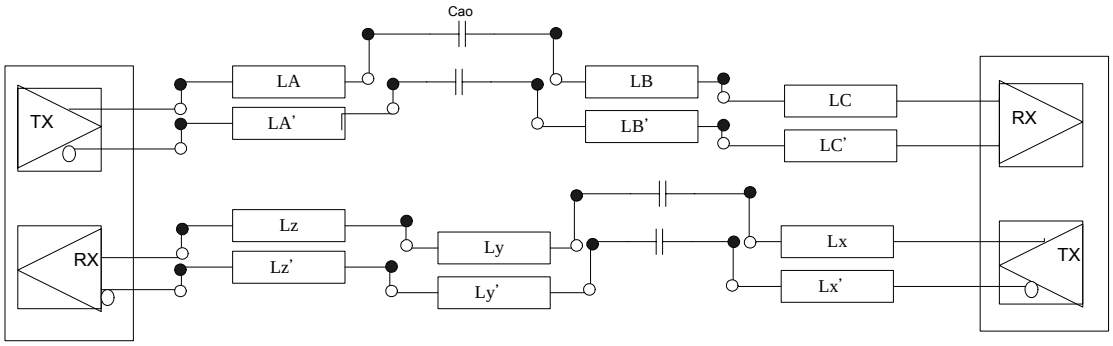


Figure 2-30 Trace Length Matching in Each Segment

2.5.3.2 Board Stack-up Considerations

Table 2-16 shows the PCI Express trace width and spacing for Micro-strip and Strip-line based on the six layer board stack-up. Keep the required trace impedance for better signal integrity.

Table 2-16 PCI Express Trace Width and Spacing for Micro-strip and Strip-line

	Trace Width	Differential Pair Trace Spacing	Adjacent Pair / Trace Spacing	Differential Pair Length Matching	Breakout Guideline	Nominal Trace Impedance (Zo)
Micro-strip	5 mils	10 mils	20 mils	5 mils	5 mil trace width, 10mil separation to both the differential pair signals and adjacent traces for up to 250 mils	100 Ω±20% (Differential)
Strip-line	5 mils	10 mils	20 mils	5 mils	Only 5 mil trace width on 10 mils spacing is allowed	100 Ω±20% (Differential)

2.5.3.3 PCI Express Topology #1 – Device Down Routing Guidelines

The device down topology as Figure 2-31 allows a maximum traces length of 15 inches from COM-Express Module to the down device. The max length takes into account all routing, including the breakout region, which should not exceed 0.25 inches per device. The routing rules are shown in Table 2-17.

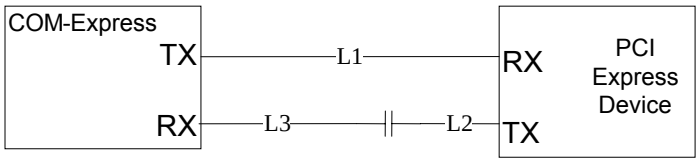


Figure 2-31 Topology #1 – COM Express to PCI Express Device Down

Table 2-17 COM Express to PCI Express D

L1	L2	L3	Capacitor Value
Max = 15 inches	Min = 0.25 inches Max = 14.75 inches	Max = 15 inches – L2	75 nF to 200 nF, Tolerance = 20%,

The TX and RX pairs can be routed “interleaved”, such that the pairs alternate between TX and RX on the carrier board, or “non-interleaved”, where the TX and RX pairs are routed next to each other. Only interleaved routing can be used for micro-strip routing topologies. For strip-line routing, it is recommended to route the TX and RX differential pairs as interleaved to reduce the crosstalk. Figure 2-32 shows the example.

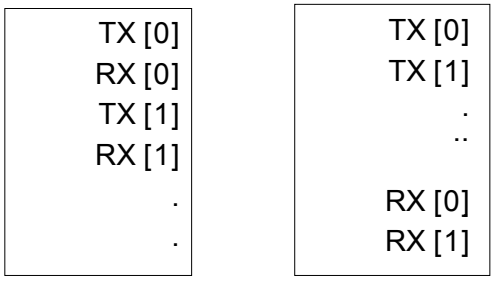


Figure 2-32 “interleaved” (left) and “non-interleaved” pairs (right)

2.5.3.4 PCI Express Topology #2 and #3 – Device Up Routing

Both the ExpressCard and the docking topologies allow a maximum of 9 inches from COM-Express connector to the device up docking connector as Figure 2-33. The maximum length takes into account all routing, including the breakout region. The Table 2-18 shows the traces length limitation and capacitors value. The TX and RX pairs must be routed interleaved to reduce the crosstalk effect on the micro-strip and strip-line traces.

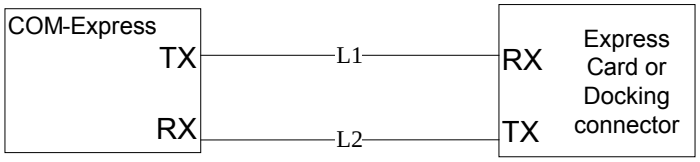


Figure 2-33 Topology #2 and #3 – COM Express to Express Card or Docking Connector

Table 2-18 COM Express to Express Card		
L1	**L2**	**Capacitor Value**
Max = 9 inches	Max = 9 inches	75 nF to 200 nF, Tolerance = 20%,

2.6 SDVO

The SDVO function shares the pins of the PEGX16 interface. SDVO ports should be working with proper intel Video BIOS for setting SDVOB and SDVOC configurations.

For CH7308 SDVO to LVDS application, the reference schematic of Advantech carrier board is as in Figure 2-34.

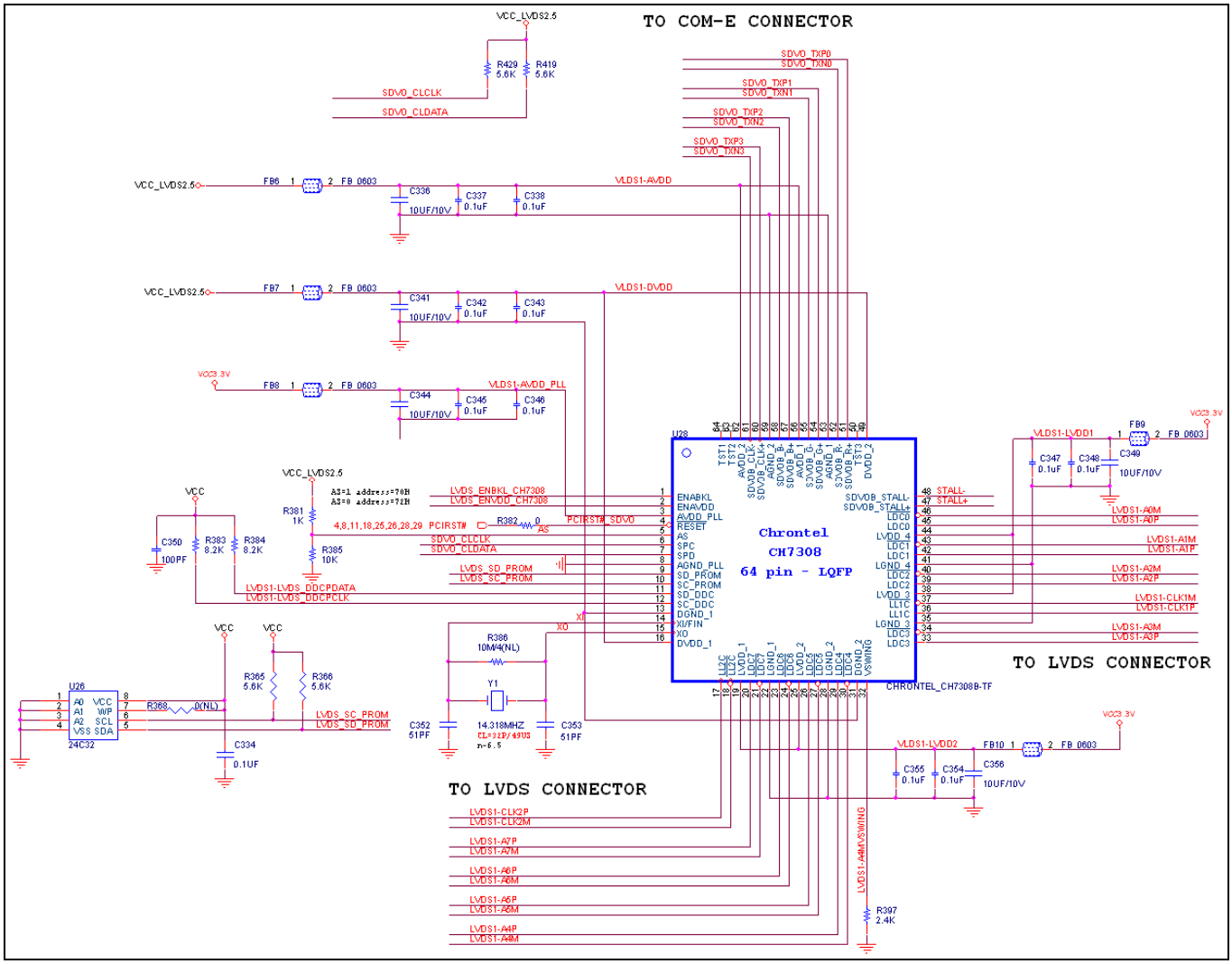


Figure 2-34 CH7308 Schematic

The pulled-up resistors of SDVO_DAT and SDVO_CLK should be populated on the carrier board to let COM-Express Module enable the SDVO. Contact Advantech for advanced technical support.

Polarity Inversion and Lane Reversal are NOT supported on SDVO signals which are sharing the PCI Express Graphics X16 interface pins.

For the application of all SDVO devices, please refer to the schematic and layout guidelines from the SDVO device vendor and request Advantech technical support.

2.7 PCI-Bus

COM-Express provides a PCI Bus interface that is compliant with the PCI Local Bus Specification, Revision 2.3. The implementation is optimized for high-performance data streaming when COM-Express is acting as either the target or the initiator on the PCI bus. For more details of the PCI Bus interface, please refer to the PCI spec.

2.7.1 Signal Description

Table 2-19 shows the COM-Express PCI bus signals.

Table 2-19 PCI Signal Description

Pin	Signal	I/O	Description
D50	PCI_CLK	O	PCI 33 MHz clock output
D48	PCI_CLKRUN#	I/O	Bidirectional pin used to support PCI clock run protocol for mobile systems
C22,C19, C17,D20	PCI_REQ[0..3]	I	Bus Request signals for up to 4 external bus mastering PCI devices. When asserted, a PCI device is requesting PCI bus ownership from the arbiter.
C20,C18, C16,D19	PCI_GNT[0..3]	O	Grant signals to PCI Masters. When asserted by the arbiter, the PCI master has been granted ownership of the PCI bus.
-	PCI_AD[0..31]	I/O	PCI Address and Data Bus Lines. These lines carry the address and data information for PCI transactions.
D26,C33, C38,C44	PCI_C/BE[0..3]	I/O	PCI Bus Command and Byte Enables. Bus command and byte enables are multiplexed in these lines for address and data phases, respectively.
D32	PCI_PAR	I/O	Parity bit for the PCI bus.
D33	PCI_SERR#	I/O OD	System Error. Asserted for hardware error conditions such as parity errors detected in DRAM.
C34	PCI_PERR#	I/O	Parity Error. For PCI operation per exception granted by PCI 2.1 Specification.
C15	PME#	I	Power management event.
C35	PCI_LOCK#	I/O	Lock Resource Signal. This pin indicates that either the PCI master or the bridge intends to run exclusive transfers.
C36	PCI_DEVSEL#	I/O	Device Select, active low. When the target device has decoded the address as its own cycle, it will assert DEVSEL#.
D35	PCI_TRDY#	I/O	Target Ready. This pin indicates that the target is ready to complete the current data phase of a transaction.
C37	PCI_IRDY#	I/O	Initiator Ready. This signal indicates that the initiator is ready to complete the current data phase of a transaction.
D34	PCI_STOP#	I/O	Stop. This signal indicates that the target is requesting that the master stop the current transaction.
D36	PCI_FRAME#	I/O	Cycle Frame of PCI Buses. This indicates the beginning and duration of a PCI access.
C23	PCI_RESET#	I	PCI Bus Reset. This is an output signal to reset the entire PCI Bus. This signal is asserted during system reset.
C49,C50, D46,D47	PCI_IRQ[A…D]	I	PCI interrupt request lines.
D49	PCI_M66EN	I	Module input signal indicates whether an off-module PCI device is capable of 66 MHz operation. Pulled to GND by Carrier Board device or by Slot Card if the devices are NOT capable of 66 MHz operation. If the module is not capable of supporting 66 MHz PCI operation, this input may be a no-connect on the module. If the module is capable of supporting 66 MHz PCI operation, and if this input is held low by the Carrier Board, the module PCI interface shall operate at 33 MHz.

2.7.2 DC Specifications

Table 2-20 DC specifications for 5V signaling of PCI Bus

Symbol	Parameter	Min	Max	Units	Note
Vcc	Supply Voltage	4.75	5.25	V	
Vih	Input High Voltage	2.0	Vcc+0.5	V	
Vil	Input Low Voltage	-0.5	0.8	V	
Voh	Output High Voltage	2.4	-	V	
Vol	Output Low Voltage	-	0.55	V	*1

Table 2-21 DC specifications for 3.3V signaling of PCI Bus

Symbol	Parameter	Min	Max	Units	Note
Vcc	Supply Voltage	3.0	3.6	V	
Vih	Input High Voltage	0.5Vcc	Vcc+0.5	V	
Vil	Input Low Voltage	-0.5	0.3Vcc	V	
Vipu	Input Pull-up Voltage	0.7Vcc	-	V	*1
Voh	Output High Voltage	0.9Vcc	-	V	
Vol	Output Low Voltage	-	0.1Vcc	V	

*1. This specification should be guaranteed by design. It is the minimum voltage to which pulled-up resistors are calculated to pull a floated network. Applications sensitive to static power utilization must assure that the input buffer is conducting minimum current at this input voltage.

2.7.3 AC Specifications

Refer to “PCI Local Bus Specification Revision 2.3” chapter 4.2 for details.

2.7.4 Schematic Guidelines

2.7.4.1 Differences among PCI Slots

Most PCI signals are connected in parallel to all the slots (or devices). The exceptions are the following pins from each slot or device as Table 2-22:

Table 2-22 Carrier PCI Slots	
IDSEL	: Connected (through resistor) to a different AD line for each slot.
CLK	: Connected to a different COM-Express PCI clock signal for each slot.
INTA#, INTB#, INTC#, INTD#	: Connected to a different COM-Express interrupt signal for each slot.
REQ#	: Connected to a different COM-Express request signal for each slot, if used.
GNT#	: Connected to a different COM-Express grant signal for each slot, if used.

Each signal connects differently for each of the four possible slots or devices as summarized in the following PCI Slots/Devices Table 2-23.

