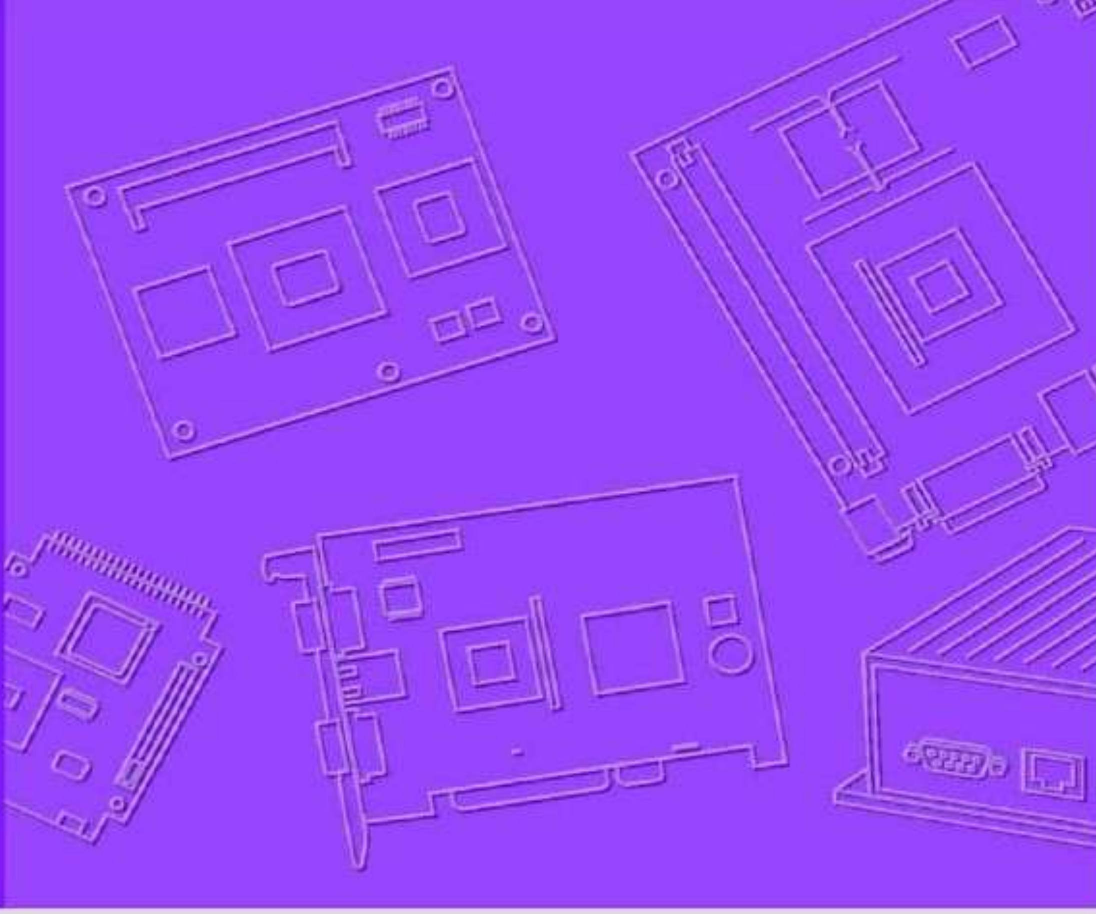


ROM-7420

RISC On Module

Carrier Board Design Guide

Version 1.0

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/Revisione	Date Dat	Description
Version 1.0	***May, 2013***	***Initial release***

Part No.	*Edition 1.0*
Printed in Taiwan	*May, 2013*

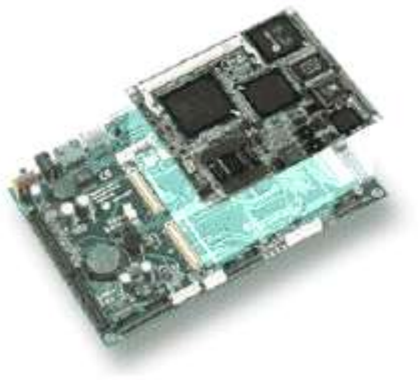
Chapter 1 Introduction

This design guide is created for ROM-7420 RISC extension module by Advantech. This document describes the key features and function block as well as the mechanical and electrical characteristics on ROM-7420 to help *users who want to develop their own carrier board with the module.*

1.1 RISC On Module (ROM) Structures

Advantech provides next generation performance of the smallest state of the art embedded modules. With a scalable solution that meets customer's advanced CPU application development needs and reduces time-to-market. Advantech's business offered helps customers develop custom CPU board solutions extremely fast and with lower investment. Using ROM-DTOS, customers can reduce traditional customized CPU board development time and costs by as much as 80%. Advantech offers a wide range of industrial standard interfaces on ROM product to cater to each customer's demands. The modular designs allow upgrade ability and add more flexibility to the system. The form factor allows the ROM module to be easily and securely mounted on a customized solution board. The design and well verified platform eliminate CPU integration worries and allow fast application support for the most dynamic embedded needs.

ROM is a series of reliable and widely used ARM's CPU cores with high integration features. ROM-7420 supports wide range of accessibility to reach the latest technical trend such like SDHC and USB 2.0. Not only does ROM allow quick design, it also provides the benefits of easy installation, maintenance and upgrade.



Though small in size, ROM takes care of most complicated CPU architectures and basic common circuits. Many system integrators are finding an Advantech ROM solution already covers 80% of their feature requirements. This makes ROM a *powerful time and money saver.*

ROM + Customer Solution Board = Your Customized Platform. RISC On Modules save time and money. Using ROM-DTOS allows customers to realize cost savings and most importantly, faster time to market, two keys to help ensure your product's *success in the market.*

1.2 Terminology

Table 1.1 Conventions and Terminology	
CODEC	***CO**MPRESSOR and **DEC**OMPRESSOR*
CPU	*Central processing Unit*
CRT	*Cathode Ray Tube*
DDR	*Double Data Rate SDRAM memory technology*
SRAM	*Static Random Access Memory*
GPMC	*General Purpose Memory Controller*
GPIO	*General Purpose Input Output*
EMI	*Electromagnetic Interference*
ESR	*Equivalent Series Resistance*
ESD	*Electrostatic Discharge*
I2C	*Inter-IC (a two wire serial bus created by Philips)*
IrDA	*Infrared Data Association*
LCD	*Liquid Crystal Display*
LVDS	Low Voltage Differential Signaling: A high speed, low power data transmission standard used for display connections to LCD panels.
NTSC	*National Television Standards Committee*
PAL	*Phase Alternate Line*
RTC	*Real Time Clock*
ROM	***R**ISC **O**n **M**odule*
CSB	*Customer Solution Board*
PCB	*Printed Circuit Board*
USB	*Universal Serial Bus*
UART	*Universal Asynchronous Receiver/Transmitter*
DAC	*Digital Analog Converter*
N/C	*Not Connected*
N/A	*Not Applied in this case*
NL/	*No Load (do not populate this part on PCB)*
SDHC	*Secure Digital High Capacity*
DSP	*Digital Signal Processing*
RISC	*Reduced Instruction Set Computing*
EEPROM	*Electrically Erasable Programmable Read Only Memory*
Kbps	*Kilo-bit per second*
SPI bus	*Serial Peripheral Interface bus*
mil	*Length unit, 1000 mils = 1 inch*

1.3 Specification of ROM-7420

Advantech's new ROM-7420 is the ultimate powerful ROM module able to drive the most demanding embedded applications requiring high performance CPU processing power & graphics support. With support for Freescale® i.MX6 processor & enhanced power-saving technology, the ROM-7420 offers developers a low power and scaleable solution that fits a moderate range of needs. The processors feature Freescale's advanced implementation of the quad ARM Cortex™-A9 core, which operates at speeds up to 1 GHz. Integrated 2D and 3D 1080p video processing multimedia accelerator strengthened by internal Video Processing Unit (VPU) up to1920 x 1080 x 30fps (1080P). The i.MX6 CPU supports up to 1GB addressing space of 533MHz DDR3 DRAM (i.e. DDR3-1066) by means of 4 Chip-Select pin architecture and 4Gbit memory size per chip-select pin. Data width of DDR3 memory could be either 32-bit or 64-bit.

ROM-7420 Main Features:

- Embedded Freescale® i.MX6 processor
- Supports 1 host USB2.0 port and 1 USB OTG port
- Supports 1 SD/MMC/SDHC interfaces with 4-bit data pin
- Supports 1 10/100/1000 Fast Ethernet port
- Supports 2 CAN BUS ports
- Supports 1 SATA-II port
- Support 1 PCI-e Gen2 x1 port
- Support 1 HDMI 1.4 output
- Support 1 VGA output
- Support 8 GPIO ports
- Integrated 4GB EMMC NAND Flash

- Be able to boot from external SD interface or from internal SPI flash memory
- Supports 2 independent ports single channel 24-bit LCD display interface up to 1080p @ 60fps data rate