Table 2-23 Carrier PCI Slots/Devices Interrupt Routing Table

COM-EXPRESS	PCI Slot 0	PCI Slot 1	PCI Slot 2	PCI Slot 3
AD20 (Pin D39)	IDSEL	-	-	-
AD21 (Pin C42)	-	IDSEL	-	-
AD22 (Pin D40)	-	-	IDSEL	-
AD23 (Pin C43)	-	-	-	IDSEL
INTA# (Pin C49)	INTA#	INTB#	INTC#	INTD#
INTB# (Pin C50)	INTB#	INTC#	INTD#	INTA#
INTC# (Pin D46)	INTC#	INTD#	INTA#	INTB#
INTD# (Pin D47)	INTD#	INTA#	INTB#	INTC#

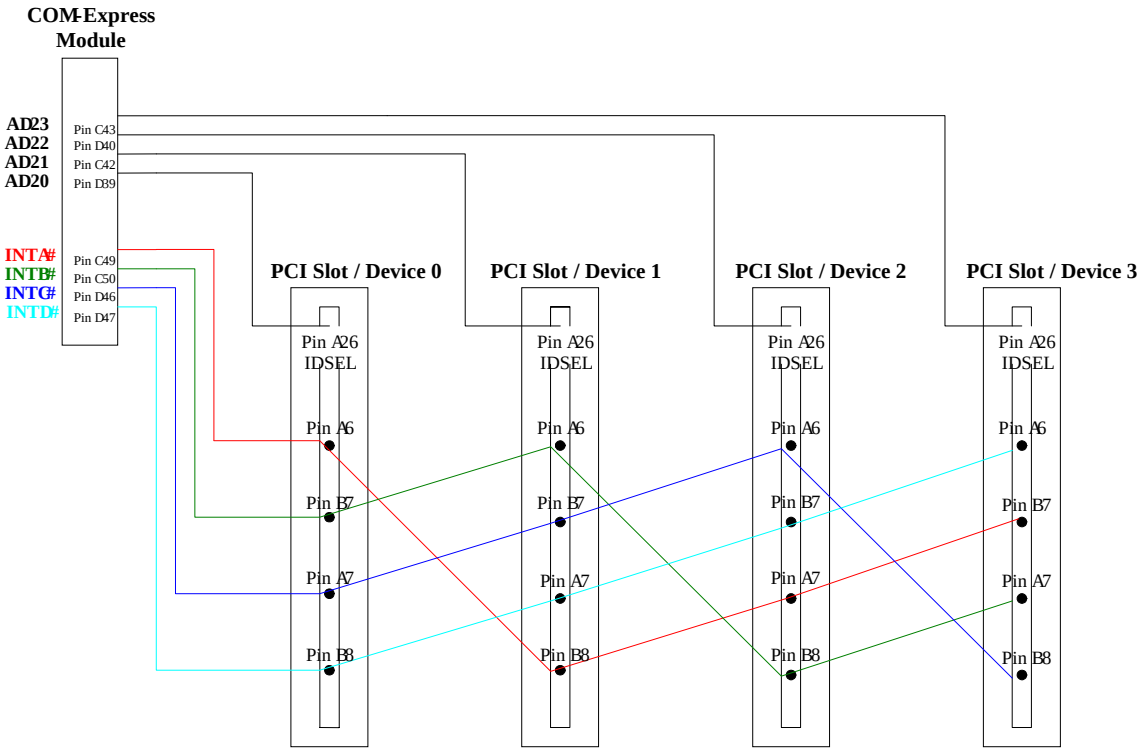


Figure 2-35 Interrupt Routing of PCI Slots / Devices on carrier board

Due to different system configurations, IRQ line routing to the PCI slots should be made to minimize the sharing of interrupts between both internal chipset functions and PCI device functions. In this case, the INTA# pin of the device should not necessarily be connected to the COM-Express Connector INTA# signal. Please refer to Figure 2-35. The Pin A6 is connected to INTA# on Device 0 but INTD# on Device 1. The Pin B7 is connected to INTB# on Slot 0 but INTC# on Slot 1. The schematic reference of INTx# and IDSEL example are shown in Figure 2-36.

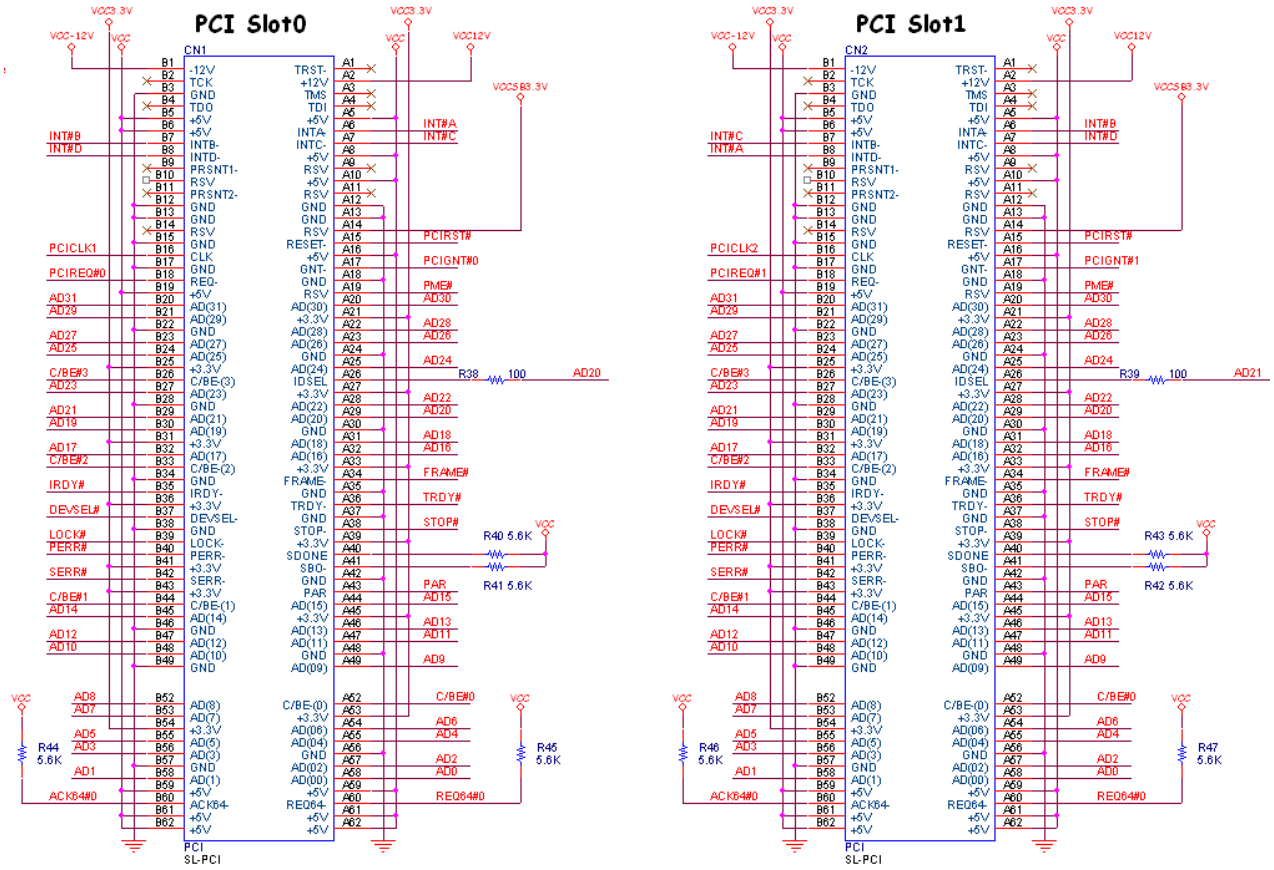


Figure 2-36 Interrupt Routing and IDSEL Schematic Reference

2.7.4.2 PCI Clock and Clock Skew

The trace length for all PCI clocks should be matched and controlled. PCI clock routes should be separated as far from other signal traces as possible. PCI clock signals should be routed as impedance controlled traces, with trace impedance of 55 Ω. Only one PCI device or slot could be driven directly from the COM-Express PCI clock output pin, or the clock buffer should be implemented.

The maximum allowable clock skew is 2 ns, as the Tskew shown in the Table 2-24 and Figure 2-37. The specification applies not only at a single threshold point, but at all points on the clock edge that fall in the switching range. The maximum skew is measured between any two components rather than between connectors. To correctly evaluate clock skew, the system designer must take into account clock distribution on the add-in card.

Table 2-24 Clock Skew Parameters

Symbol	3.3 V Signaling	5 V Signaling	Units
Vtest	0.4 Vcc	1.5	V
Tskew	2 (max)	2 (max)	ns

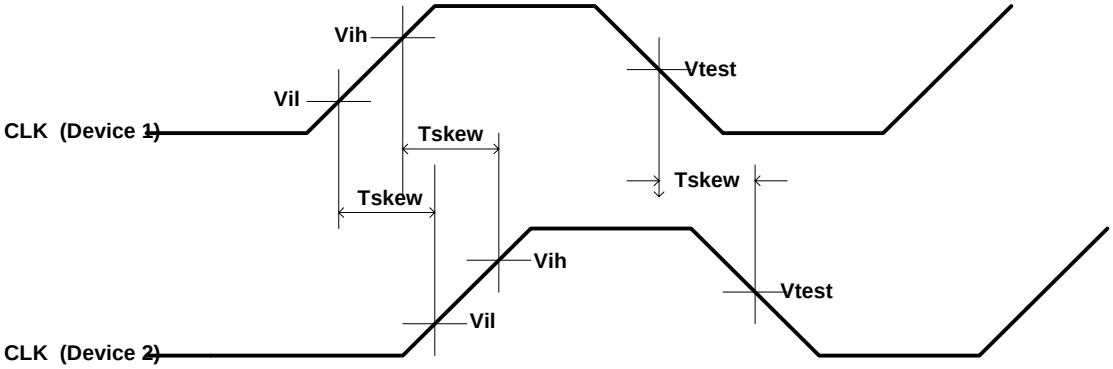


Figure 2-37 Clock Skew of PCI

2.7.4.3 Clock Buffer and PCI Arbiter

REQ/GNT signals are used by bus-mastering PCI devices. Most COM-Express Modules do not have enough REQ/GNT pairs and pins available to support the bus-mastering device on each slot. The PCI arbiter should be implemented while the extra REQ/GNT pairs are required. Figure 2-38 shows the design example for PCI arbiter.

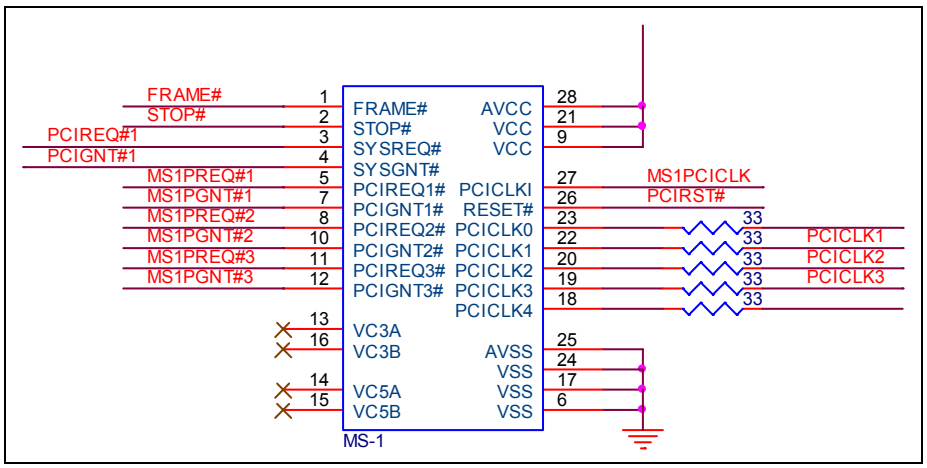


Figure 2-38 Design Example: PCI Arbiter

If there are less than four REQ/GNT pairs available for external devices, It should be assigned starting at the first pair as REQ0# / GNT0#. Therefore, external bus-mastering devices should be placed in the lowest number and the non-bus mastering devices should be placed in the highest-number of REQx# / GNTx#.

2.7.4.4 Non-necessary Signals for Individual PCI device

A PCI device implemented directly on the carrier board uses a subset of the signals shown on the slot connector. Some pins on the slot connector are used for slot and PCI card management functions and are not necessary for the operation of the PCI device.

An individual PCI device will not have pins REQ64, ACK64, M66EN, PRSNT1, PRSNT2, SDONE, SBO#, or the reserved pins. Most devices do not implement the test pins TCK, TDO, TDI, TMS, and TRST. Most PCI devices use INTA# only and do not have a connection for INTB#, INTC# or INTD#.

2.7.4.5 Carrier Board PCI slot Power Requirements

All PCI connectors require four power rails: +5 V, +3.3 V, +12 V, and -12 V. Systems that provide PCI connectors are required to provide all four rails in every system with the current budget. Systems may optionally supply 3.3 Vaux power. Systems that do not support PCI bus power management must treat the 3.3 Vaux pin as reserved.

There are no specific system requirements for current per connector on the 3.3 V and 5 V rails; this is system dependent. Note that an add-in card must limit its total power consumption to 25 watts (from all power rails). The system provides a total power budget for add-in cards that can be distributed between connectors in an arbitrary way, as Table 2-25. The PRSNTn# pins on the connector allow the system to optionally assess the power demand of each add-in card and determine if the installed configuration will run within the total power budget.

Table 2-25 PCI Add-in Card Maximum Loading Current via Each Power Rail

Power Rail	**Add-in card**
3.3 V+/-0.3 V	7.6 A Max (System dependent)
5 V+/- 5 %	5 A Max (System dependent)
12 V+/- 5 %	500 mA Max.
-12 V +/- 5 %	100 mA Max.

2.7.4.6 COM-Express PCI interface supply voltage

The COM-Express PCI interface is a 3.3V signaling environment with 5V tolerance for I/O signals. If a universal PCI connector is used at the carrier board, a jumper design to select Vio for 5V and 3.3V is necessary. Otherwise, a suitable Vio voltage should be designed for the 5V or 3.3V connector.

Table 2-26 Add-in Card Supplied Power Selection

Symbol	3.3 V Connector	5 V Connector	Universal Connector
Vio	3.3 V	5 V	Jumper select

Note:

1. Note the riser card supply voltage and do not plug it into the wrong supply voltage. If a universal connector is used, make sure the Vio jumper setting is correct when plugged into the riser card.
2. Advantech's demo carrier board provides a 5V connector and 5V Vio for PCI slots. Plugging a 3.3 V riser card in the wrong direction will damage the carrier board or riser card.

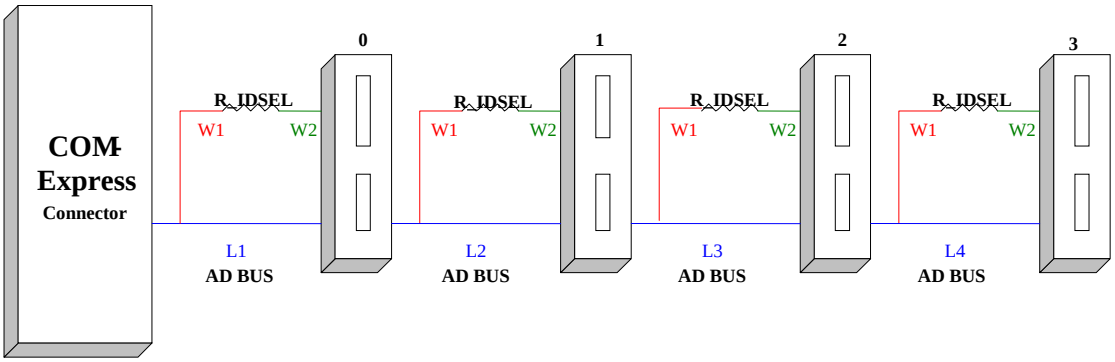
2.7.5 Layout Guidelines

The following represents a summary of the routing guidelines for PCI devices. Simulations assume that PCI cards follow the PCI Local Bus Specification, Revision 2.3, trace length guidelines.

2.7.5.1 PCI Bus Layout Example with IDSEL

The following guidelines apply to platforms with nominal impedances of 55 Ω ± 10%.

PCI AD Bus should be routed as daisy chain to PCI expansion slots.



PCI AD Bus should be routed as daisy chain to PCI expansion slots

Figure 2-39 PCI Bus Layout Example with IDSEL

Table 2-27 PCI Data Signals Routing Summary

Trace Impedance	PCI Routing Requirements	Topology	Maximum Trace Length (unit: inch)			
			L1	L2	L3	L4
55 Ω +/- 10%	5 mils width, 5 mils spacing (based on stack-up assumptions)	2 Slots W1 = W2 = 0.5 inches, R_IDSEL = 300 to 900.	10	1.0		
		3 Slots W1 = W2 = 0.5 inches, R_IDSEL = 300 to 900.	10	1.0	1.0	
		4 Slots W1 = W2 = 0.5 inches, R_IDSEL = 300 to 900.	10	1.1	1.1	1.1

2.7.5.2 PCI Clock Layout Example

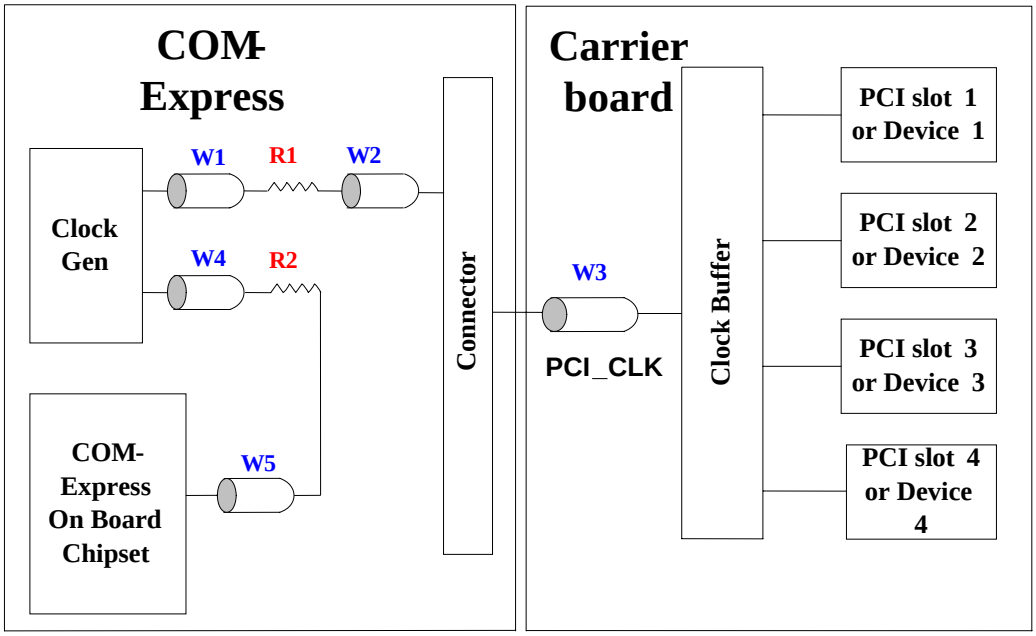


Figure 2-40 PCI Clock Layout Example

Table 2-28 PCI Clock Signals Routing Summary

Trace Impedance	PCI Routing Requirements	Topology	Maximum trace Length on Carrier Board	Damping Resistor
55 Ω +/- 10%	5 mils width, 50 mils spacing (based on stack-up assumptions)	2 ~ 4 Devices	W3: 15 inches	R1: 33 Ω R2: 33 Ω

Note:
Clock skew between PCI slots/devices should be less than 2 ns@ 33 MHz and 1 ns@ 66 MHz. The recommended value of the clock trace tolerance of W3 (a,b,c,d) is 5 inches (Maximum).

2.8 Universal Serial Bus (USB)

The Universal Serial Bus (USB) provides a bi-directional, isochronous, hot-attachable Plug and Play serial interface for adding external peripheral devices such as game controllers, communication devices and input devices on a single bus. COM-Express Modules provide up to eight USB 2.0 ports.

2.8.1 Signal Description

Table 2-29 shows COM-Express USB signals, including pin number, signals, I/0 and descriptions.

Table 2-29 USB Signals Description

Pin	Signal	I/O	Description
- -	USB[0:7]+ USB[0:7]-	I/O	USB differential pairs, channels 0 through 7
B44	USB_0_1_OC#	I	USB over-current sense, USB channels 0 and 1. A pull-up for this line **shall** be present on the module. An open drain driver from a USB current monitor on the Carrier Board **may** drive this line low.
A44	USB_2_3_OC#	I	USB over-current sense, USB channels 2 and 3. A pull-up for this line **shall** be present on the module. An open drain driver from a USB current monitor on the Carrier Board **may** drive this line low.
B38	USB_4_5_OC#	I	USB over-current sense, USB channels 4 and 5. A pull-up for this line **shall** be present on the module. An open drain driver from a USB current monitor on the Carrier Board **may** drive this line low.
A38	USB_6_7_OC#	I	USB over-current sense, USB channels 6 and 7. A pull-up for this line **shall** be present on the module. An open drain driver from a USB current monitor on the Carrier Board **may** drive this line low.

2.8.2 DC Specifications

Table 2-30 DC specification of USB signals

Symbol	Parameter	Min	Max	Unit
V_{bus}	High-power port supply voltage	4.75	5.25	V
V_{bus}	Low-power port supply voltage	4.75	5.25	V
V_{IL}	Input Low Voltage	-	0.8	V
V_{IH}	Input High Voltage(driven)	2.0	-	V
V_{IHZ}	Input High Voltage(floating)	2.7	3.6	V
V_{OL}	Output Low Voltage	0	0.3	V
V_{OH}	Output High Voltage	2.8	3.6	V

Notes: For high power function, the Max Supply current for each port is 500mA.

2.8.3 USB Spec.

Refer to “Universal Serial Bus Specification Revision 2.0, April 27, 2000”

2.8.4 Schematic Guidelines

The Figure 2-41 shows the USB connections for COM-Express USB signals. The ESD are recommended and the capacitors are reserved for EMI compliance which are usually not loaded.

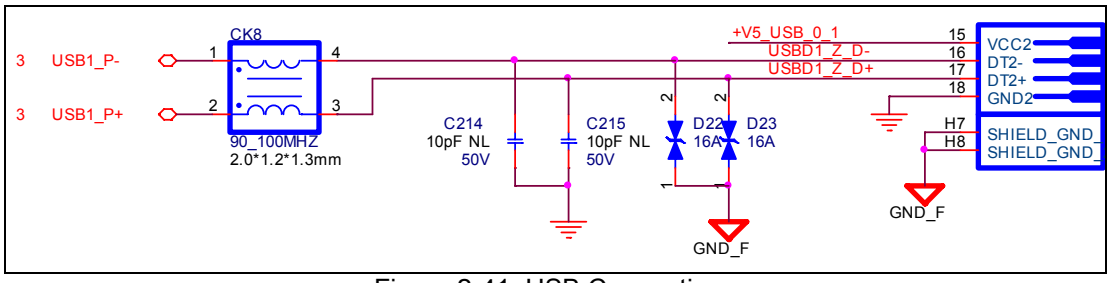


Figure 2-41 USB Connections

2.8.4.1 Low ESR Capacitor

The hot plug function is one of the popular features of the USB devices. The design of the USB power-decoupling circuits must absorb the momentary current surge from hot plugging an unpowered device. Reducing the capacity of decoupling capacitors is not recommended. These USB power capacitors should be selected as low ESR and low inductance.

2.8.4.2 ESD or EMI suppression components

Additional ESD or EMI suppression components could be implemented on the USB data lines. It's important to place the ESD and EMI components near the external USB connector and make it grounded by the low-impedance ground plane.

The common mode choke is recommended to be used for USB2.0 EMI consideration. Figure 2-42 shows the schematic of a typical common mode choke and ESD suppression component, which are placed as close as possible to the USB connector signal pins.

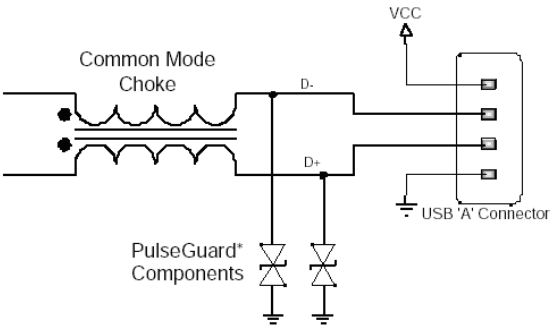


Figure 2-42 Common Mode Choke and ESD suppression design

The ESD components are generally needed for ESD testing. The common mode chocks are generally adopted for USB 2.0 interface.

2.8.4.3 Over-Current Protection

The Over-current protection on the external USB power lines is required to prevent the power faults from external USB devices or cables. The USBOC# signal is used to input over-current conditions to the system hardware and software. The over-current protection mechanism typically allows relatively high currents to flow for small periods before the current goes over-limit or is interrupted.

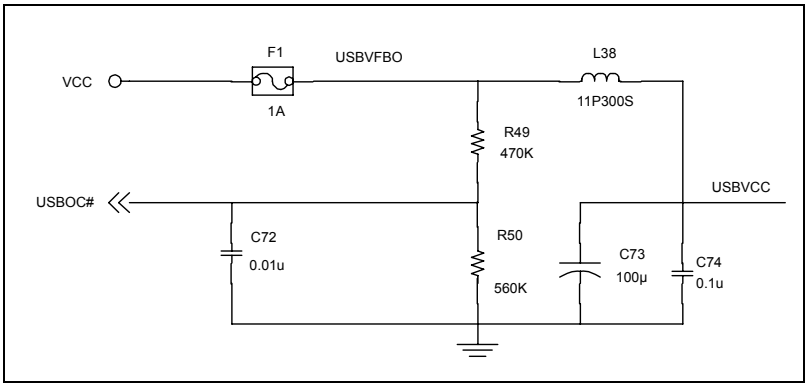


Figure 2-43 Overcurrent Circuit

The poly-switch in Figure 2-43 generally could not switch off fast enough. Over-current caused by an external USB device may impact the power supply of the carrier board. For fast response of sensing and power cutting, the active protection circuits shown in Figure 2-44 are recommended. These devices may be used for per port protection of the USB power lines and direct connected to the USB_X_X_OC# signal.

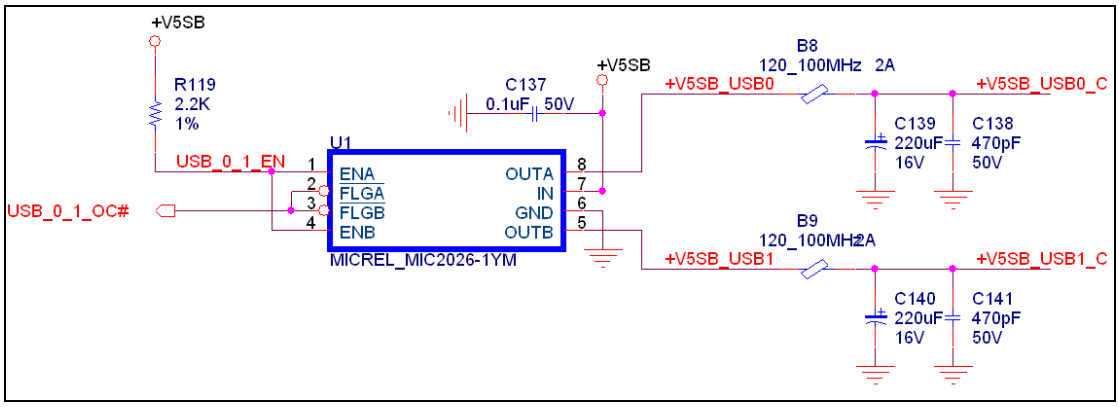


Figure 2-44 Power Switch with Overcurrent Protection Circuits

The over-current protection circuit is not implemented on the COM-Express Module. It should be implemented on the carrier board.

2.8.5 Layout Guideline

2.8.5.1 Differential pairs

The USB data pairs (ex. USB [0]+ and USB [0]-) should be routed on the carrier board as differential pairs, with a differential impedance of 90 Ω. The proper trace width and spacing to achieve the impedance is based on the impedance calculation of adopted PCB stack-up design.

The two data traces of each USB pair should be matched in length and kept at the same spacing as Figure 2-45. Sharp corners should be avoided. The loop routing areas should be minimized near the pins of COM-Express Module connector and USB connector. USB data pairs should be routed as far from other signals as possible for reducing crosstalk.

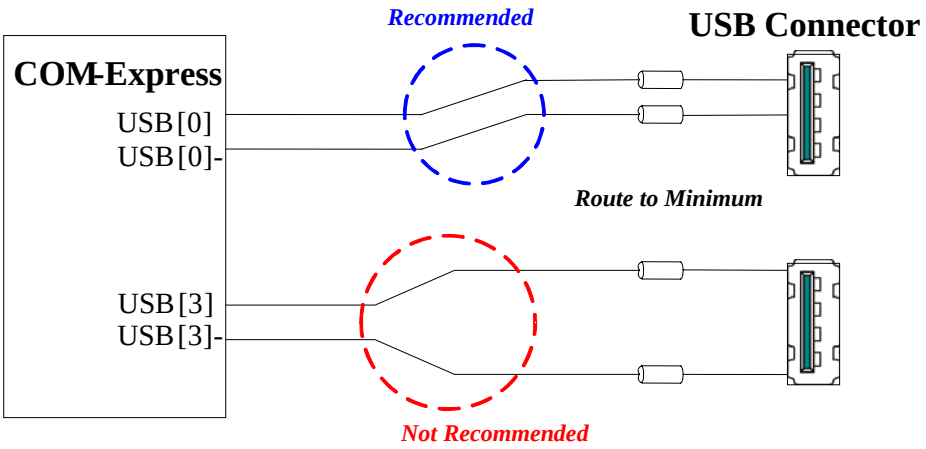


Figure 2-45 USB Layout Guidelines

2.8.5.2 Crossing a plane split

Figure 2-46 shows the data lines crossing a plane split. This will cause the return path currents poor and cause EMI problems. If a plane split crossing cannot be avoided, the capacitor bridge on the plane split is needed as shown in Figure 2-46.

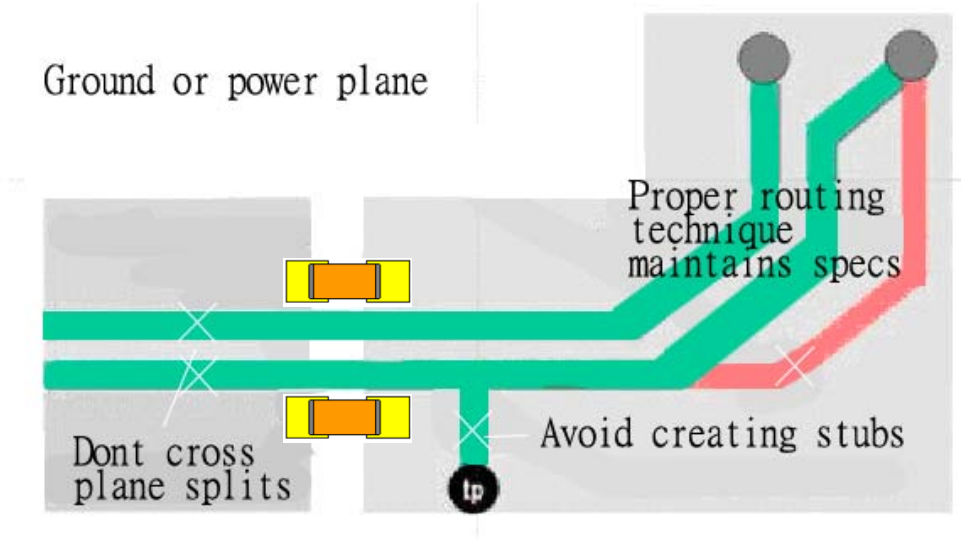


Figure 2-46 Violation of Proper Routing Techniques

2.9 LVDS

2.9.1 Signal Description

Table 2-31 shows the COM-Express LVDS signals.