Chapter 2 Pin Assignments

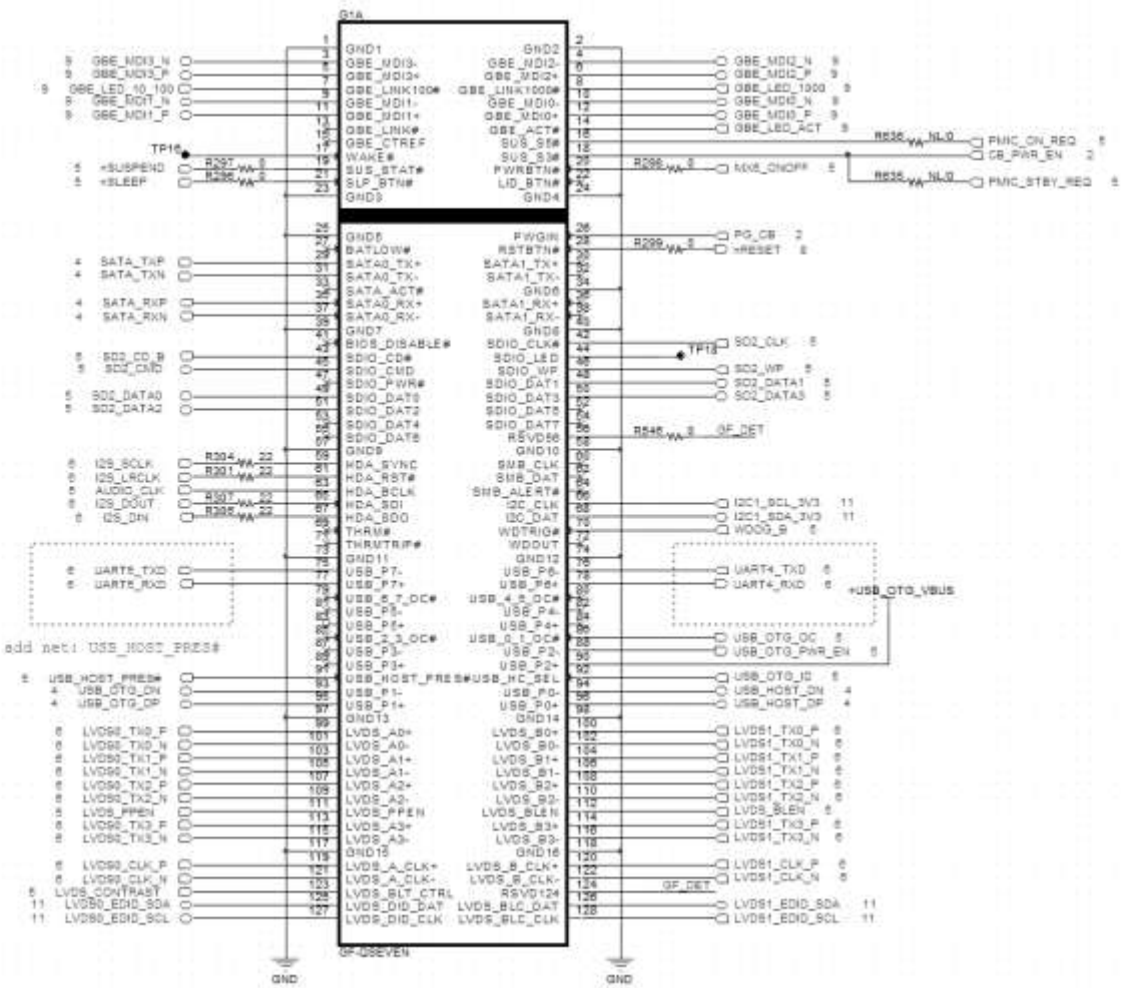
There are 115 edge fingers on the top and bottom side of the Qseven™ module that mate with the MXM connector. Table 2-1 lists the pin assignments for all 230 edge fingers.

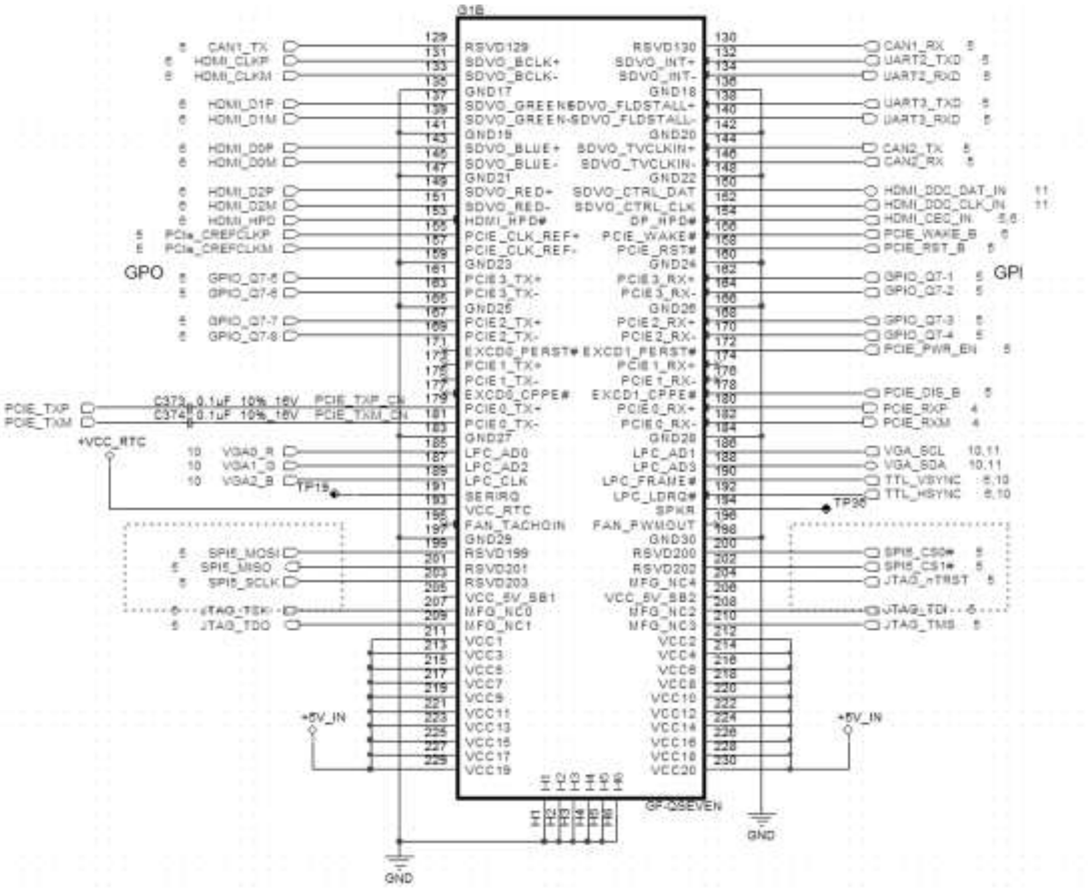
Table 2-1 Pin Assignments

Pin	Signal	Pin	Signal
1	GND	2	GND
3	GBE_MDI3-	4	GBE_MDI2-
5	GBE_MDI3+	6	GBE_MDI2+
7	GBE_LINK100#	8	GBE_LINK1000#
9	GBE_MDI1-	10	GBE_MDI0-
11	GBE_MDI1+	12	GBE_MDI0+
13	N/A	14	GBE_ACT#
15	N/A	16	N/A
17	N/A	18	CB_PWR_EN
19	N/A	20	PWRBTN#
21	SLP_BTN#	22	N/A
23	GND	24	GND
	Key		Key
25	GND	26	PWGIN
27	N/A	28	RSTBTN#
29	SATA0_TX+	30	N/A
31	SATA0_TX-	32	N/A
33	N/A	34	GND
35	SATA0_RX+	36	N/A
37	SATA0_RX-	38	N/A
39	GND	40	GND
41	N/A	42	SDIO_CLK#
43	SDIO_CD#	44	N/A
45	SDIO_CMD	46	SDIO_WP
47	N/A	48	SDIO_DAT1
49	SDIO_DAT0	50	SDIO_DAT3
51	SDIO_DAT2	52	N/A
53	N/A	54	N/A
55	N/A	56	GF_DET1
57	GND	58	GND
59	I2S_CLK	60	N/A
61	I2S_WS	62	N/A
63	I2S_MCLK	64	N/A
65	I2S_SDO	66	I2C_CLK
67	I2S_SDI	68	I2C_DAT
69	N/A	70	WDTRIG#
71	N/A	72	N/A
73	GND	74	GND

75	UART5_TX	76	UART4_TX
77	UART5_RX	78	UART4_RX
79	N/A	80	N/A
81	N/A	82	N/A
83	N/A	84	N/A
85	N/A	86	USB_0_1_OC#
87	N/A	88	USB_OTG_PEN
89	N/A	90	USB_OTG_VBUS
91	USB_CC	92	USB_ID
93	USB_P1-	94	USB_P0-
95	USB_P1+	96	USB_P0+
97	GND	98	GND
99	LVDS_A0+	100	LVDS_B0+
101	LVDS_A0-	102	LVDS_B0-
103	LVDS_A1+	104	LVDS_B1+
105	LVDS_A1-	106	LVDS_B1-
107	LVDS_A2+	108	LVDS_B2+
109	LVDS_A2-	110	LVDS_B2-
111	LVDS_PPEN	112	LVDS_BLEN
113	LVDS_A3+	114	LVDS_B3+
115	LVDS_A3-	116	LVDS_B3-
117	GND	118	GND
119	LVDS_A_CLK+	120	LVDS_B_CLK+
121	LVDS_A_CLK-	122	LVDS_B_CLK-
123	GP_PWM_OUT0	124	GF_DET2
125	GP_I2C_DAT	126	LVDS_BLC_DAT
127	GP_I2C_CLK	128	LVDS_BLC_CLK
129	CAN0_TX	130	CAN0_RX
131	TMDS_CLK+	132	UART2_TX
133	TMDS_CLK-	134	UART2_RX
135	GND	136	GND
137	TMDS_LANE1+	138	UART3_TX
139	TMDS_LANE1-	140	UART3_RX
141	GND	142	GND
143	TMDS_LANE0+	144	CAN1_TX
145	TMDS_LANE0-	146	CAN1_RX
147	GND	148	GND
149	TMDS_LANE2+	150	HDMI_CTRL_DAT
151	TMDS_LANE2-	152	HDMI_CTRL_CLK
153	HDMI_HPD#	154	N/A
155	PCIE_CLK_REF+	156	PCIE_WAKE#
157	PCIE_CLK_REF-	158	PCIE_RST#
159	GND	160	GND
161	GPIO_4	162	GPIO_0
163	GPIO_5	164	GPIO_1
165	GND	166	GND
167	GPIO_6	168	GPIO_2
169	GPIO_7	170	GPIO_3