Table 2-31 LVDS signals description			
Pin	**Signal**	**I/O**	**Description**
A71,73,75,78 A72,74,76,79	LVDS_A[0:3]+ LVDS_A[0:3]-	O	LVDS Channel A differential pairs
A81 A82	LVDS_A_CK+ LVDS_A_CK-	O	LVDS Channel A differential clock
B71,73,75,77 B72,74,76,78	LVDS_B[0:3]+ LVDS_B[0:3]-	O	LVDS Channel B differential pairs
B81 B82	LVDS_B_CK+ LVDS_B_CK-	O	LVDS Channel B differential clock
A77	LVDS_VDD_EN	O	LVDS panel power enable
B79	LVDS_BKLT_EN	O	LVDS panel backlight enable
B83	LVDS_BKLT_CTRL	O	LVDS panel backlight brightness control
A83	LVDS_I2C_CK	O	I2C clock output for LVDS display use
A84	LVDS_I2C_DAT	O	I2C data line for LVDS display use

Note: The LVDS I2C voltage level is 3.3V main power.

2.9.2 DC Specifications

Table 2-32 LCD I/O Voltage					
Symbol	**Parameter**	**Min**	**Max**	**Unit**	**Note**
V_{IL}	Input Low Voltage	-0.5	0.8	V	
V_{IH}	Input High Voltage	2.0	Vcc+0.5	V	
V_{OL}	Output Low Voltage	-	0.55	V	Iol= 4.0mA
V_{OH}	Output High Voltage	2.4	-	V	Ioh=-1.0mA

2.9.3 Schematic Guidelines

The LVDS signals can be routed directly from the COM-Express module to the LVDS connectors.Figure 2-47 shows one pair of LVDS connections. Each pair can use the common-mode choke for EMI compliance if needed.

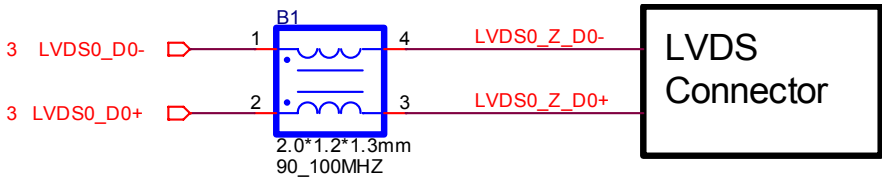


Figure 2-47 One LVDS Differential Pair with Choke Design

2.9.4 Layout Requirements

The timing skew minimization requires trace length matching between chipset die-pad to the pins of the LVDS connector. Match the package length difference between each signal group to minimize the timing variance. The COM-Express module has well designed routing lengths to compensate for the mismatch length of the chipset package.

Be sure to match the trace length on the carrier board. Table 2-33 shows the LVDS Signals Trace Length Mismatch Mapping.

Table 2-33 LVDS Signals Trace Length Mismatch Mapping

Siganl group	Data Pair	Signal matching	Clocks Associated with the channel	Clock Matching	Data To Associated Clock Matching
CHANNEL A	LVDS_A[0]+ LVDS_A[0]-	±10 mils	LVDS_A_CK+ LVDS_A_CK-	±20 mils	±20 mils
	LVDS_A[1]+ LVDS_A[1]-	±10 mils			
	LVDS_A[2]+ LVDS_A[2]-	±10 mils			
	LVDS_A[3]+ LVDS_A[3]-	±10 mils			
CHANNEL B	LVDS_B[0]+ LVDS_B[0]-	±10 mils	LVDS_B_CK+ LVDS_B_CK-	±20 mils	±20 mils
	LVDS_B[1]+ LVDS_B[1]-	±10 mils			
	LVDS_B[2]+ LVDS_B[2]-	±10 mils			
	LVDS_B[3]+ LVDS_B[3]-	±10 mils			

Each LVDS signal should be trace length matched to its associated clock strobe within ±10 mils. The Channel A clock strobe pair must also be trace length matched to the Channel B clock strobe pair within ±10 mils.

Routing for LVDS transmitter signals of different traces are terminated across 100Ω ± 15% and should be routed as following points:

- It is necessary to maintain the differential impedance, Zdiff = 100Ω ± 15%, where all traces are closely routed in the same area on the same layer.
- Isolate all other signals from the LVDS signals to prevent coupling from other sources to the LVDS lines.
- The LVDS transmitter timing domain signals have maximum trace length of 10 inches. Be sure that the max trace length routed on the carrier board is 7.5 inches. Please refer to Advantech layout checklist for detailed info.
- Clocks must be matched to the associated data signals to within ± 10 mils.
- Channel-to-Channel clock length must be matched to within ± 10 mils.
- Minimum spacing between neighboring trace pair is 20 mils.
- Traces must be ground referenced.
- When choosing cables, it is important to remind that the differential impedance of cable should be 100Ω. The cable length should be less than 0.5 meter for better signal quality.

2.10 VGA

COM-Express Module provides analog display signals. There are three signals -- red, green, and blue -- which send color information to a VGA monitor. Analog levels between 0 (completely dark) and 0.7 V (maximum brightness) on these control lines tell the monitor what intensities of these three primary colors to combine to make the color of a dot (or pixel) on the monitor's screen.

2.10.1 Signal Description

Table 2-35 shows COM-Express VGA signals

Table 2-34 VGA signals description

Pin	Signal	I/O	Description
B89	VGA_RED	O	Red analog video output signal for CRT monitors, designed to drive a 37.5 Ω equivalent load.
B91	VGA_GRN	O	Green analog video output signals for CRT monitors, designed to drive a 37.5 Ω equivalent load.
B92	VGA_BLU	O	Blue analog video output signals for CRT monitors, designed to drive a 37.5 Ω equivalent load.
B93	VGA_HSYNC	O	Horizontal Sync: This output supplies the horizontal synchronization pulse to the CRT monitor.
B94	VGA_VSYNC	O	Vertical Sync: This output supplies the vertical synchronization pulse to the CRT monitor.
B95	VGA_I2C_CK	I/O	DDC clock line. It can be used for a DDC interface between the graphics controller chip and the CRT monitor
B96	VGA_I2C_DAT	I/O OD	DDC data line. It can be used for a DDC interface between the graphics controller chip and the CRT monitor

2.10.2 DC Specifications

Table 2-35 Hsync and Vsync signals specification

Symbol	Parameter	Min	Max	Unit	Note
V_{IL}	Input Low Voltage	0	0.5	V	
V_{IH}	Input High Voltage	2.4	5.5	V	
V_{OL}	Output Low Voltage	-	0.8	V	
V_{OH}	Output High Voltage	2.0	-	V	

Table 2-36 RGB Voltage

Symbol	Parameter	Min	Max	Unit	Note
R	Red analog video output signal **Max.** luminance voltage	0.665	0.77	V	
G	Green analog video output signal **Max.** luminance voltage	0.665	0.77	V	
B	Blue analog video output signal **Max.** luminance voltage	0.665	0.77	V	
R	Red analog video output signal **min.** luminance voltage	0 (Typical)		V	
G	Green analog video output signal **min.** luminance voltage	0 (Typical)		V	
B	Blue analog video output signal **min.** luminance voltage	0 (Typical)		V	

The HSYNC / VSYNC are TTL signals, RED/GREEN/BLUE are 0.7V peak-to-peak (0V for black level, 0.7V for full color intensity).

2.10.3 VGA Specifications

Please refer to “VESA and Industry Standards and Guidelines for Computer Display Monitor Timing Version 1.0, Revision 0.8” for the monitor timing specification.

2.10.4 Schematic Guidelines

The reference schematic of VGA is shown in FIG 2-48.
The VGA_I2C_CK and VGA_I2C_DAT signals must connect to the CRT monitor to detect the plug-and-play and monitor-type info.

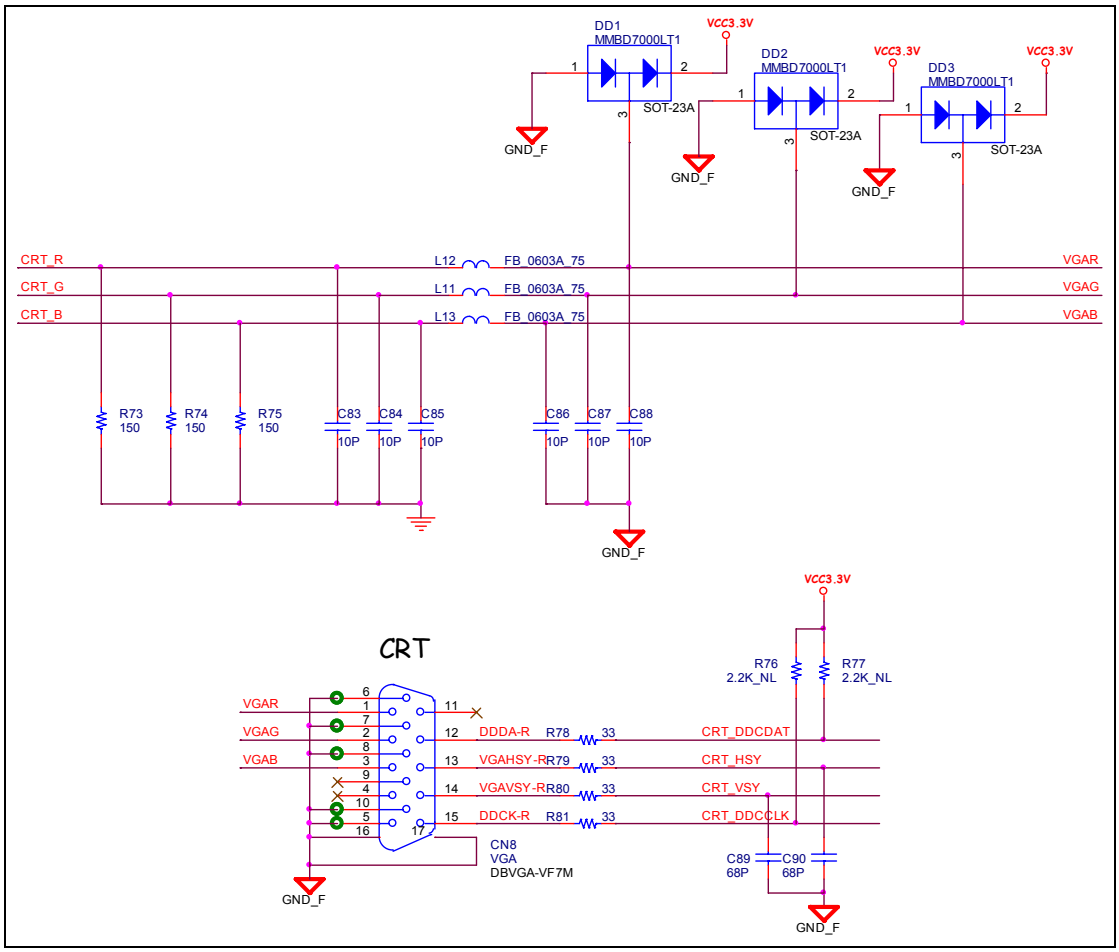


Figure 2-48 VGA reference schematic

The DDC pulled-up and ESD protection voltage could be 3.3V or 5V depends on the power map of the carrier board.

2.10.5 Layout Guideline

2.10.5.1 RLC Components

The RGB outputs are current sources and therefore require 150 Ω load resistors from each RGB line to analog ground to create the output voltage (approximately 0 to 0.7 volts). These resistors should be placed near the VGA port (a 15-pin D-SUB connector). Serial ferrite beads for the RGB lines should have high frequency characteristics to eliminate relative noise. The 27Ω ~ 33Ω series damping resistors

for HSY and VSY should be placed near the D-SUB connector. Please refer to Advantech layout checklist for detail recommended resistor value.

The impedance control of VGA is important for VGA signal quality. The RGB traces with proper width (for 50Ω impedance) should be routed between the 150Ω resistor and COM-Express connector. And the routing section between 150Ω resistor and VGA connector should be kept as short as possible with proper trace width for 75Ω impedance.

2.11 TV-Out

The TV-out display (TV DAC) interface consists of 3 outputs which can be used in different combinations to support component video, S-video or composite video.

2.11.1 Signal Description

Table 2-37 TV signals description

Pin	Signal	I/O	Description
B97	TV_DAC_A	O	TVDAC Channel A Output supports the following: Composite video: CVBS Component video: Chrominance (Pb) analog signal S-Video: not used
B97	TV_DAC_B	O	TVDAC Channel B Output supports the following: Composite video: not used Component video: Luminance (Y) analog signal S-Video: Luminance analog signal
B97	TV_DAC_C	O	TVDAC Channel C Output supports the following: Composite video: not used Component video: Chrominance (Pr) analog signal S-Video: Chrominance analog signal

2.11.2 Schematic Guidelines

2.11.2.1 Termination resistor, output filter and ESD protection diodes

There are three DAC output pins: TV_DAC_A, TV_DAC_B, and TV_DAC_C. There are two 150Ω ± 1% resistors in each signal. One of the 150 Ω ± 1% parallel termination resistors is already implemented on the COM-Express Module. The other 150 Ω ± 1% parallel termination resistor should be implemented on the carrier board. Figure 2-49 shows the connection diagram of TV-out.

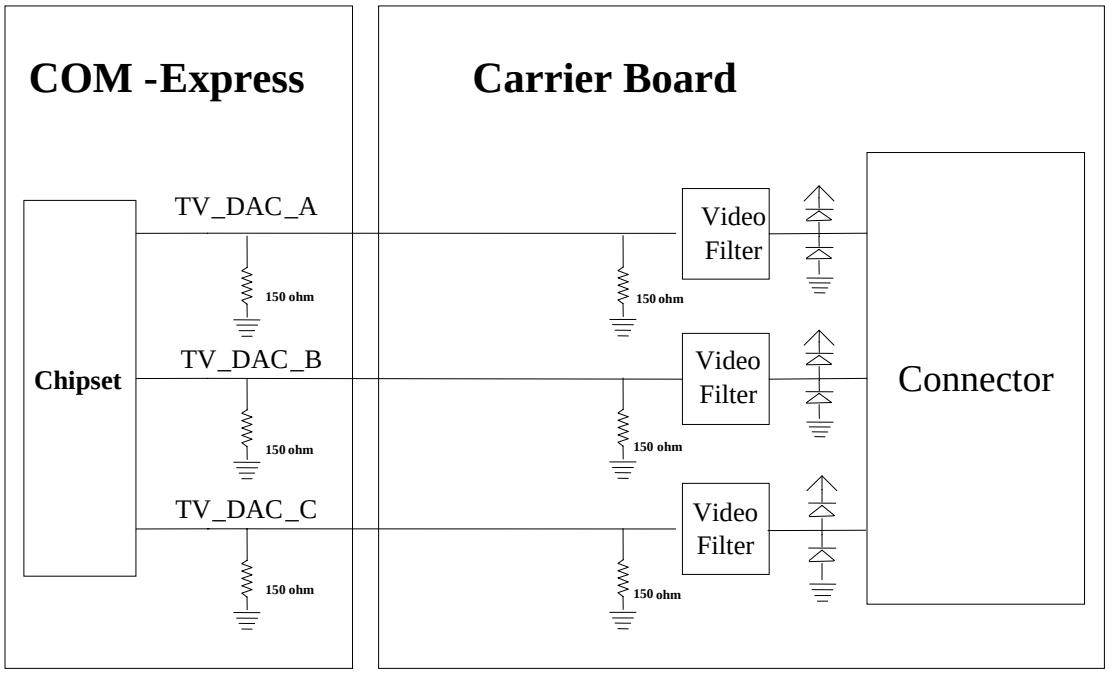


Figure 2-49 Connection Diagram of TV-out

2.11.2.2 ESD Diode

ESD diodes are required for each TV DAC channel output. The diodes should connect between the VCC power plane (from the regulator) and ground. These diodes should have a low C rating (~ 5 pF max) and a small leakage current (~ 10 uA at 120℃).The diodes should be placed to keep the inductance of the VCC power rail connection as low as possible. The diode placement should be similar for all three output signals and should not be close to any other signal, especially video or clock signals. In addition, one decoupled capacitor, C1= 0.1 uF should be placed closely and across the ESD diodes to reduce noise on the VCC. The VCC could be 3.3V or 5V power rail.

2.11.2.3 TV DAC Video Filter

A video filter is required for each TV DAC channel output signal. This video filter is to be placed close to the TV connector. The separation between each of the 3 video filters for the TV DAC channels should be far than a minimum of 50 mils or greater to minimize crosstalk. This is especially important for the TVDAC_B and TVDAC_C channels (S-video signals). Figure 2-50 shows the TV DAC Video Filter.

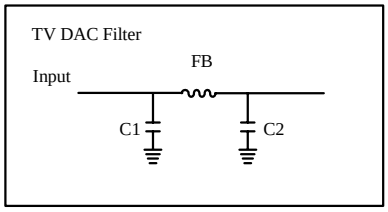


Figure 2-50 TV DAC Video Filter

The video filter is designed for a cutoff frequency of at least 30 MHz and a gain of -3 dB. Table 2-38 shows the TV DAC Video Filter component.

Table 2-38 TV DAC Video filter components				
Component	**Value**	**Tolerance**	**Voltage/Current**	**Type**
C1	6 pF	± 20%	16 V	Ceramic
C2	6 pF	± 20%	16 V	Ceramic
FB	150 Ω@ 100 MHz	± 25%	100 mA	

2.11.3 Layout Guidelines

2.11.3.1 TV DAC routing

The minimum spacing between each TV DAC signal is 40 mils, but 50 mils is preferred. A maximum amount of spacing should be used between each TV DAC signal as well as to all other toggling signals. This helps prevent crosstalk between the TV DAC signals and other signals. The routing for each TV DAC signal should also be matched and balanced. All TV DAC signals should be routed on the same layer, have a similar number of bends, the same number of vias, etc. All routing should be done with ground referencing as well. Figure 5-37 shows the TV DAC routing topology. Total length of each signal should be less than 12 inches. Intra-pair length difference should be less than 200 mils.

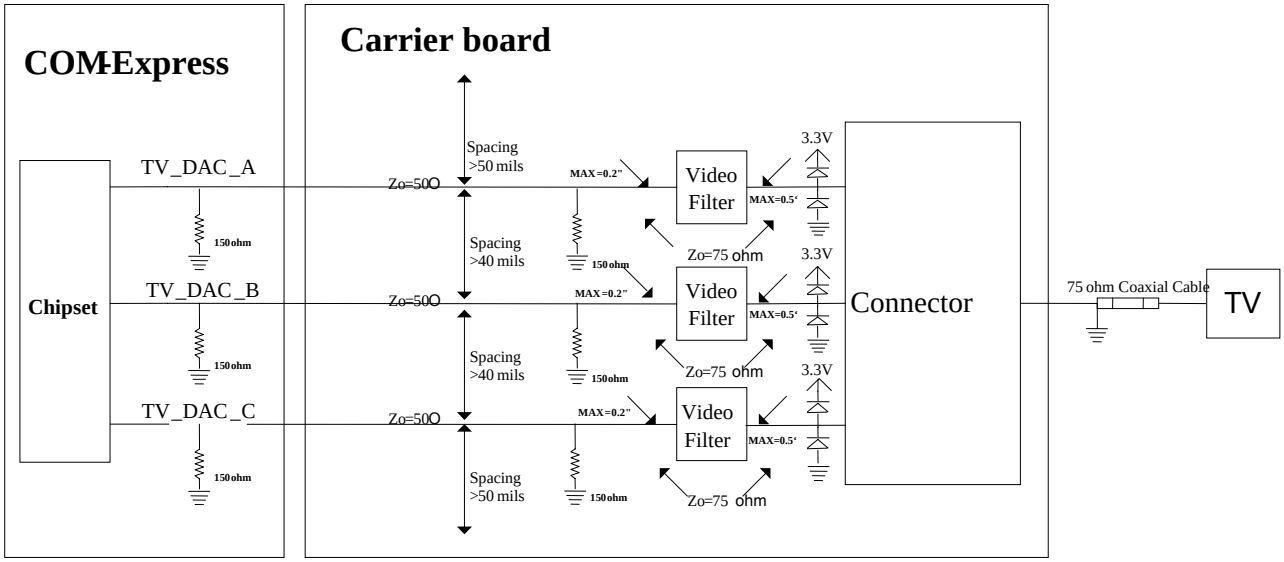


Figure 2-51 TV DAC Schematic

2.12 LPC

COM-Express provides a LPC interface to some devices as Super I/O, FWH, TPM, and others.

2.12.1 Signal Description

Table 2-39 shows COM-Express LPC signals.

Table 2-39 LPC signals description			
Pin	**Signal**	**I/O**	**Description**
B4,5,6,7	LPC_AD[0:3]	O	LPC multiplexed address, command and data bus

B3	LPC_FRAME#	O	LPC frame indicates the start of an LPC cycle
B8,B9	LPC_DRQ[0:1]#	O	LPC serial DMA request
A50	LPC_SERIRQ	O	LPC series interrupt
B10	LPC_CLK	O	LPC clock output – 33MHz nominal

2.12.2 Schematic Guidelines

The schematic guidelines are:

LPC_CLK should be connected to pin B10 of COM-E connector.
LRESET should be connected to CB_RESET# as pin B50 of COM-E connector.
LPC_FRAME# (cycle termination) is shared with FWH and the SIO.

Some legacy interfaces are using Super I/O, as Serial port, Parallel port, Floppy, IR, KBC, via the LPC Bus of COM-Express pins. Figure 5-54 shows the architecture of the LPC interface. More information can be found in the Super I/O datasheet.

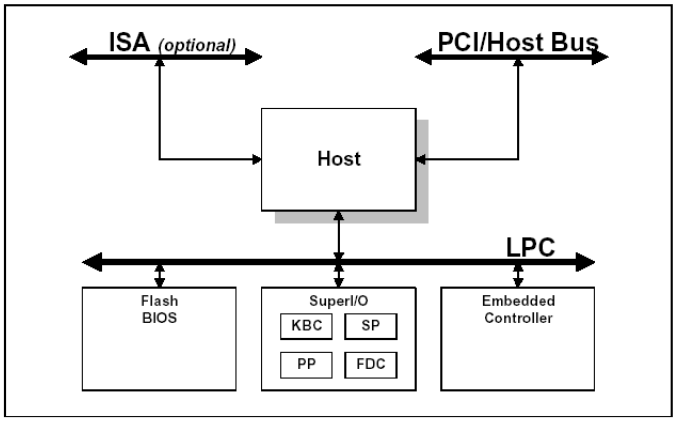


Figure 2-52 Architecture of LPC interface

The I/O address for Super I/O is generally set as 4Eh by Advantech. For example, the W83627HG Pin 51 (signal RTSA#) should be pulled-high 4.7K ohm to 5V for I/O address 4Eh strapping. If another address is requested as 2Eh, then a customized BIOS is needed. Contact Advantech for any BIOS requests for specific Super I/O configuration.

The reset signal of theLPC interface should be connected to the CB_RESET# pin of the COM-Express Module.

2.12.3 Layout Guidelines

If there is a TPM device, place it close to the other LPC peripherals such as FWH and Super I/O.

2.12.4 Application Notes for Super I/O Functions

2.12.4.1 Serial Port

For complying with EMI limits, separate the digital ground to the frame ground by adding beads and capacitors to the carrier board. Figure 2-53 shows the Serial Bus Connection with EMI considerations.

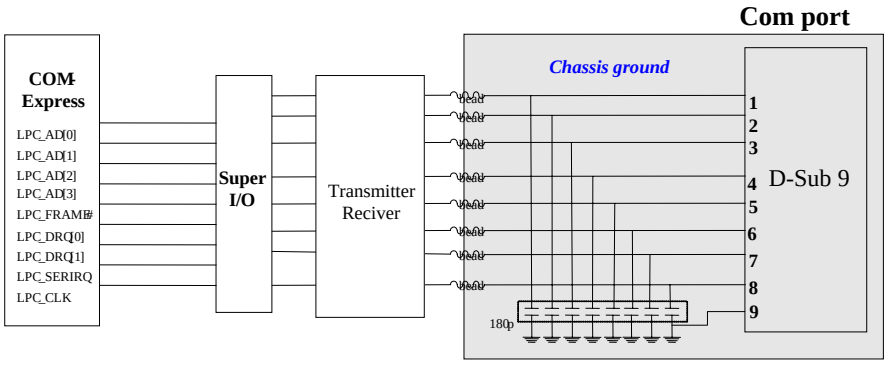


Figure 2-53 Serial Bus Connection

2.12.4.2 PS/2 Keyboard and Mouse

For a general design consideration, the keyboard and mouse should be far away from audio and VGA signal traces to avoid crosstalk. According to the general keyboard and mouse power specifications, the traces of keyboard and mouse power trace should be routed to afford 1A. The power can be Vcc main power, or 5VSB if wake-up from S3 is needed. The reference schematic is shown in Figure 2-54.

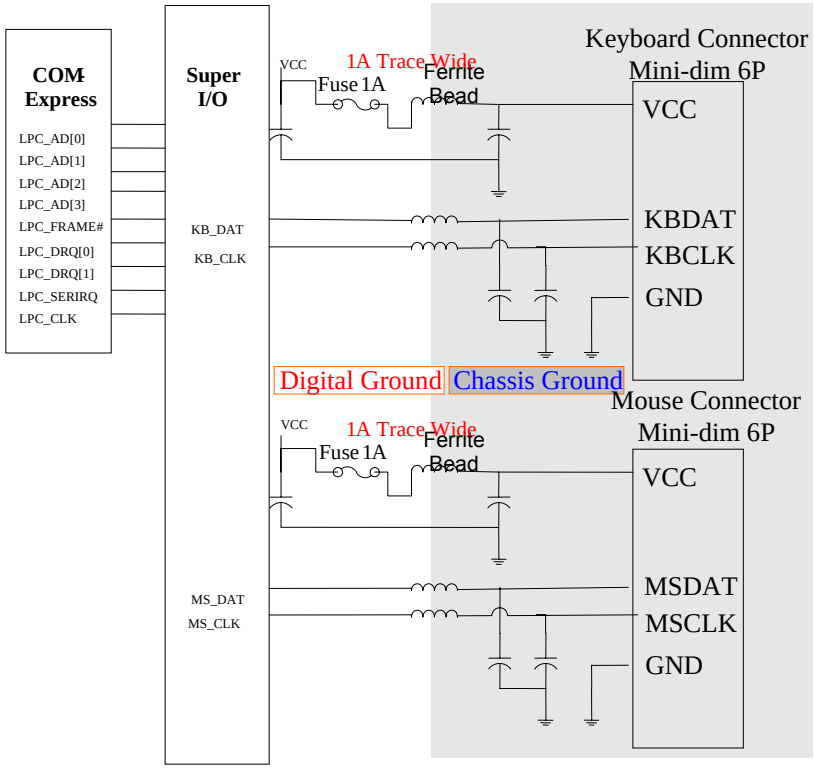


Figure 2-54 Keyboard and Mouse connection

To avoid EMI and ESD, the ground plane of the keyboard / mouse connector and other digital ground planes, should be separated with an isolation moat, which is recommended to be at least 40 mils in width. Digital ground and chassis ground should be connected via screw holes to assure the integrity of the ground plane.

2.12.4.3 LPT / Floppy

The pulled-up resistors of LPT and Floppy are shown in Figure 2-55 and Figure 2-56.

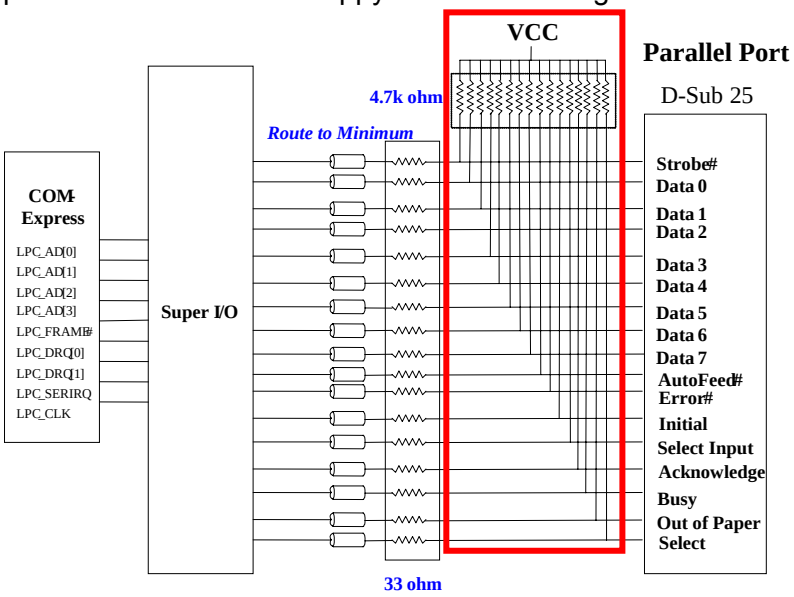


Figure 2-55 LPT Connection

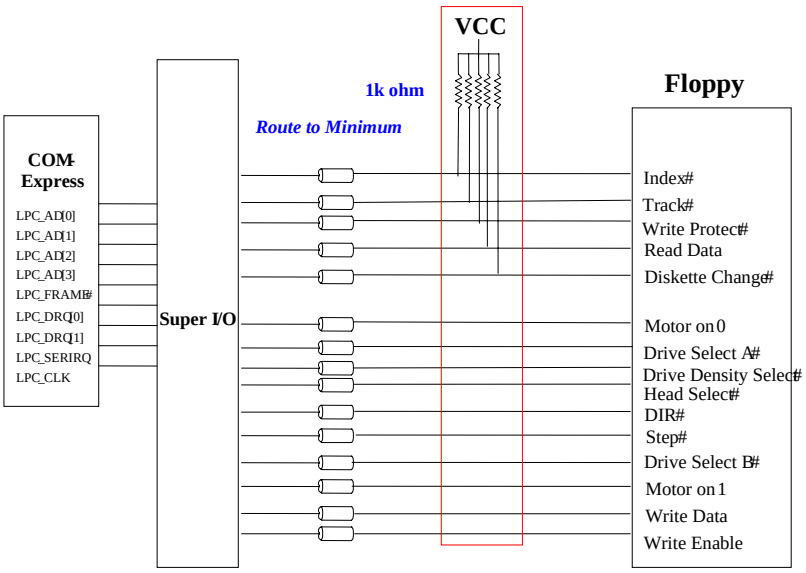


Figure 2-56 Floppy Connection

2.12.4.4 EMI Considerations

Legacy I/O such as LPT, Floppy or COM ports should be physically isolated between digital circuitry, analog circuitry, power and ground planes. This isolation prevents noise sources located elsewhere on the PCB from corrupting susceptible circuits. An example is power plane noise from digital circuits entering the power pins of analog devices, audio components, I/O filters and interconnects.