171	N/A	172	N/A
173	N/A	174	N/A
175	N/A	176	N/A
177	N/A	178	PCIE_DIS#
179	PCIE0_TX+	180	PCIE0_RX+
181	PCIE0_TX-	182	PCIE0_RX-
183	GND	184	GND
185	VGA0_RED	186	VGA_I2C_CK
187	VGA0_GRN	188	VGA_I2C_DAT
189	VGA0_BLU	190	VGA_VSYNC
191	N/A	192	VGA_HSYNC
193	VCC_RTC	194	N/A
195	N/A	196	N/A
197	GND	198	GND
199	SPI_MOSI	200	SPI_CS0
201	SPI_MISO	202	SPI_CS1
203	SPI_SCK	204	JTAG_TRST#
205	N/A	206	N/A
207	JTAG_TCK	208	JTAG_TDI
209	JTAG_TDO	210	JTAG_TMS
211	VCC	212	VCC
213	VCC	214	VCC
215	VCC	216	VCC
217	VCC	218	VCC
219	VCC	220	VCC
221	VCC	222	VCC
223	VCC	224	VCC
225	VCC	226	VCC
227	VCC	228	VCC
229	VCC	230	VCC




Q1B
GPO
GPI
CAN1_TX
HDMI_CLKP
HDMI_CLKM
HDMI_D1P
HDMI_D1M
HDMI_D0P
HDMI_D0M
HDMI_D2P
HDMI_D2M
HDMI_HPD
PCIe_CREFCLKP
PCIe_CREFCLKM
GPIO_Q7-5
GPIO_Q7-6
GPIO_Q7-7
GPIO_Q7-8
PCIE_TXP
PCIE_TXM
C373 0.1uF 10% 16V PCIE_TXP_CN
C374 0.1uF 10% 16V PCIE_TXM_CN
+VCC_RTC
VGA0_R
VGA1_G
VGA2_B
TP19
SPI5_MOSI
SPI5_MISO
SPI5_SCLK
JTAG_TCK
JTAG_TDO
+5V_IN
GND
CAN1_RX
UART2_TXD
UART2_RXD
UART3_TXD
UART3_RXD
CAN2_TX
CAN2_RX
HDMI_DDC_DAT_IN
HDMI_DDC_CLK_IN
HDMI_CEC_IN
PCIE_WAKE_B
PCIE_RST_B
GPIO_Q7-1
GPIO_Q7-2
GPIO_Q7-3
GPIO_Q7-4
PCIE_PWR_EN
PCIE_DIS_B
PCIE_RXP
PCIE_RXM
VGA_SCL
VGA_SDA
TTL_VSYNC
TTL_HSYNC
TP36
SPI5_CS0#
SPI5_CS1#
JTAG_nTRST
JTAG_TDI
JTAG_TMS
GF-QSEVEN

Chapter 3 Carrier Board Design Guidelines

3.1 Ethernet

ROM-7420 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.

3.1.1 Signal Descriptions

Table 3-1 shows ROM-7420 Ethernet signals, including pin number, signal naming, I/O, and descriptions.

Table 3-1 GBE signals

Signal	Pin	I/O	Description
GBE_MDI[0:3]+ GBE_MDI[0:3]-	3,4,5,6, 9,10,11, 12	I/O	Media Dependent Interface (MDI) differential pair 0~3 The MDI can operate in 1000, 100, and 10Mbit/sec modes. This signal pair is used for all modes.
GBE_ACT#	14	OD	Gigabit Ethernet Controller 0 activity indicator, active low.
GBE_LINK100#	7	OD	Gigabit Ethernet Controller 0 100 Mbit / sec link indictor, active low.
GBE_LINK1000#	8	OD	Gigabit Ethernet Controller 0 1000 Mbit / sec link indictor, active low.

3.1.2 Schematic Guidelines

3.1.2.1 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 3-1 and Figure 3-2 show the 10/100M Ethernet and Gigabit Ethernet Connections.

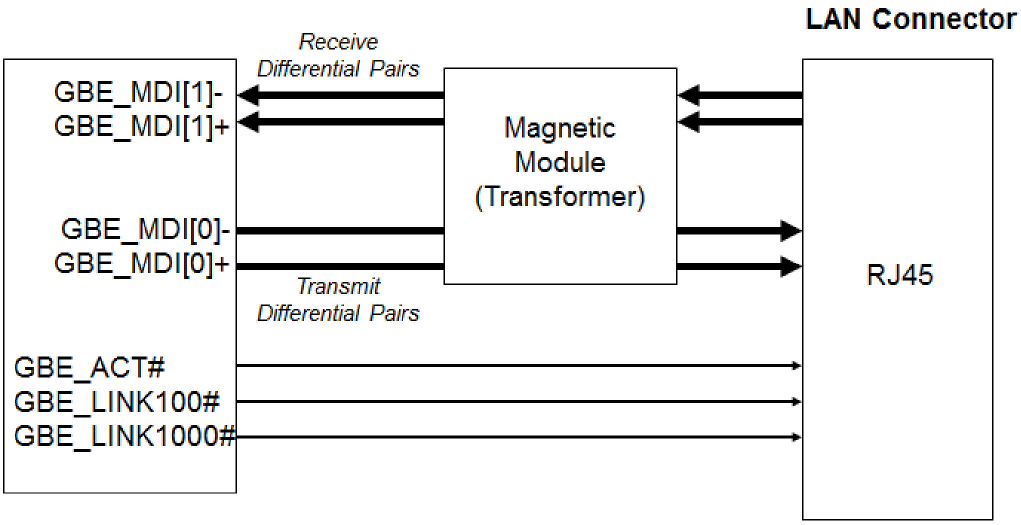


Figure 3-1 10/100M Ethernet Connections

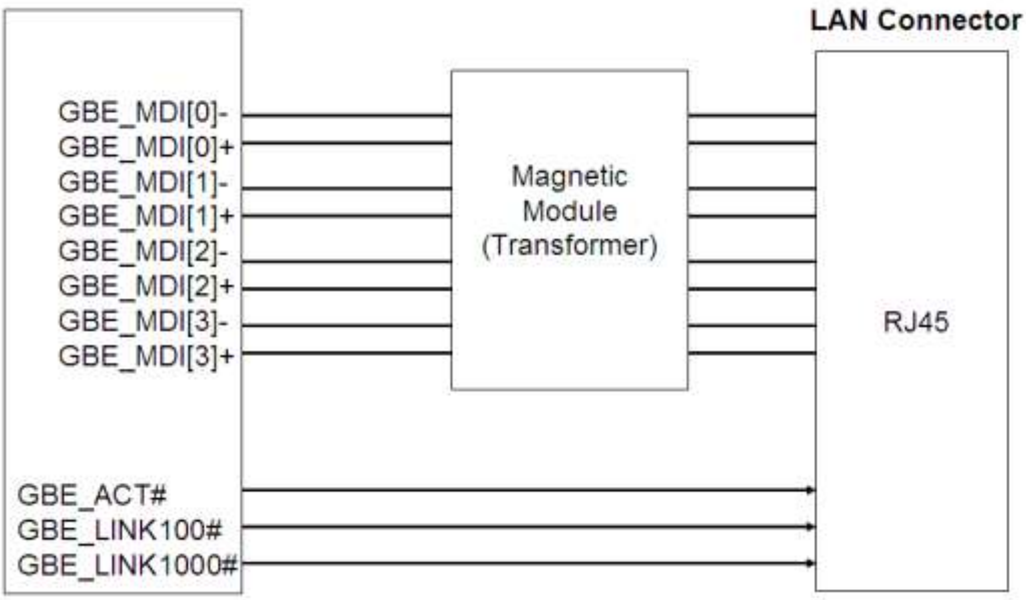


Figure 3-2 Gigabit Ethernet Connections

3.1.2.2 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 3-3 shows the integrated magnetic schematic.

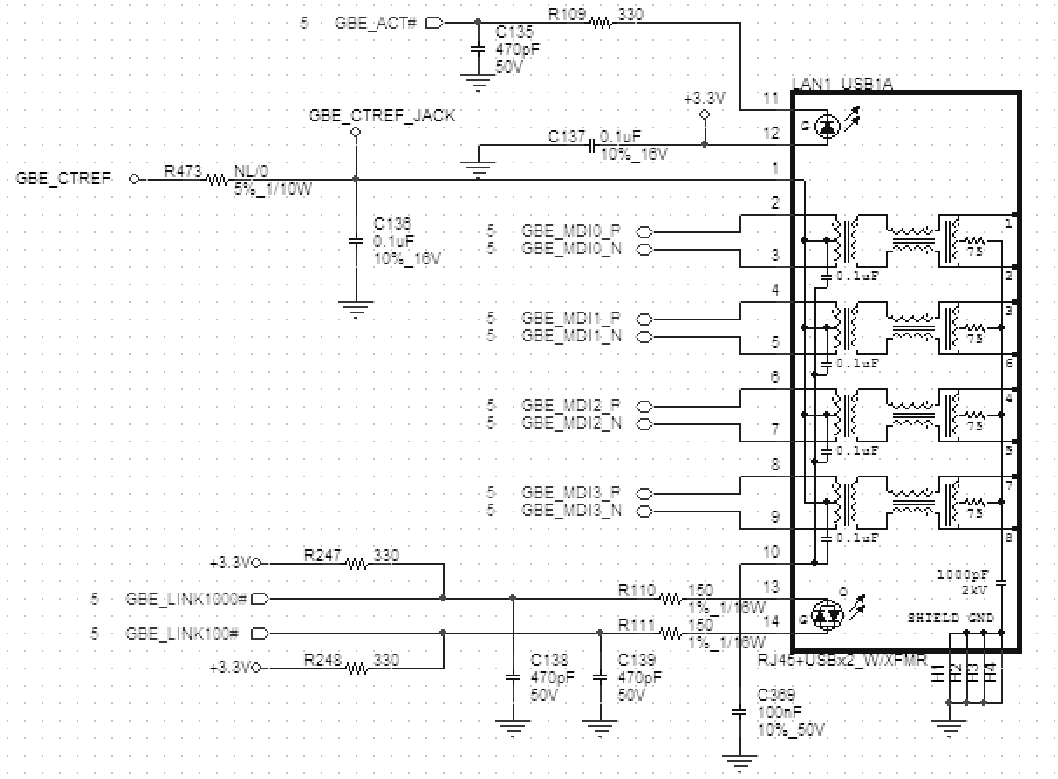


Figure 3-3 Gigabit Ethernet Connections with Integrated Magnetic

3.1.2.3 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 ROM-7420 Module’s GBE_ACT#, GBE_LINK100#, and GBE_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 330 Ω.

3.1.3 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 ROM-7420 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 ROM-7420

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.

3.1.3.1 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 6.8 inches. Keep the length of each differential pair under 6.8 inches. Figure 3-4 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. Figure 3-4 shows an example. It’s recommended to keep length L3 longer than 50 mils.

● Do not route any other signal traces (including other differential pairs) parallel to the differential traces or closer than 20 mils. Figure 3-4 shows an example. It’s recommended to keep length L3 longer than 50 mils to the differential traces.
.

● 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 3-5 shows the example.

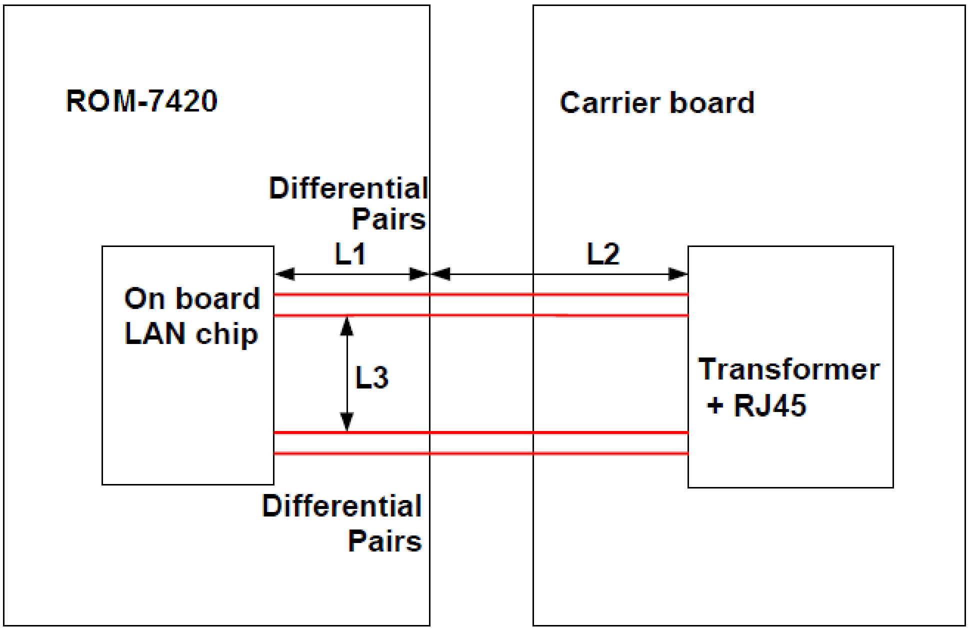


Figure 3-4 Differential signals route example

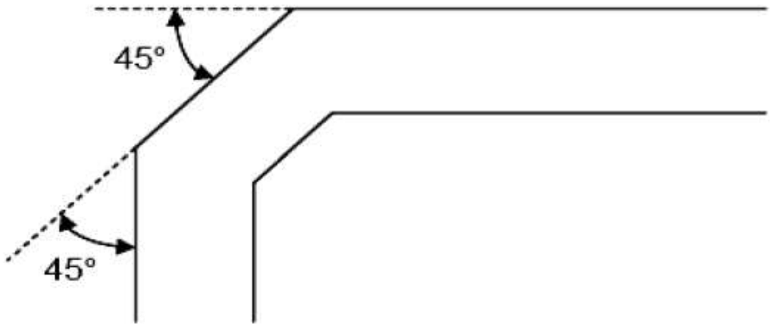


Figure 3-5 Bend layout example

3.1.3.2 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 3-6 and Figure 3-7 shows the 10/100M and Gigabit Ethernet layout with discrete magnetic.

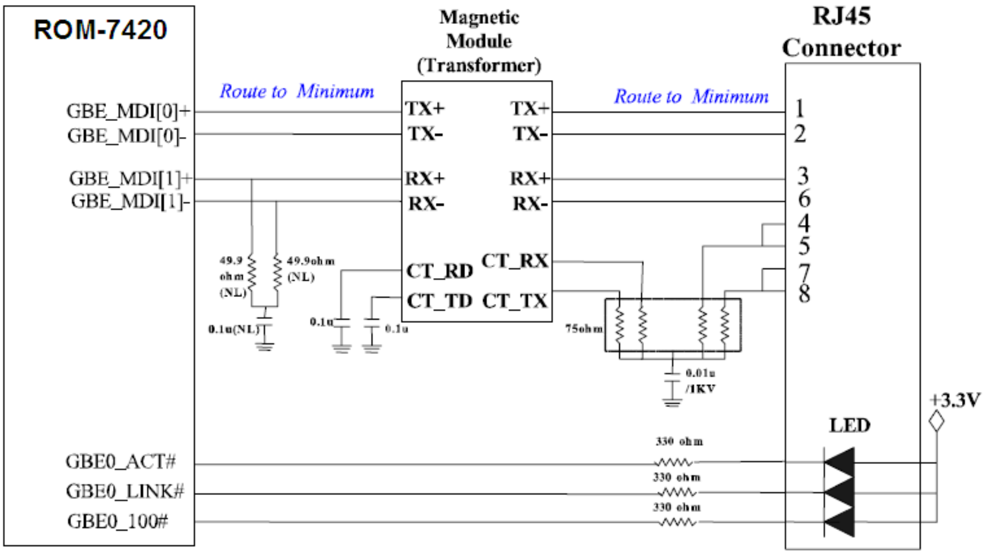


Figure 3-6 10/100M Ethernet Interconnection

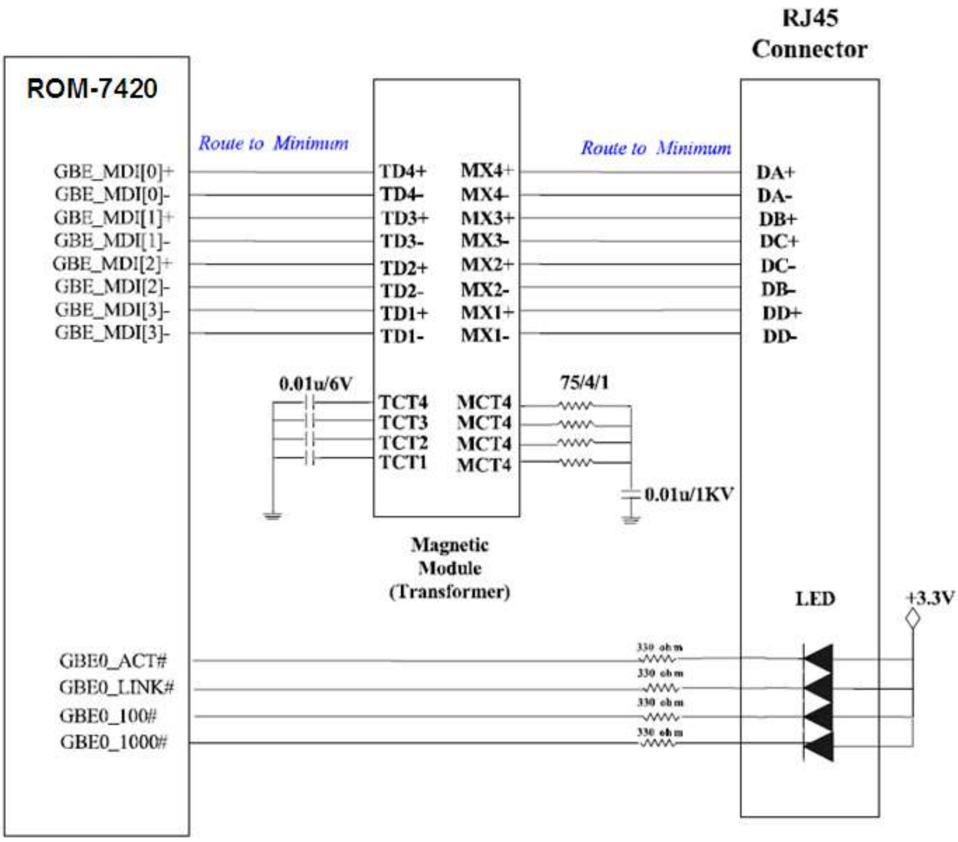


Figure 3-7 Gigabit Ethernet Interconnection

3.1.3.3 Power Considerations

In general, any section of traces that is 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.

3.1.3.4 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 3-8 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 ROM-7420 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 6.8 inches. The high-speed signals propagating through these traces require the shortest distances between these components.

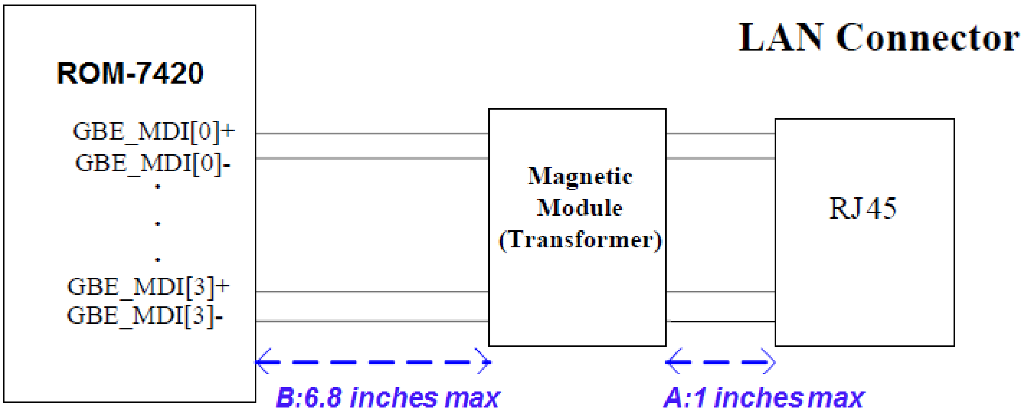


Figure 3-8 Critical Dimensions

3.2 Audio Interface

ROM-7420 module provides one Inter-IC Sound(I2S) interface, through audio CODEC IC SGTL5000 with I2S interface. Customer may establish his own application on carrier board.