Each I/O port must have a partitioned ground/power plane. Lower frequency I/O ports may be bypassed with high-frequency capacitors located near the connectors. Trace routing on the PCB must be controlled to avoid coupling RF currents into the cable

shield. A clean ground must be located at the point where cables leave the system. Both power and ground planes must be treated equally, as these planes act as a path for RF return currents.

To implement a clean ground, use of a partition or moat is required. The clean area should be:

1. 100% isolated with I/O signals entering and exiting via an isolation transformer or an optical device.
2. Data line filtered;
3. Filtered through a high-impedance common-mode inductor (choke) or protected by a ferrite bead-on-lead component.

2.12.4.5 ESD Protection

The PCB should be against the electro-static discharge (ESD) events which might enter I/O signal and electrical connection points. In order to prevent component or system failures due to external ESD impulses that may be propagated through both radiated and conducted mechanisms.

For ESD protection that may be implemented on a PC for high-level pulse suppression include the following components:

1. High voltage disc-ceramic capacitors must be rated at 1KV minimum. Lower-voltage capacitors may be damaged by the first occurrence of an ESD pulse. This capacitor must be located adjacent to the I/O connector.

2. TVS components are semiconductor devices specifically designed for transient voltage suppression applications. The features are the stable and fast time constant to avalanche, and a stable clamping level after avalanche.

3. LC filters is a combination of an inductor and a capacitor to ground. This constitutes a low-pass LC filter that prevents high-frequency ESD energy from entering the system. The inductor presents a high-impedance source to the pulse, thus attenuating the impulse energy that enters the system. The capacitor located on the input side of the inductor will shunt high-frequency ESD spectral level components to ground. An additional benefit is the enhancement of radiated EMI noise suppression.

2.13 Power Management Signals

B12	PWRBTN#	I	Power button to bring system out of S5 (soft off), active on rising edge.
B49	SYS_RESET#	I	Reset button input. Active low input. System is held in hardware reset while this input is low, and comes out of reset upon release.
B50	CB_RESET#	O	Reset output from module to Carrier Board. Active low. Issued by module chipset and may result from a low SYS_RESET# input, a low PWR_OK input, a VCC_12V power input that falls below the minimum specification, a watch dog timeout, or may be initiated by the module software.
B24	PWR_OK	I	Power OK from main power supply. A high value indicates that the power is good
B18	SUS_STAT#	O	Indicates system suspend operation; used to notify LPC devices.
A15	SUS_S3#	O	Indicates system is in Suspend to RAM state. Active low output.
A18	SUS_S4#	O	Indicates system is in Suspend to Disk state. Active low output.
A24	SUS_S5#	O	Indicates system is in Soft Off state. Also known as “PS_ON” and can be used to control an ATX power supply.
B66	WAKE0#	I	PCI Express wake-up signal.
B67	WAKE1#	I	General purpose wake-up signals. May be used to Implement wake-up on PS2 keyboard or mouse activity.
A27	BATLOW#	I	Indicates that external battery is low.
B35	THRM#	I	Input from off-module temp sensor indicating an over-temp situation.
A35	THRMTRIP#	I	Active low output indicating that the CPU has entered thermal shutdown.

2.13.1 Power Good / Reset Input

The COM-Express Power OK (PWR_OK) input signal could be connected to an external power good circuit, or used as a manual reset input by grounding the pin with a momentary-contact push button switch. If an external circuit asserts this signal, it should be driven by an open-drain driver and held low for a minimum of 15ms to initiate a reset. The Figure 2-57 shows the application. Use of this power input is optional. The COM-Express Module generates its own power-on reset based on the internal monitor on the main input voltage and / or the internal power supply.

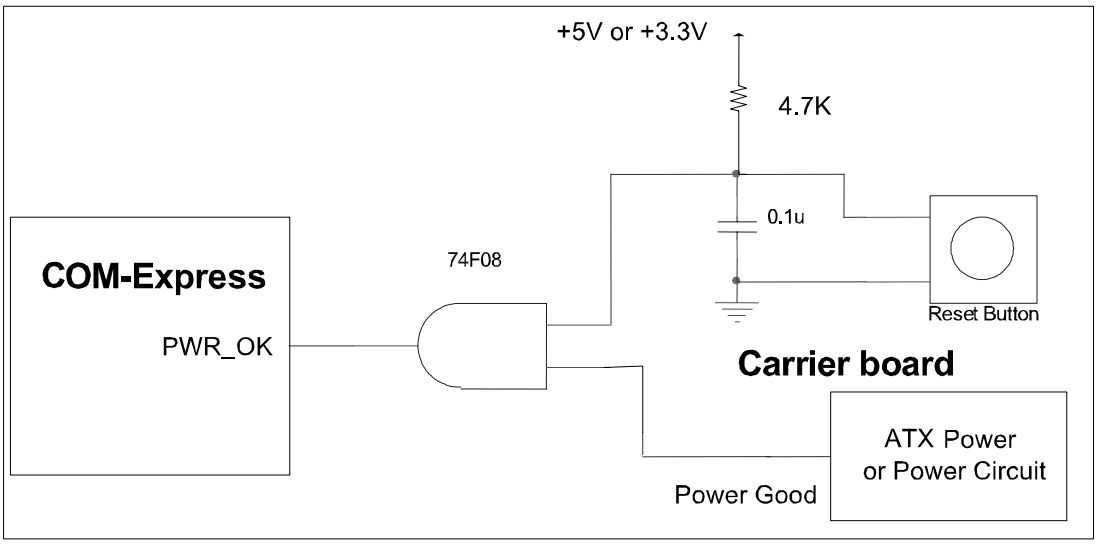


Figure 2-57 Power Good / Reset Input Applications

2.13.2 Reset signals

There are reset signals with inputs and outputs functions listed in the Table 2-40 Reset Signals.

Table 2-40 Reset Signals

Pin Name	Pin	Pin Type	Description	Where Use	Remark
SYS_RESET#	B49	I / 3.3VSB CMOS	Reset button input. Active low input. System is held in hardware reset while this input is low, and comes out of reset upon release.	From Reset Button to Module	
KBD_RST#	A86	I / 3.3V CMOS	Input to module from (optional) external keyboard controller that can force a reset. Pulled high on the module. This is a legacy artifact of the PC-AT.	Keyboard Controller(KBC)	Not Connected if KBC is not implemented on the carrier board
CB_RESET#	B50	O / 3.3V CMOS	Reset output from module to Carrier Board, active low.	LPC / PCIE / CPLD / others	.
PCI_RESET#	C23	O / 3.3V (5V tolerance) CMOS	PCI Reset output, active low.	PCI	
IDE_RESET#	D18	O / 3.3V CMOS	Reset output to IDE device, active low.	IDE / CF	
EXCD0_PERST#	A48	O / 3.3V CMOS	PCI ExpressCard slot 1: reset, active low.	Express Card	
EXCD1_PERST#	B47	O / 3.3V CMOS	PCI ExpressCard slot 2: reset, active low.	Express Card	

The reset signals input to COM-Express Module as SYS_RESET# and KBD_RST# can be Not Connected if not used.

Be sure to use the corresponding reset output from COM-Express Module. The CB_RESET# is the general reset for LPC / PCIE or other devices if PCI and IDE is not used as the COM-Express Type 1.

2.14 Miscellaneous

2.14.1 Miscellaneous Signal Descriptions

Table 2-41 Miscellaneous signal descriptions

Pin	Signal	I/O	Description
B32	SPKR	O	This is the PC speaker output signal from the COM-Express Module. Please connect this signal to the speaker.
B33	I2C_CK	O	General purpose I2C port clock output.
B34	I2C_DAT	I/O OD	General purpose I2C port data I/O line.
B13	SMB_CK	I/O OD	System Management Bus bidirectional clock line. Power sourced through 3.3 V standby rail.
B14	SMB_DAT	I/O OD	System Management Bus bidirectional data line. Power sourced through 3.3 V standby rail.
B15	SMB_ALERT#	I	System Management Bus Alert – active low input can be used to generate a SMI# (System Management Interrupt) or to wake the system. Power sourced through 3.3V standby rail.
A34	BIOS_DISABLE#	I	Module BIOS disable input. Pull low to disable module BIOS. Used to allow off-module BIOS implementations.
A86	KBD_RST#	I	Input to module from (optional) external keyboard controller that can force a reset. Pulled high on the module. This is a legacy artifact of the PC-AT.
A87	KBD_A20GATE	I	Input to module from (optional) external keyboard controller that can be used to control the CPU A20 gate line. The A20GATE restricts memory access to the bottom megabyte and is a legacy artifact of the PC-AT. Pulled high on the module.
B27	WDT	O	Output indicating that a watchdog time-out event has occurred.

2.14.2 SPKR

The SPKR output from the COM-Express Module is a CMOS level signal. It can control an external FET or logic gate that drives an external PC speaker. The SPKR output should NOT be directly connected to either a pulled-up or a pulled-down resistor. The SPKR signal is often used as a configuration strapping for the chipset on the COM-Express Modules. A pulled-up or pulled-down resistor on SPKR may override the internal strapping on the module and result in malfunction.

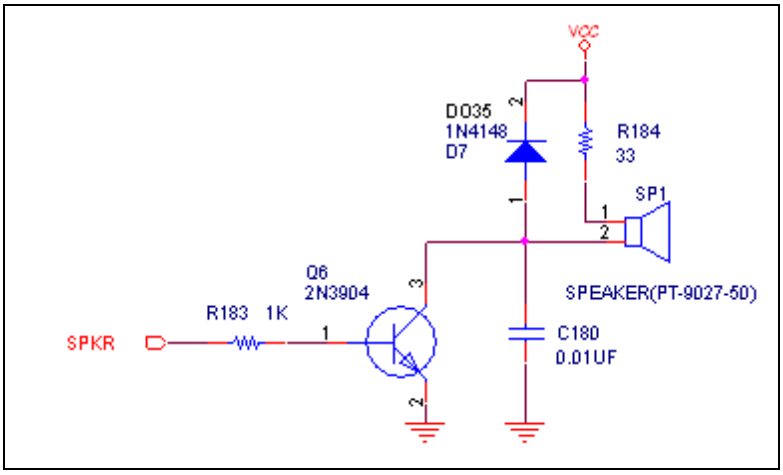


Figure 2-58 Speaker Schematics

2.14.3 I2C Bus

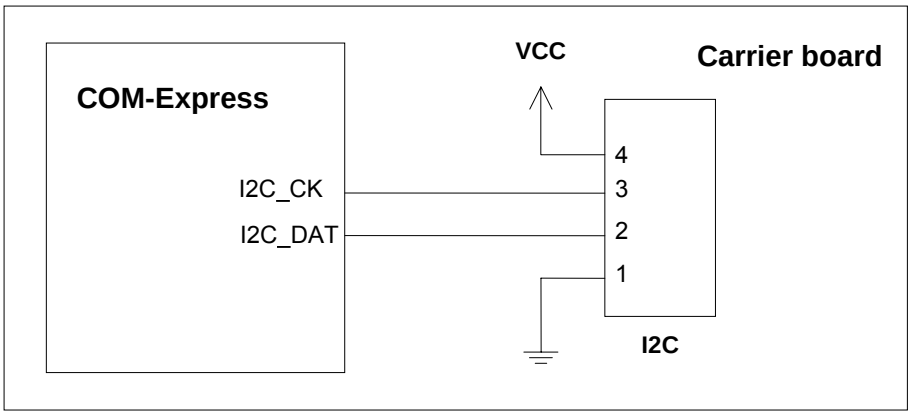


Figure 2-59 I2C Bus Connections

Most COM-Express Modules provide a software-driven I2C port for communication with external I2C slave devices. The Vcc is 3.3V main power.

2.14.3.1 DC Specifications

Table 2-42 I2C I/O Voltage

Symbol	Parameter	Min	Max	Unit	Note
V_{IL}	Input Low Voltage	-0.5	0.3Vcc	V	
V_{IH}	Input High Voltage	0.7Vcc	Vcc+0.5	V	
V_{OL}	Output Low Voltage	0	0.4	V	

*1. The I2C Bus Specification V2.1.
*2. Vcc is the voltage which the pulled-up resistor are connected.

2.14.4 I2C Spec

Please refer to “THE I 2C-BUS SPECIFICATION VERSION 2.1 JANUARY 2000” for the DAC AC Characteristics

2.14.5 SMBus

Most COM-Express Modules provide a SMBus port for communication with external SMBus slave devices. This port is also used internally in the COM-Express Module to communicate with onboard SMBus devices such as the SPD EEPROMs on SO-DIMMS, clock-generator chips, and hardware monitoring devices. The port is externally available on the COM-Express pins SMB_DAT and SMB_CK.

The addresses for any external SMBus devices must be chosen so that they do not conflict with the addresses that are used internally on the COM-Express Module. If the device offers externally controllable address options, it is desirable to implement carrier board resistor straps to allow the device to be set to at least two possible SMBus addresses.

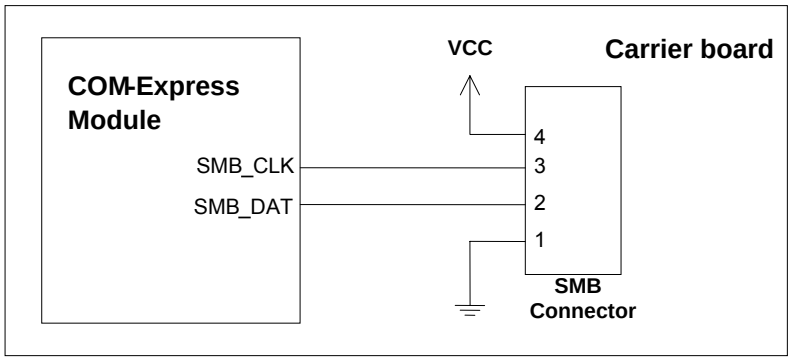


Figure 2-60 SMB Bus Connections

2.14.5.1 DC Specifications

Table 2-43 SMBus I/O Voltage

Symbol	Parameter	Min	Max	Unit	Note
V_{IL}	Input Low Voltage	-	0.8	V	
V_{IH}	Input High Voltage	2.1	Vcc	V	
V_{OL}	Output Low Voltage	-	0.4	V	

*1. System Management Bus (SMBus) Specification v2.0.
*2. Vcc is the voltage which the pulled-up resistor are connected.

2.14.5.2 SMBus Spec.

Refer to “System Management Bus (SMBus) Specification Version 2.0 August 3, 2000”

2.14.6 WDT

COM-Express Module provides a watch-dog function via the pin WDT. To prevent the system from hanging for a long time, the watchdog can generate a signal to reset the system and carrier board. The WDT pin is active-high output for carrier board power management applications.

Chapter 3 Power Management and Power Delivery

This chapter provides the ATX / AT power supply design recommendations for customer's reference.

3.1 Power Design Guidelines

3.1.1 ATX Power Delivery Block Diagram

The general ATX power source can supply 12V, -12V, 5V, -5V, 3.3V, 5VSB power. If other voltage is required (such as 3.3 VSB or LAN 2.5…) on the carrier board, the additional switching regulator or LDO should be implemented as Figure 3-1.

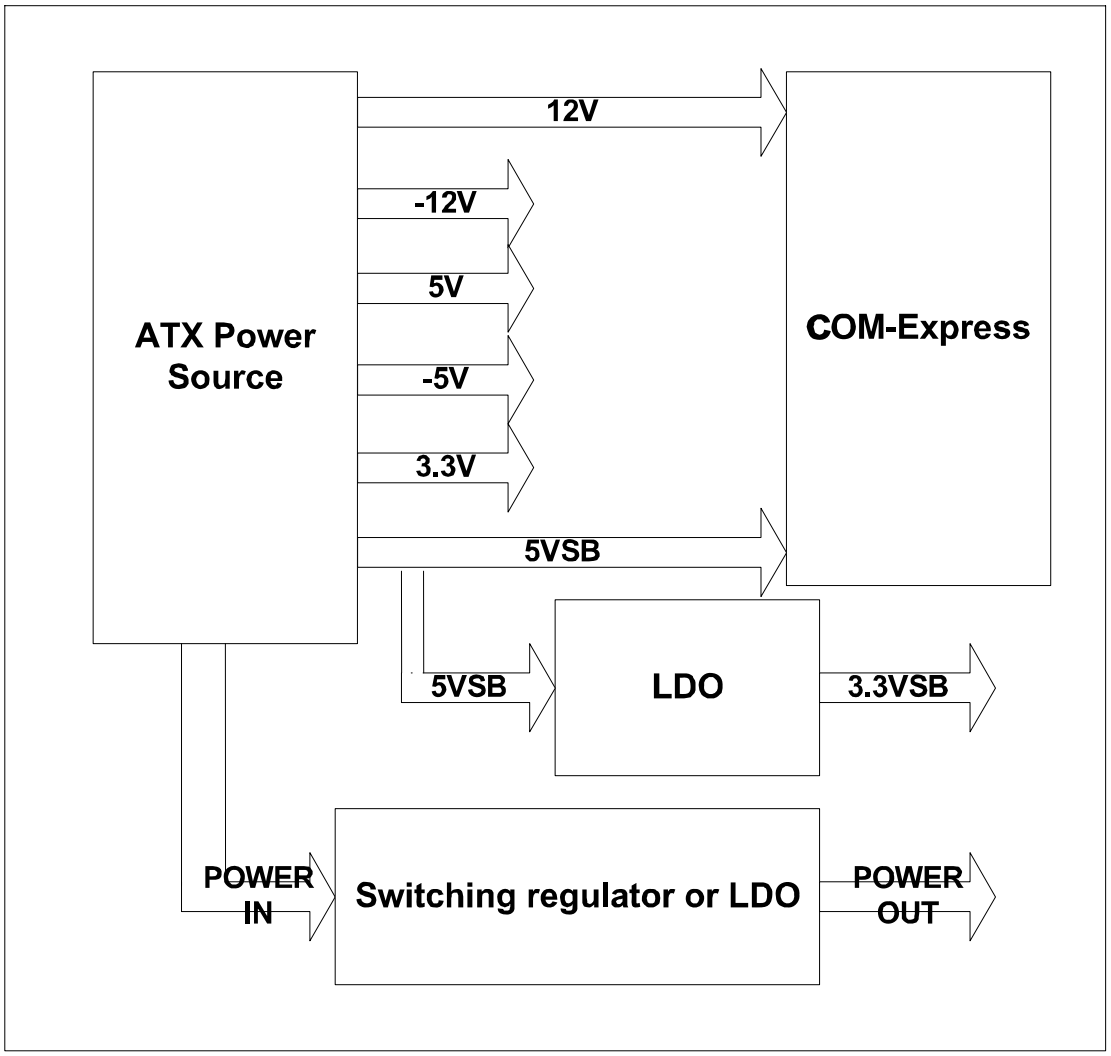


Figure 3-1 ATX Power Delivery Block Diagram

3.1.2 ATX power with Super I/O

If there is a Super I/O as Nuvoton (Winbond) W83627HG on the carrier board of Advantech, the PS-ON# is controlled by Super I/O with proper BIOS. While the power button designed as active high is pressed, the Super I/O will output an active low PWRBTN# signal to COM-E Module, then the SLP_S3# will be output high by chipset on the module which will inform the Super I/O output the PS-ON# active low to let the ATX power start supplying the main power.

3.1.3 ATX power w/o Super I/O

If there is without Super I/O to output the PS_ON# signal to ATX power switch, the SLP_S3# or SLP_S5# can be used to control the PS_ON# through a switch as Figure 3-2 shows.

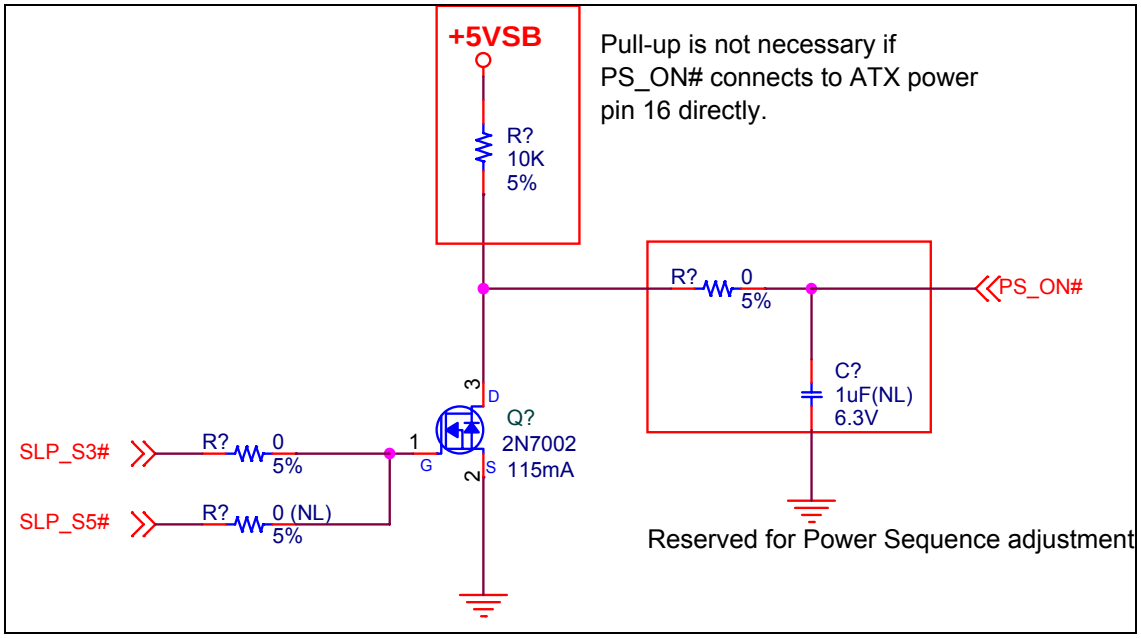


Figure 3-2 power circuits for ATX power supply w/o SIO

If the system supports S3 operation, the SLP_S3# should be used.

If S3 function is not needed, the SLP_S5# should be used to control the PS_ON# by changing the resistor to connect SLP_S5# and to disconnect SLP_S3# in Figure 3-2.

3.1.4 AT Power Delivery Block Diagram

An AT power source can provide 12 V supply to the COM-Express module, or the battery may provide a wide range of power. An additional switching regulator or LDO will be required to simulate the ATX power (5V, 3.3V, …) .There will be no standby voltage when an AT power source is used as Figure 3-2.

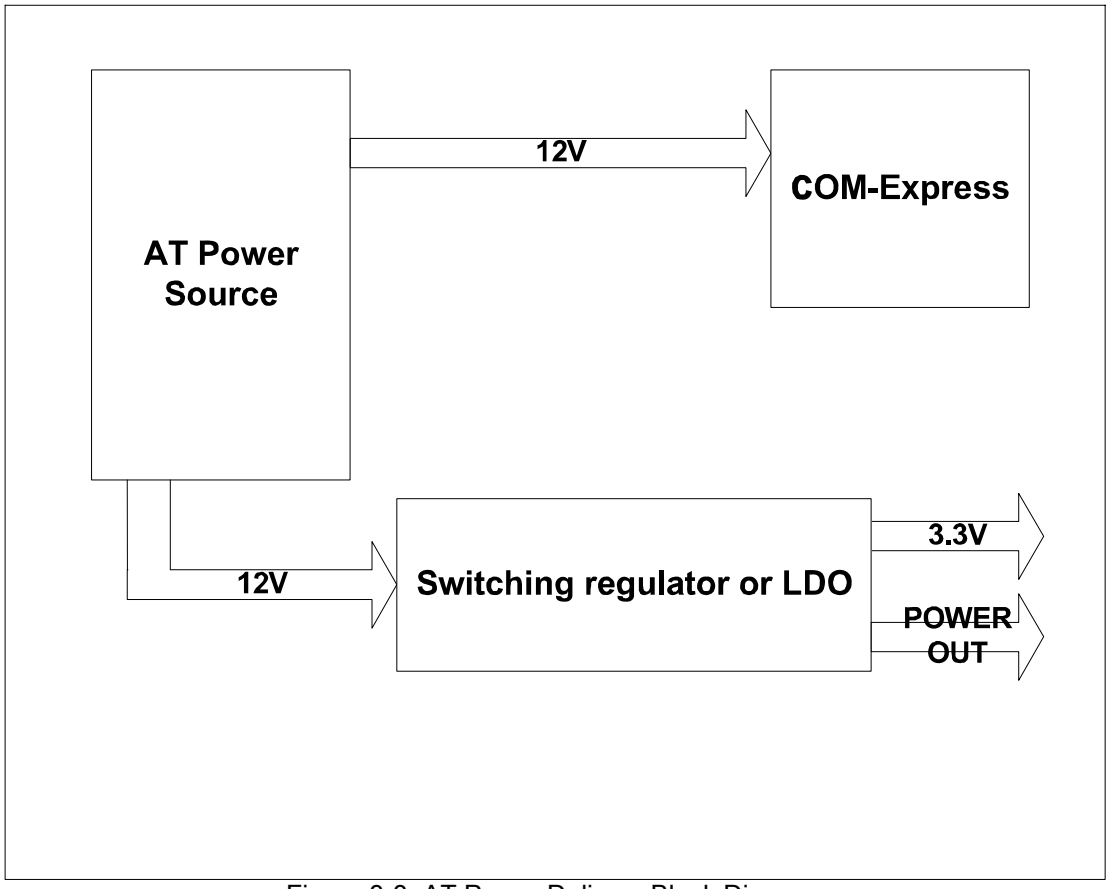


Figure 3-3 AT Power Delivery Block Diagram

3.1.5 AT Power Working as ATX Mode

An AT power source generally provides 12V but without 5VSB standby power.

If the application is working in ATX mode using AT power, there is a way to control the power management of ATX.

5V standby (5VSB) power is needed in ATX Mode. The 5VSB is sourced from the DC-input of AT power as 12V. The chipsets on COM-Express Module needs the 5VSB to enable the control mechanism for the power on/off and ACPI functions. The standby power 5VSB should be generated by carrier board power circuits from 12V AT power source.

The ATX working model can be implemented by using a switch to turn-on the AT power 12V input to the COM-E Module in S0/S1, and Turn-off the AT power input power in S3 /S4/ S5.

The schematic for the design of the ATX working model is below in Figure 3-4. Use the signal SLP_S3# to control the PMOS switch and Q1 for controlling the AT power input to the COM-Express Module. For ATX power, pressing the power button can turn-on or turn-off the system power controlled by SLP_S3#. In S3 mode, the 5VSB in COM-E Module is still supplied by the carrier board, but the AT power input 12V should be turned-off on the carrier board, without supplying the 12V to the COM-Express Module.

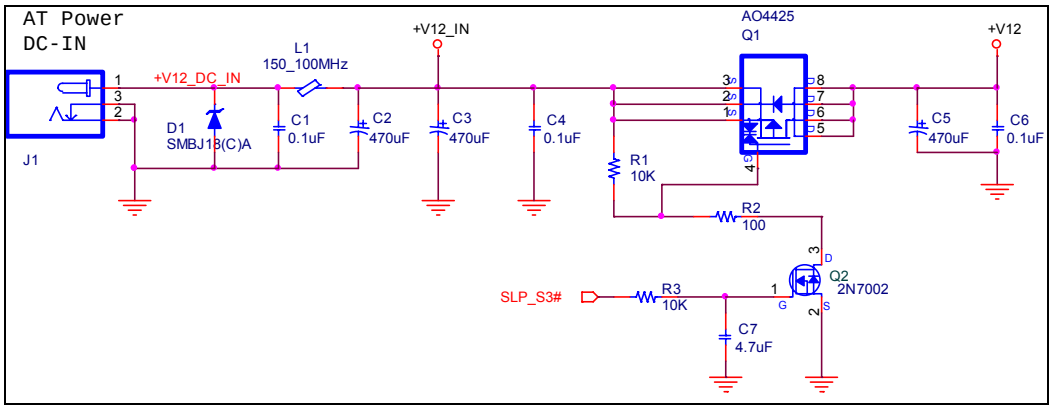


Figure 3-4 ATX Working Model Based on AT Power Input Reference Schematics

Chapter 4 PCB Layout Guidelines

A brief description of the Printed Circuit Board (PCB) for COM-Express based boards is provided in this section. From a cost- effectiveness point of view, the four-layer board is the target platform for the motherboard design. For better quality, the six-layer or 8-layer board is preferred.

4.1 Nominal Board Stack-Up

The trace impedance typically noted (55 Ω ± 10%) is the “nominal” trace impedance for a 5-mil wide external trace and a 5-mil wide internal trace. However, some stack-ups may lead to narrower or wider traces on internal or external layers in order to meet the 55-Ω impedance target, that is, the impedance of the trace when not subjected to the fields created by changing current in neighboring traces. Note the trace impedance target assumes that the trace is not subjected to the EMI fields created by changing current in neighboring traces.

It is important to consider the minimum and maximum impedance of a trace based on the switching of neighboring traces when calculating flight times. 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. In order to minimize the effects of trace-to-trace coupling, the routing guidelines in this section should be followed. Also, all high speed, impedance controlled signals should have continuous GND referenced planes and cannot be routed over or under power/GND plane splits.

4.1.1 Four-layer board stack-up

Figure 4-1 illustrates an example of a four-layer stack-up with two signal layers and two power planes. The two power planes are the power layer and the ground layer. The layer sequence as “component-ground-power-solder” is the most common stack-up arrangement from top to bottom. Table 4-1 Recommended Four-Layer Stack-Up Dimensions shows an example of general dielectric thickness with proper routing trace width / spacing for impedance control.

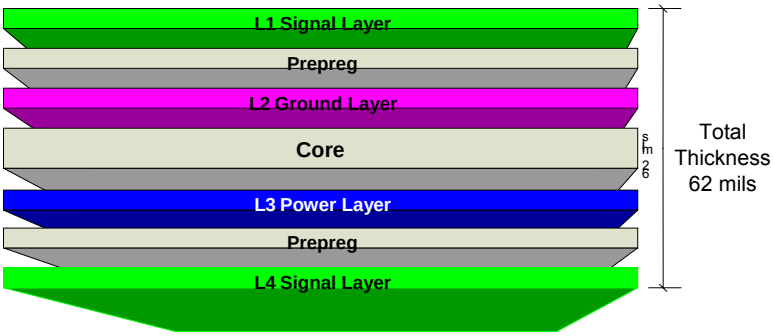


Figure 4-1 Four-Layer Stack-up

Table 4-1 Recommended Four-Layer Stack-Up Dimensions								
Layer		Dielectric Thickness (mil)	Signal-End Signals		Differential Signals		Differential Signals	
No	Type		Width/Space (mil)	Impedance (Ω)	Width/Space (mil)	Impedance (Ω)	Width/Space (mil)	Impedance (Ω)
L1	Signals	1.7	**5/5**	**55+/-10%**	**5/6**	**90+/-10%**	**5/10**	**100+/-10%**
	Prepreg	4						
L2	Ground	1.2						
	Core	47.6						
L3	Power	1.2						
	Prepreg	4						
L4	Signals	1.7	**5/5**	**55+/-10%**	**5/6**	**90+/-10%**	**5/10**	**100+/-10%**

Notes:
Target PCB Thickness: total 62 mils (0.062 inches) +/- 10%

4.1.2 Six-layer board stack-up

Figure 4-2 illustrates an example of a six-layer stack-up with 4 signal layers and 2 power planes. The two power planes are the power layer and the ground layer. The layer sequence of component-ground-IN1-IN2-power-solder is the most common stack-up arrangement from top to bottom. Table 4-2 shows an example of general dielectric thickness with proper routing trace width / spacing for impedance control.