3.2.1 Signal Description

Table 3.2 shows ROM-7420 module audio interface signals, including pin numbers, signals, I/O and descriptions.

Table 3-2 I2S signals

Signal	Pin	I/O	Description
I2S_CLK	59	O	I2S bit clock
I2S_WS	61	O	I2S frame clock
I2S_MCLK	63	O	System master clock
I2S_SDI	65	I	I2S data output
I2S_SDO	67	O	I2S data input

3.2.2 Schematic Guidelines

The Figure 3-9 shows the connections for ROM-7420 I2S Audio signals with Codecs.

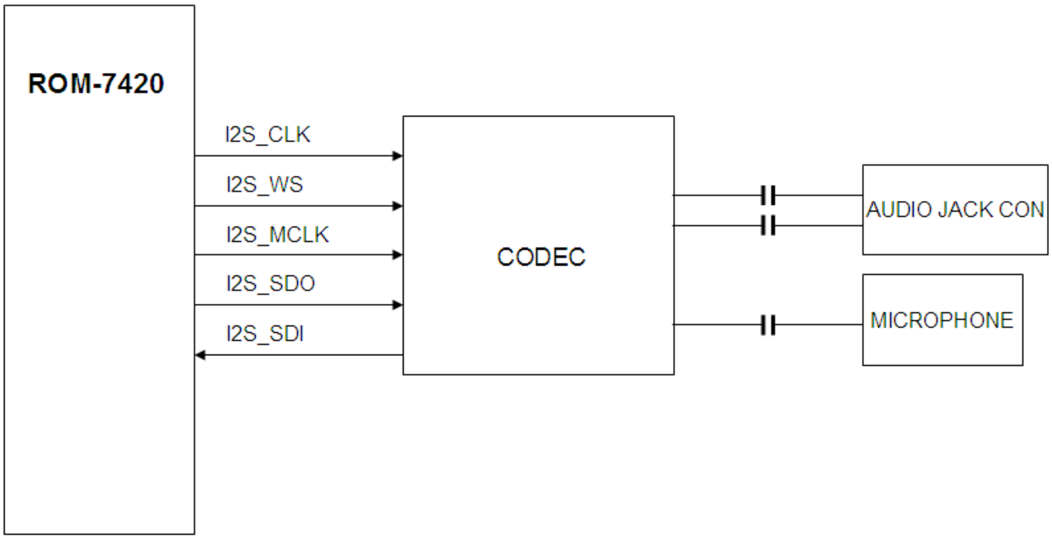


Figure 3-9 I2S Audio Connections

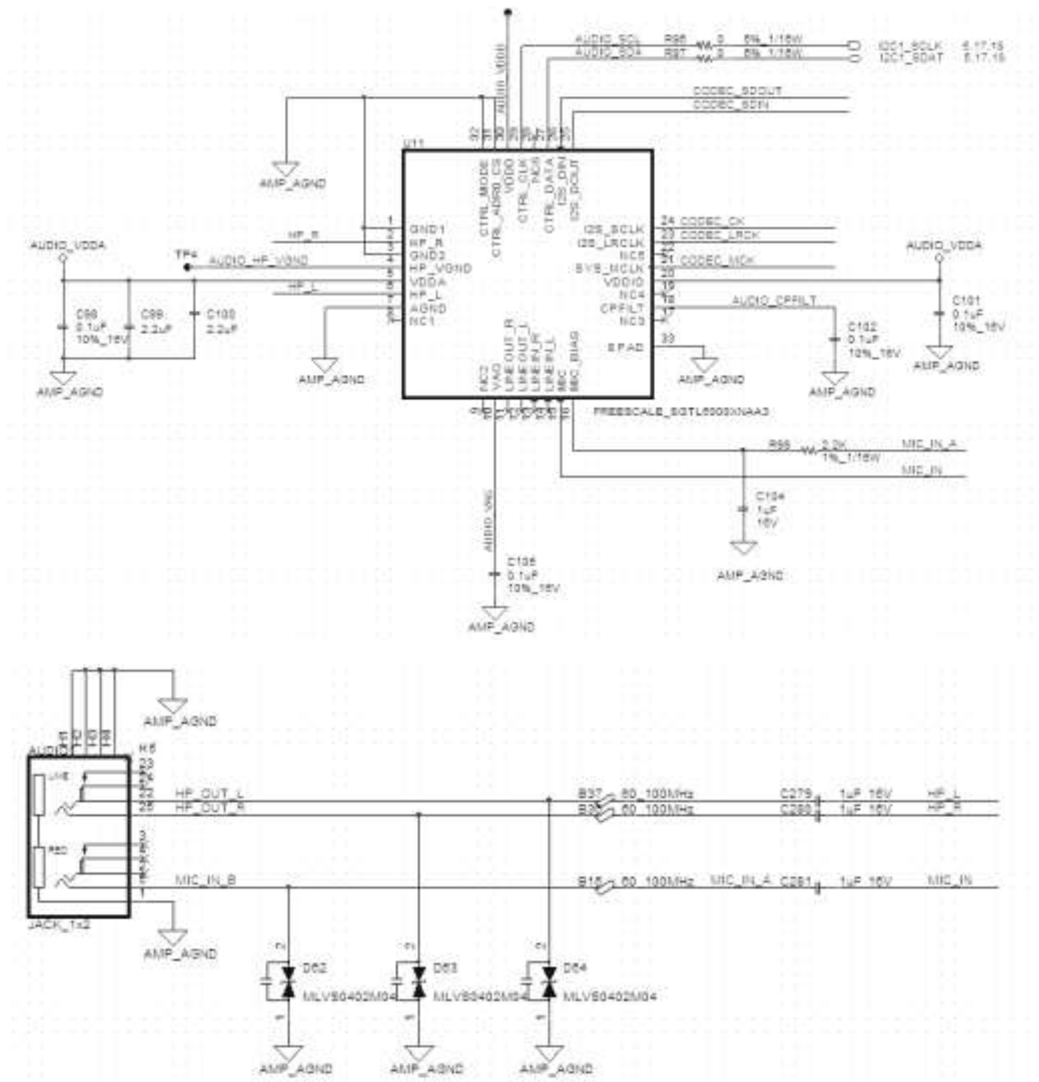


Figure 3-10 Reference Audio Schematic

3.2.3 Layout Guidelines

3.2.3.1 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 I2S Audio trace impedance from codec to ROM-7420 Module should be 50 Ω ± 10%.

3.2.3.2 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.

3.2.3.3 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 3-11 shows the grounding example for layout..

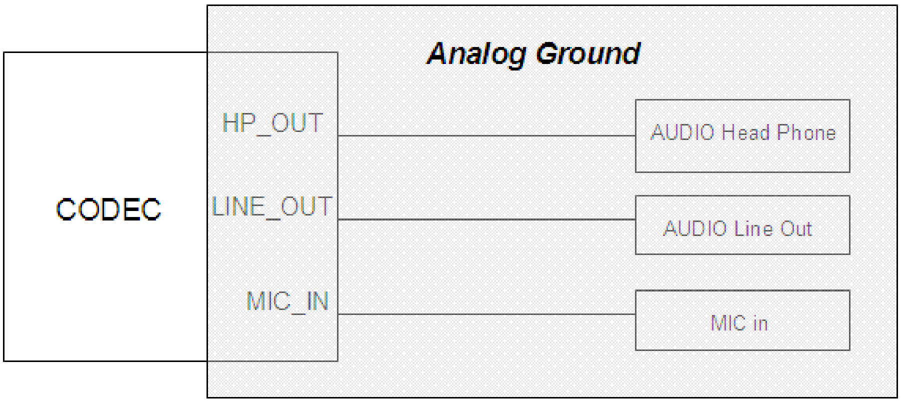


Figure 3-11 Audio Ground Guidelines

3.3 Universal Asynchronous Receiver/Transmitter (UART)

ROM-7420 provides 4 UART ports.All are 2-wire type UART signals; one port provides only TX and RX pins.

3.3.1 Signal Description

Table 3.3 shows the UART signals on the ROM-7420 module, including pin numbers, signals, I/Os and descriptions.

UART2 to 5 are in 2-wire type signal definition. All signals are in CMOS level (3.3V).

Table 3-3 UART signals

Signal	Pin	I/O	Description
UART2_TX	132	O	UART signal: Transmit
UART2_RX	134	I	UART signal: Receive
UART3_TX	138	O	UART signal: Transmit
UART3_RX	140	I	UART signal: Receive
UART4_TX	76	O	UART signal: Transmit
UART4_RX	78	I	UART signal: Receive
UART5_TX	75	O	UART signal: Transmit
UART5_RX	77	I	UART signal: Receive

3.3.2 Schematic Guidelines

Figure 3-12 gives an example of RS-232 implementation. Chosen transceiver is TI MAX3243IDB, which provides ±15kV ESD protection on transceiver side with (Human Body Model, HBM). MAX3243 operates with 3.3V or 5V power supply and designed to run at a data rate of 250kbps.

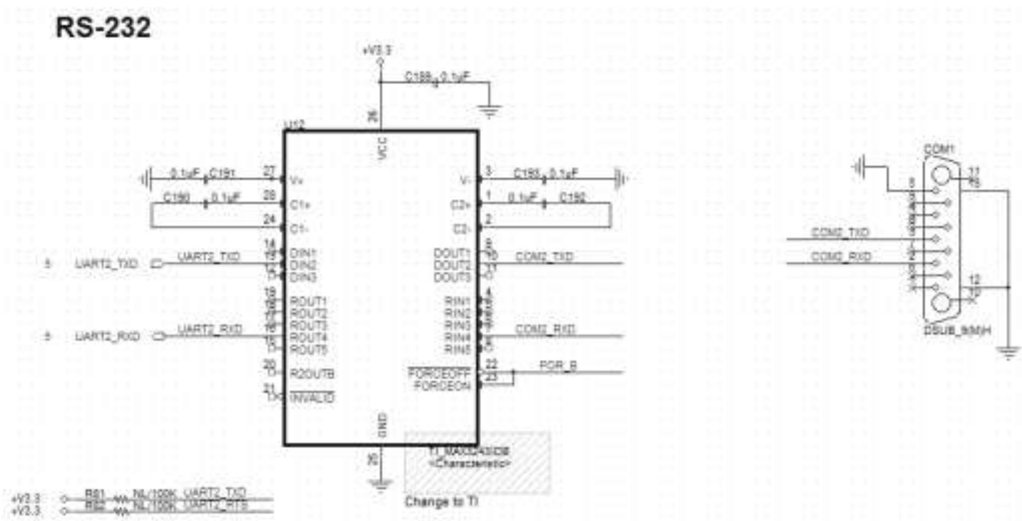


Figure 3-12 Reference Schematics of RS-232

3.3.2.1 Designing RS-232 Transceiver Level Circuits

The Electronics Industries Association (EIA) defined RS-232 standard since 1969, original standard output voltage of transmitter is +12V and -12V and input voltage only requires to distinguish +3V from -3V. The ±12V is so-called Transceiver Level, some vendors design their RS-232 transceiver only achieved the range from +5V to -5V and proclaim that it is “RS-232 compatible” devices. Using such RS-232 compatible transceivers does not precluded carrier board designers from changing device’s functionality, but if cable length is concerned in long distance communication the “normal” transceiver is recommended.

The five UART ports on ROM-7420 module support baud rate up to 115,200bps (115Kbps) on Advantech carrier board. This baud rate has been well verified by Advantech verification engineers through over-weekend loop back test. Baud rate limitation main factor is the RS-232 transceiver; Advantech chose 250kbps capability one on carrier board and customer could enhance baud rate by replacing a 1Mbps transceiver.

3.3.3 Layout Guidelines

The UART trace impedance should be 50 Ω ± 10%.

3.4 SD/MMC/SDIO Interface

ROM-7420 provides one set of SD/MMC interfaces for carrier board and each port supports following specifications:

● Full compliance with MMC command/response sets as defined in the Multimedia Card System Specification version 4.4. Including high-capacity (size > 2GBytes) cards

● Full compliance with SD command/response sets as defined in the SD Memory Card Specifications version 3.0. Including high-capacity cards SDHC up to 32GBytes.

● Fully compliant with SD Card Specification, Part A2, SD Host Controller Standard Specification, v2.00.

● Fully compliant with SDIO command/response sets and interrupt/read-wait mode as defined in the SDIO Card Specification, Part E1, v1.10

3.4.1 Signal Description

Table 3-4 SD/MMC signals			
Signal	**Pin**	**I/O**	**Description**
SDIO_CLK	42	O	SD/MMC clock signal
SDIO_CMD	45	I/O	SD/MMC command signal
SDIO_DATA0	49	I/O	SD/MMC data bit 0
SDIO_DATA1	48	I/O	SD/MMC data bit 1
SDIO_DATA2	51	I/O	SD/MMC data bit 2
SDIO_DATA3	50	I/O	SD/MMC data bit 3
SDIO_CD#	43	I/O	Card detect signal, Low Active
SDIO_WP	46	I/O	Card write protect, Hi Active

3.4.2 Design Guidelines

ROM-7420 provides one SD/MMC port which has 4-bit data width. It can be used to boot system, storage or other peripherals which are compatible with SD/MMC interface.

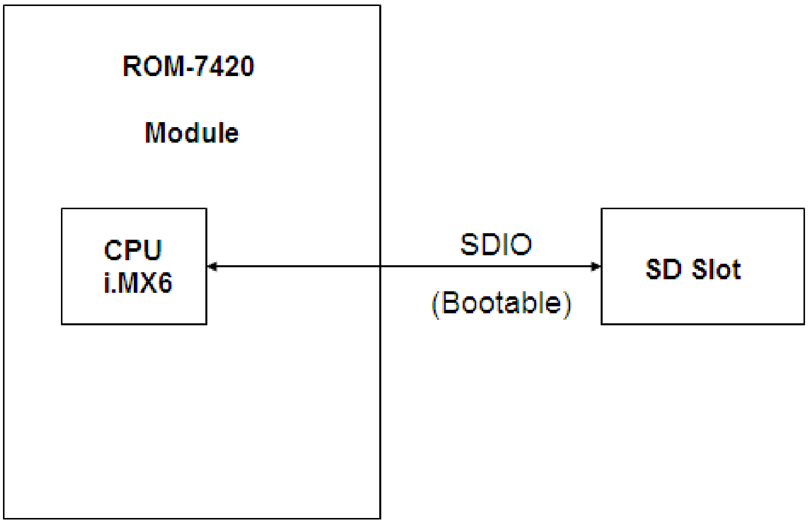


Figure 3-13 SD/MMC Interface Block Diagram

3.4.2.1 Designing SD Card Slot

SD/MMC port on ROM-7420 provide detection event for card insertion.
User could use SDIO_CD# to implement the detection event, those signals are low-active and Module has internal Pull-Up 10KΩ to 3.3V.

3.4.2.2 SD Slot Implementation

Figure 3-14 is an example of placing a SD card slot on carrier board. Q15 is a P channel MOS-FET which supplies power to SD slot after SD card inserted into it. This design is mainly to cut off supply power for card slot while SD card is not inserted into slot.

User can choose an appropriate way to power the SD card. Please note that the power supply sequencing of SD/MMC device must be considered if user requires the booting from SD flash. The timing of supplying power must be provided in the same time with SYS_3v3 supplied to module.

That's because the module will try to access SD device to boot after power-on reset de-asserted; therefore, the SD device need be powered in time before de-asserting power-on reset event on the module.

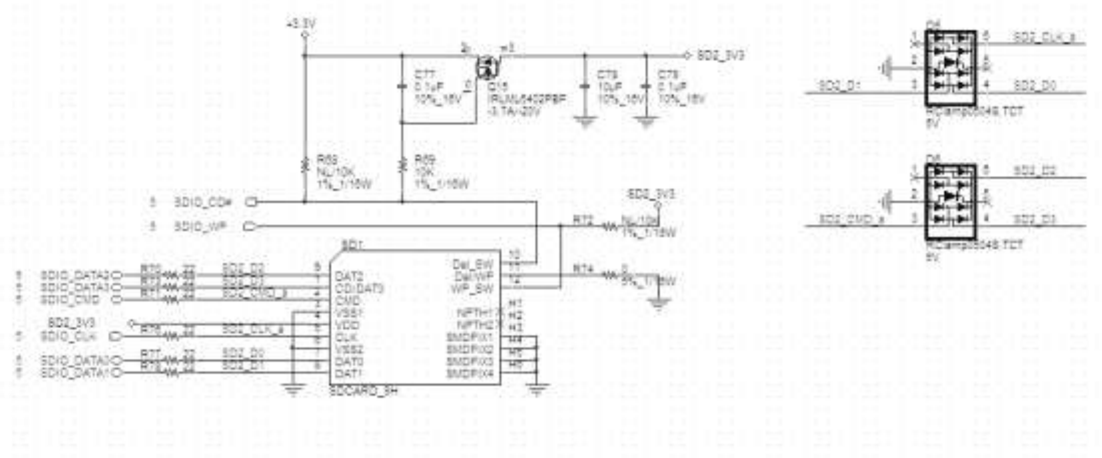


Figure 3-14 SD/MMC Reference Schematics

3.4.3 Layout Guidelines

Table 3-5 lists guidelines of SD/MMC signals in PCB routing stage, note that trace width and spacing of single-ended signals differ from PCB stack and copper thickness; even differ from each layer inside one PCB.

Table 3-5

Parameter	**Trace Routing**
Maximum trace length allowance for all signals on carrier board	5 inches
Single-ended impedance	50Ω ±10%
Length mismatch to clock SD/MMC group	All trace length mismatch ≦ 200mil

Table 3.5 SD/MMC Trace Routing Guidelines

3.5 LVDS Interface

3.5.1 Signal Description

Table 3-6 shows the ROM-7420 LVDS signals.

Table 3-6 LVDS signals

Signal	Pin	I/O	Description
LVDS_A [0~3]+ LVDS_A [0~3]-	99, 101, 103, 105,107,109, 113,115	O	LVDS Primary Channel differential pairs
LVDS_A_CLK+ LVDS_A_CLK-	119, 121	O	LVDS Primary Channel differential clock
GP_I2C_DAT	125	I/O	I2C data output for LVDS Primary Channel display use
GP_I2C_CLK	127	I/O	I2C clock output for LVDS Primary Channel display use
LVDS_B [0~3]+ LVDS_B [0~3]-	100,102,104, 106,108,110, 114,116	O	LVDS Secondary Channel differential pairs
LVDS_B_CLK+ LVDS_B_CLK-	120, 122	O	LVDS Secondary Channel differential clock
LVDS_BLC_DAT	126	I/O	I2C data output for LVDS Secondary Channel display use
LVDS_BLC_CLK	128	I/O	I2C clock output for LVDS Secondary Channel display use
LVDS_PPEN	111	O	LVDS panel power enable
LVDS_BLEN	112	O	LVDS panel backlight enable
GP_PWM_OUT0	123	O	LVDS panel backlight brightness control

3.5.2 Schematic Guidelines

The LVDS signals can be routed directly from the ROM-7420 module to the LVDS connectors.Figure 3-15 shows one pair of LVDS connections. Each pair can use the common-mode choke for EMI compliance if needed.

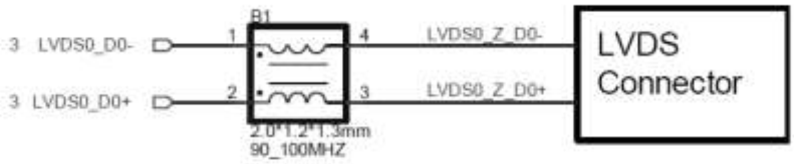


Figure 3-15 One LVDS Differential Pair with Choke Design

3.5.3 Layout Requirements

The timing skew minimization requires trace length matching between chipset diepad to the pins of the LVDS connector. Match the package length difference between each signal group to minimize the timing variance. The ROM-7420 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 3-7 shows the LVDS Signals Trace Length Mismatch Mapping.