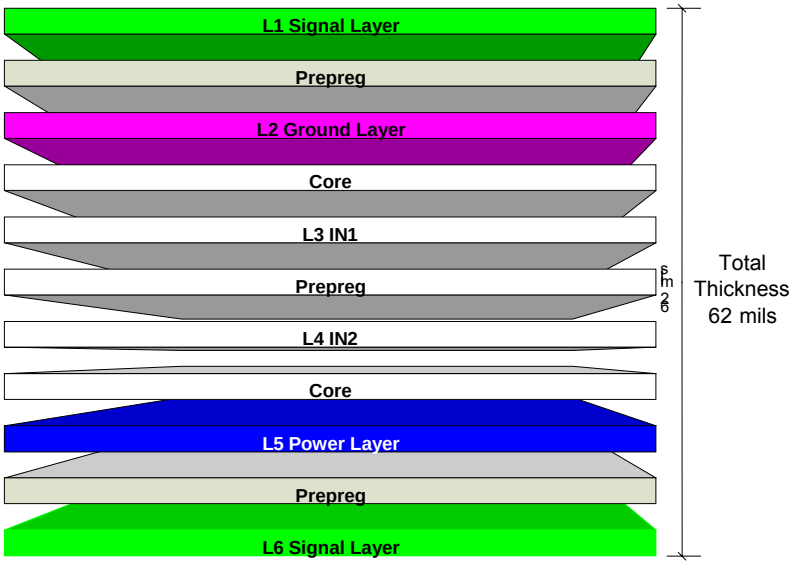


Figure 4-2 Six-Layer Stack-up

Table 4-2 Recommended Six-Layer Stack-Up Dimensions

Layer		Dielectric Thickness (mil)	Single-End Signals		Differential Signals		Differential Signals	
No	Type		Width/Space (mil)	Impedance (Ω)	Width/Space (mil)	Impedance (Ω)	Width/Space (mil)	Impedance (Ω)
L1	Signals	1.7	**5/5**	**55+/-10%**	**5/6**	**90+/-10%**	**5/10**	**100+/-10%**
	Prepreg	4						
L2	Ground	1.2						
	Core	4						
L3	IN1	1.2	**5/5**	**55+/-10%**	**5/6**	**90+/-10%**	**5/10**	**100+/-10%**
	Prepreg	38						
L4	IN2	1.2	**5/5**	**55+/-10%**	**5/6**	**90+/-10%**	**5/10**	**100+/-10%**
	Core	4						
L5	Power	1.2						
	Prepreg	4						
L6	Signals	1.7	**5/5**	**55+/-10%**	**5/6**	**90+/-10%**	**5/10**	**100+/-10%**

Notes:
The general PCB thickness: total 62 mils (0.062 inches) +/- 10%

4.2 Alternate Stack Ups

While the different stack-ups are needed (number of layers, thickness, trace width, etc.), the following key points should be noted:

1. Final post lamination, post etching, and post plating dimensions should be used for electrical model extractions.
2. All high-speed signals should reference solid ground planes through the length of their routing and should not cross plane splits. To guarantee this, both planes surrounding strip-lines should be GND.
3. High-speed signal routing should be done on internal strip-line layers. High-speed routing on external layers should be minimized to avoid EMI. Routing on external layers also introduces different delays compared to internal layers. This makes it extremely difficult to do length matching if routing is done on both internal and external layers.
4. Two layer board stack-up has poor EMI shielding and ESD protection characteristics, so it is not recommended to be designed in.

4.3 Differential Impedance Targets for Trace Routing

Table 4-3 shows the target impedance of the differential signals. The carrier board should follow the required impedance in this table.

Table 4-3 Differential Signals Impedance Requirement

Signal Type	**Impedance**
GBE LAN	100 Ω +/- 20%
LVDS	100 Ω +/- 20%
PCI Express	100 Ω +/- 20%
SATA	100 Ω +/- 20%
SDVO	100 Ω +/- 20%
USB	90 Ω +/- 20%

For the basic routing rules, please refer to the PICMG Carrier Design Guide.

Chapter 5 Mechanical Schematic Guidelines

5.1 COM-Express Mechanical Dimensions

The PCB size of the COM-Express Modules are 125mm x 95mm for COM Express Basic Modules, 95mm x 95mm for COM-Micro Modules, and 84mm x 55mm for COM-Ultra Modules. The PCB thickness is designed at 2.0mm to allow high layer stack-ups and facilitate a standard 'z' dimension between the Carrier Board and the top of the heat-spreader.

The holes shown in this drawing are intended for mounting the module / heat-spreader combination to the Carrier Board. An independent, implementation specific set of holes and spacers shall be used to attach the heat-spreader to the module. Figure 5-1 shows the COM-Express Module board mechanical dimensions. The unit is millimeters.

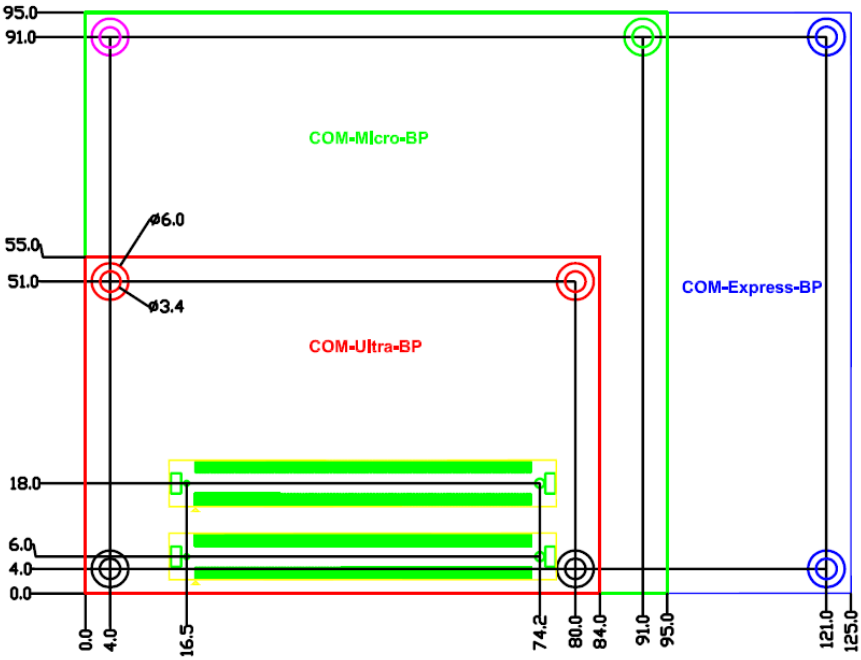


Figure 5-1 COM-Express Module Board Mechanical Dimensions

Tolerances shall be ± 0.25mm [±0.010"], unless noted otherwise. The tolerances on the module connector locating peg holes (dimensions [15.50, 6.00] and [16.50, 18.00]) shall be ± 0.10mm [±0.004"].

The 440 pin connector pair shall be mounted on the backside of the PCB and is seen "through" the board in this view.

The 5 mounting holes shown shall use 6mm diameter pads and shall have 2.7mm plated holes, for use with 2.5mm hardware. The pads shall be tied to the PCB ground plane.

5.2 COM Express Module Connector

The module connector for Pin-out Type 2 shall be a 440-pin receptacle that is composed of 2 pieces of a 220-pin, 0.5 mm pitch receptacle. The pair of connectors may be held together by a plastic carrier during assembly to allow handling by automated assembly equipment. The connectors shall be qualified for LVDS operation up to 6.25GHz, to support PCI Express Generation 2 signaling speeds.
Sources for the individual 220-pin receptacle are AMP / Tyco 3-1318490-6 0.5mm pitch Free Height 220 pin 4H Receptacle, or equivalent AMP / Tyco 8-1318490-6 0.5mm pitch Free Height 220 pin 4H Receptacle, or equivalent (same as previous part, but with anti-wicking solution applied). A source for the combined 440-pin receptacle (composed of 2 pieces of the 220 pin receptacle held by a carrier) is: AMP / Tyco 3-1827231-6 with 0.5mm pitch Free Height 440 pin 4H Receptacle or equivalent.

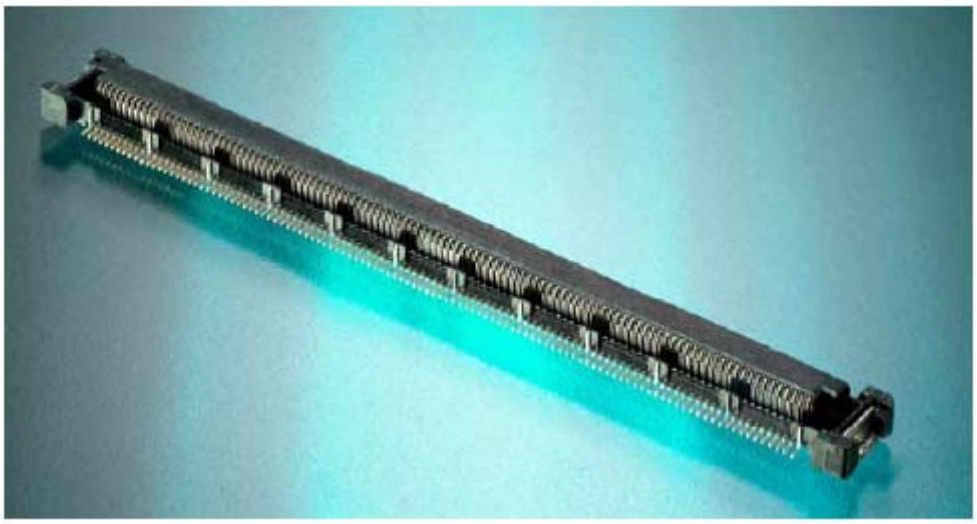


Figure 5-2 COM-Express Module Receptacle

5.3 COM Express Carrier Board Connector

The Carrier Board connector for module Pin-out Type 2 shall be a 440-pin plug that is composed of 2 pieces of a 220-pin, 0.5 mm pitch plug. The pair of connectors may be held together by a plastic carrier during assembly to allow handling by automated assembly equipment. The connectors shall be qualified for LVDS operation up to 6.25GHz, to support PCI Express Generation 2 signaling speeds. The Carrier Board plugs are available in a variety of heights. The Carrier Board shall use either the 5 mm or 8 mm heights.

A source for the individual 5mm stack height 220-pin plug is AMP / Tyco 3-1327253-6 0.5mm pitch Free Height 220 pin 5H Plug, or equivalent
A source for the combined 5mm stack height 440-pin plug (composed of 2 pieces of the 220 pin plug held by a carrier) is: AMP / Tyco 3-1827233-6 0.5mm pitch Free Height 440 pin 5H Plug, or equivalent
A source for the individual 8mm stack height 220-pin plug is AMP / Tyco 3-1318491-6 0.5mm pitch Free Height 220 pin 8H Plug, or equivalent, AMP / Tyco 8-1318491-6 0.5mm pitch Free Height 220 pin 8H Plug, or equivalent (same as previous part, but with anti-wicking solution applied)

A source for the combined 8mm stack height 440-pin plug (composed of 2 pieces of the 220 pin plug held by a carrier) is: AMP / Tyco 3-5353652-6 0.5mm pitch Free Height 440 pin 8H Plug, or equivalent.

Note: The part number above shown with a leading '8' has an anti-wicking solution applied that may help in processing with an aggressive flux. The other versions of the parts may also be made available with this solution by the vendor. The Carrier Board connector is a plug by virtue of the vendor's technical definition of a plug, and to some users it looks like a receptacle.

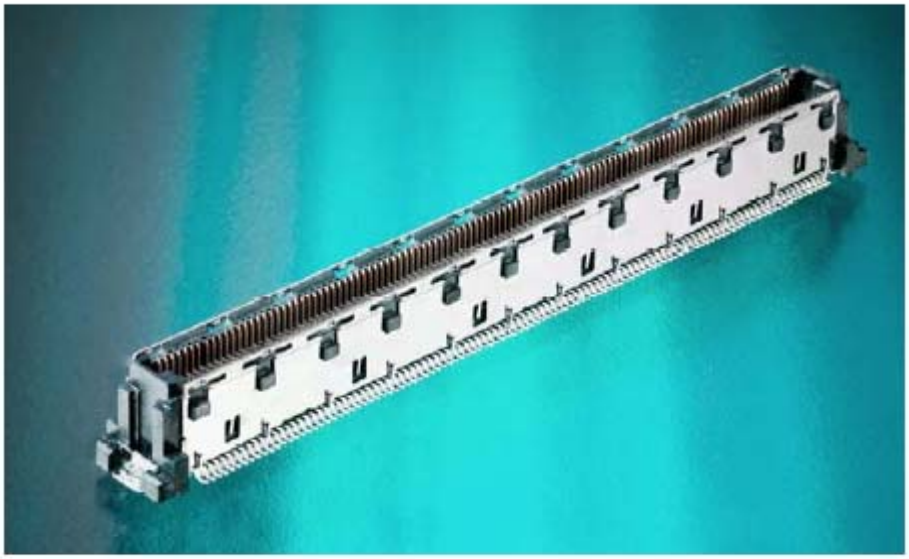
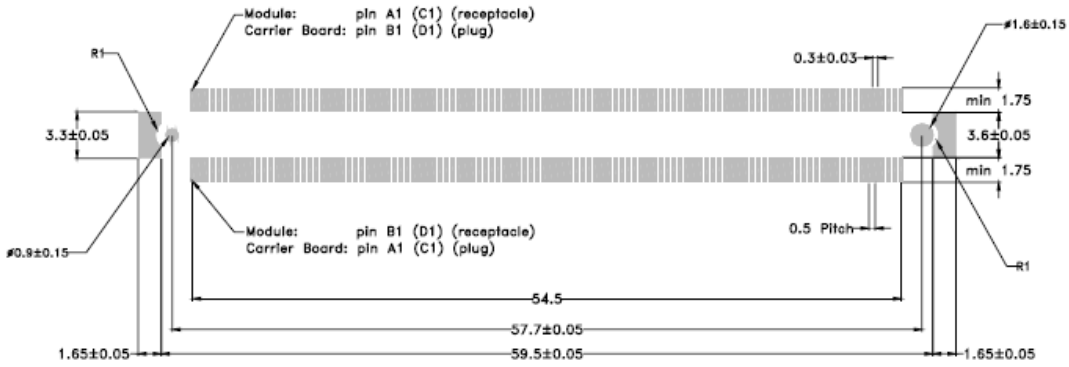


Figure 5-3 COM-Express Carrier Board Plug (8mm Version)

5.4 COM Express Connector PCB Pattern



All dimensions in mm.

Figure 5-4 COM-Express Connector PCB Pattern

To save design-in time and ensure correct connections and dimensions, Advantech provides a schematic and footprint library for COM-Express connectors. Please contact Advantech for technical support.