Table 3-7 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_A0+ LVDS_A0-	±5 mils	LVDS_A_CLK+ LVDS_A_CLK-	±5 mils	±100 mil
	LVDS_A1+ LVDS_A1-	±5 mils			
	LVDS_A2+ LVDS_A2-	±5 mils			
	LVDS_A3+ LVDS_A3-	±5 mils			
CHANNEL B	LVDS_B0+ LVDS_B0-	±5 mils	LVDS_B_CLK+ LVDS_B_CLK-	±5 mils	±100 mil
	LVDS_B1+ LVDS_B1-	±5 mils			
	LVDS_B2+ LVDS_B2-	±5 mils			
	LVDS_B3+ LVDS_B3-	±5 mils			

Each LVDS signal should be trace length matched to its associated clock strobe within ±100 mils.

Routing for LVDS transmitter signals of different traces are terminated across 100 Ω ± 10% and should be routed as following points:

● It is necessary to maintain the differential impedance, Zdiff = 100 Ω ± 10%, 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 6.8 inches. Please refer to Advantech layout checklist for detailed info.
● Clocks must be matched to the associated data signals to within ±100 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 95Ω. The cable length should be less than 0.5 meter for better signal quality.

3.6 VGA

ROM-7420 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.

3.6.1 Signal Description

Table 3-8 shows ROM-7420 VGA signals

Table 3-8 VGA signals

Signal	Pin	I/O	Description
VGA_RED	185	O	Red analog video output signal for CRT monitors, designed to drive a 75 Ω equivalent load.
VGA_GRN	187	O	Green analog video output signal for CRT monitors, designed to drive a 75 Ω equivalent load.
VGA_BLU	189	O	Blue analog video output signal for CRT monitors, designed to drive a 75 Ω equivalent load.
VGA_VSYNC	190	O	Horizontal Sync: This output supplies the horizontal synchronization pulse to the CRT monitor.
VGA_HSYNC	192	O	Vertical Sync: This output supplies the vertical synchronization pulse to the CRT monitor.
VGA_I2C_CK	186	O	DDC clock line. It can be used for a DDC interface between the graphics controller chip and the CRT monitor
VGA_I2C_DAT	188	I/O	DDC data line. It can be used for a DDC interface between the graphics controller chip and the CRT monitor

3.6.2 VGA Specifications

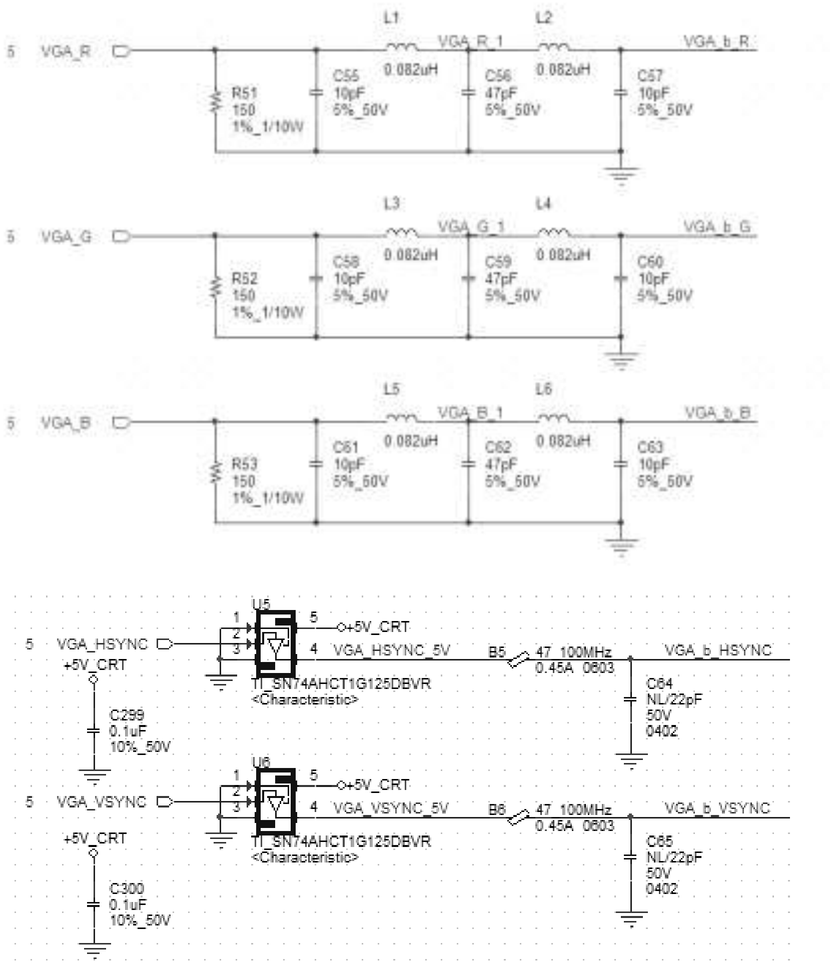
Please refer to “VESA and Industry Standards and Guidelines for Computer Display

Monitor Timing Version 1.0, Revision 0.8" for the monitor timing specification.

3.6.3 Schematic Guidelines

The reference schematic of VGA is shown in FIigure 3-16.

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, the DDC pulled-up voltage is 3.3V on module board , DO NOT pull up DDC signal on carrier board, you can through level shift circuit to change pull up voltage from 3.3V to 5V. ESD protection voltage could be 3.3V or 5V depends on the power map of the carrier board



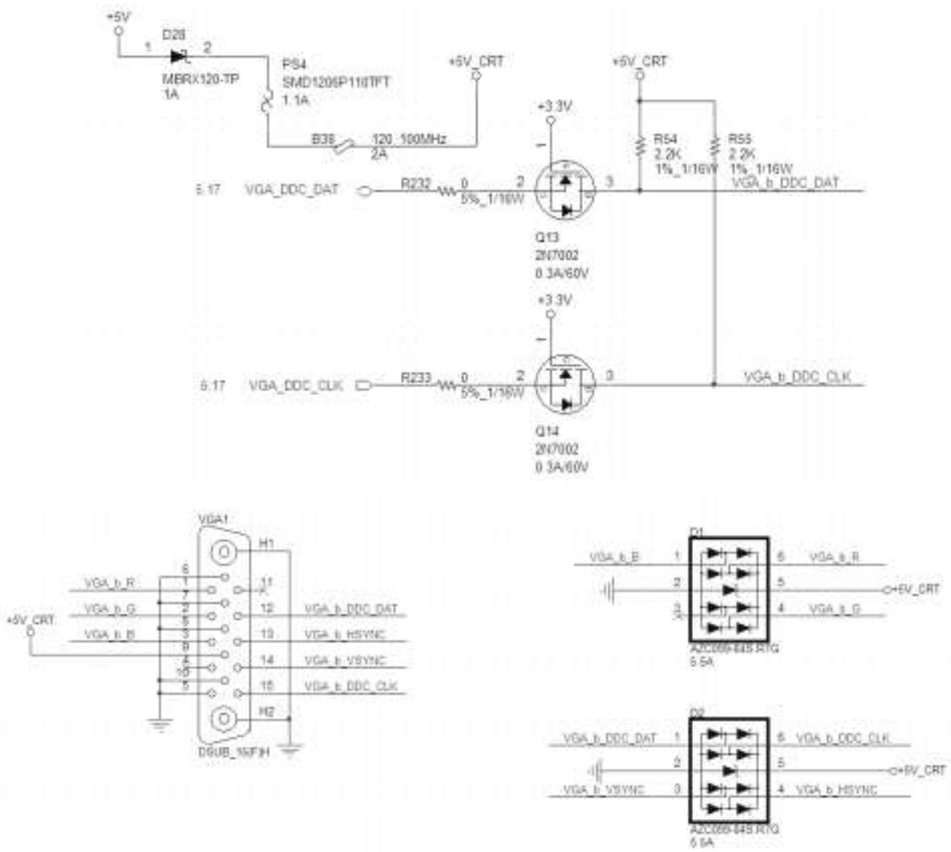


Figure 3-16 VGA reference schematic

3.6.3 Layout Guideline

3.6.3.1 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 bead 47Ω@100MHz series for HSYNC and VSYNC 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 ROM-7420 connector. And the routing section between 150Ω resistor and VGA connector should be kept as shPort as possible with proper trace width for 75Ω impedance.

3.7 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. ROM-7420 Modules provide one USB 2.0 port and one OTG port

3.7.1 Signal Description

Table 3-9 shows ROM-7420 USB signals, including pin number, signals, I/O and descriptions.

Table 3-9 USB Signals Description			
Signal	**Pin**	**I/O**	**Description**
USB_P0- USB_P0+	94, 96	I/O	Universal Serial Bus Port 0 differential pair.
USB_P1- USB_P1+	93, 95	I/O	Universal Serial Bus Port 1 differential pair. This port may be optionally used as USB client port.
USB_ID	92	I	USB Host control select pin. 1: USB Port 1 as USB Client and enable USB Client support 0: USB Port 1 as USB Host mode. N/C if OTG port not used
USB_OTG_PEN	88	O	Control the power source supplied to USB peripherals 500mA @5V in USB Host Mode. Active high.
USB_0_1_OC#	86	I	Over current indicator from carrier board to Module board. This pin is used to monitor the USB power over current of the USB Ports 0 and 1.
USB_CC	91	I	USB client present detect pin. The USB OTG port may operate as USB client or USB host.If USB OTG port is configured as client port then this pin indicates that an external USB host is connected to USB OTG port.
USB_VBUS	90	P	USB VBUS pin. 5V tolerant. VBUS resistance has to be placed on the module. VBUS capacitance has to be placed on the carrier board

3.7.2 USB Spec.

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

3.7.3 Schematic Guidelines

3.7.3.1 Reference Schematics of USB Host Port

The Figure 3-17 shows the USB connections for ROM-7420 USB signals. The ESD are recommended and the capacitors are reserved for EMI compliance which are usually not loaded.

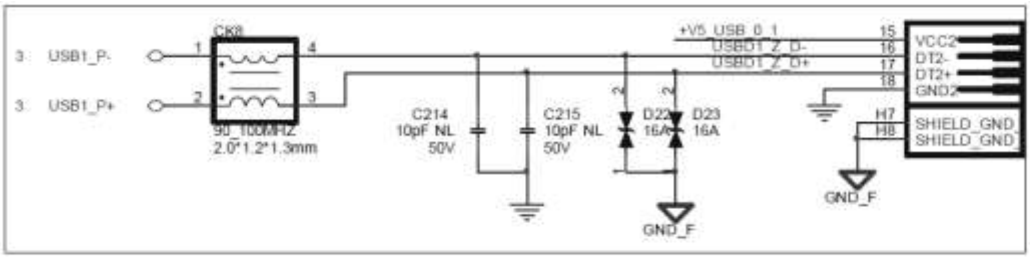


Figure 3-17 USB Host reference schematics

3.7.3.2 Reference Schematics of USB OTG Port

Figure 3-18 shows USB Port 1 design in OTG mode application, signal "USB_ID" is left pull-up and default be set to USB client mode. Signal "USB_P1+" and "USB_P1-" on this figure are connected to ROM-7420 and common mode choke is also selected 90Ω@100MHz one just the same as choke used in USB hosts.

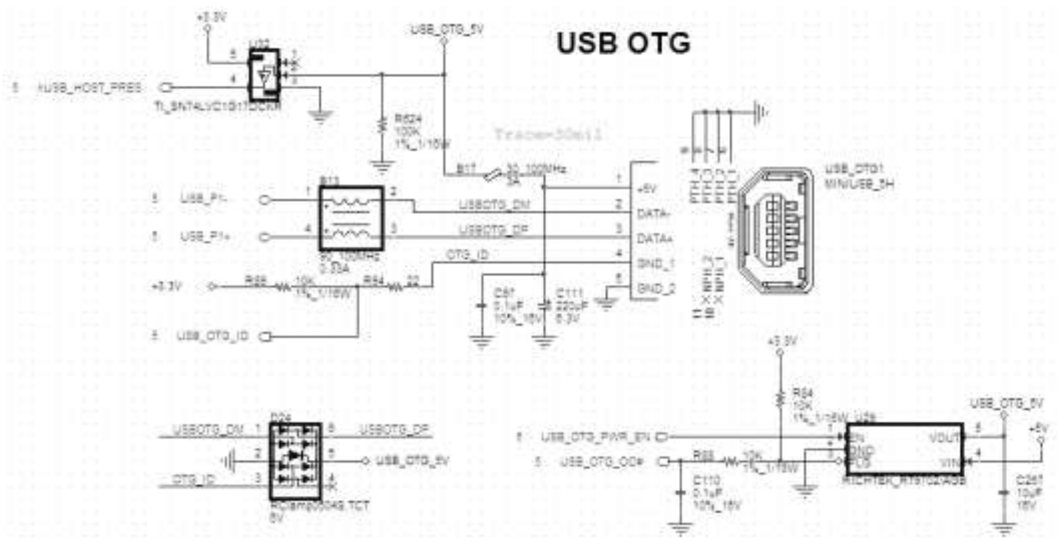


Figure 3-18 USB OTG reference schematics

3.7.3.3 How to configure USB OTG port into USB host

The USB port 1 marked as "USB OTG" differential signal pairs on pin number 93 and 95 is able to be configured to USB host function.

The way to change USB client mode into host mode needs to modify three items.

● Make signal 'USB_ID ' to low voltage state in hardware side (either by special cable or by strong pull low resistor). This makes USB OTG function configured to host mode.

● Design power rail supplied to device, standard current is 500mA @ 5V per USB port.

● Modify software program to become USB OTG compatible.

Note that the pin “USB_ID” is pull-up on ROM-7420 module, so this function would be configured to client mode. If USB OTG port is not used on carrier board, it is suggested to left this pin floating.

3.7.3.4 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.

3.7.3.5 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 3-19 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.

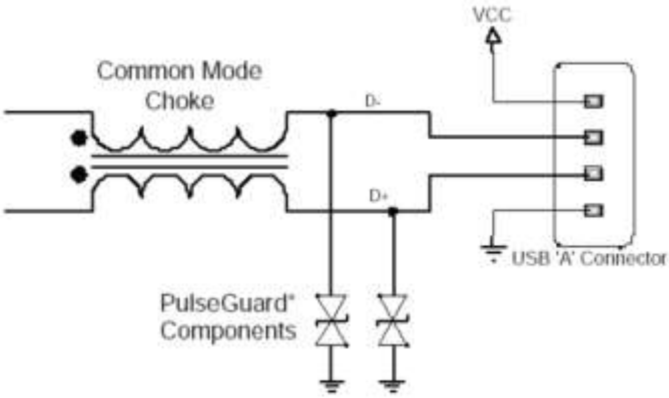


Figure 3-19 Common Mode Choke

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

3.7.3.6 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 USB_0_1_OC# 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

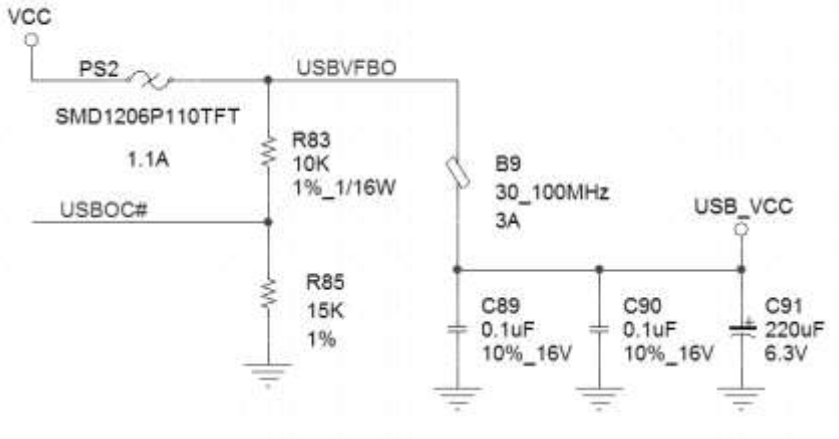


Figure 3-20 Overcurrent Circuit

The poly-switch in Figure 3-20 generally could not switch off fast enough. Overcurrent 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 3-21 are recommended. These devices may be used for per port protection of the USB power lines and direct connected to the USB_0_1_OC# signal.

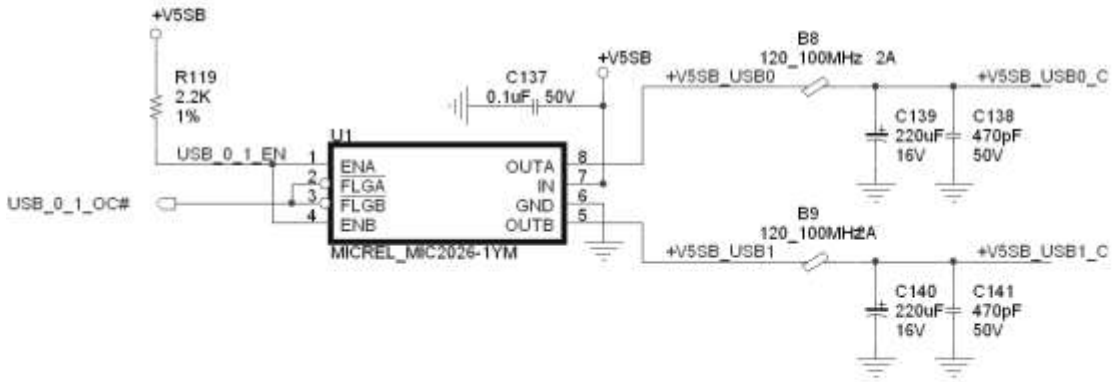


Figure 3-21 Power Switch with Overcurrent Protection Circuits

The over-current protection circuit is not implemented on the ROM-7420 Module.
It should be implemented on the carrier board.

3.7.4 Layout Guideline

3.7.4.1 Differential pairs

The USB data pairs (ex. P0- and P0+) should be routed on the carrier board as parallel differential pairs, with a differential impedance of 90 Ω. PCB layout software usually allows determining the correct trace width and spacing to achieve this impedance after the PCB stack-up configuration is known. As per usual differential pair routing practices, the two traces of each USB pair should be matched in length and kept at uniform spacing. Sharp corners should be avoided and be replaced by two 45° bends. Loop areas should be minimized and USB data pairs should be routed as far from other signals as possible.

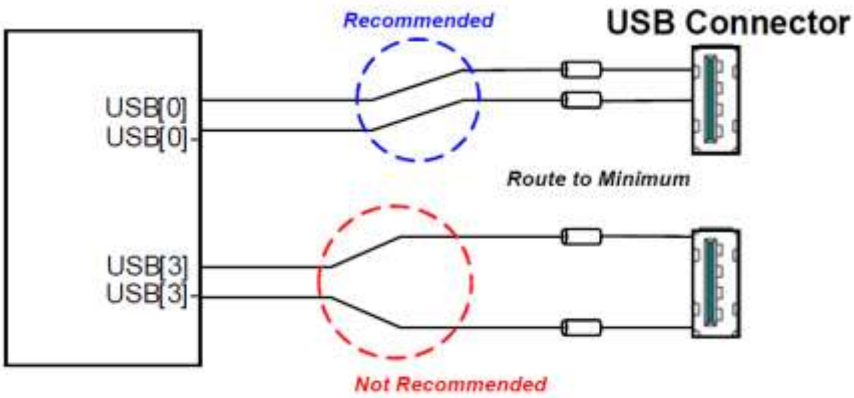


Figure 3-22 USB Layout Guidelines

3.7.4.2 Crossing a plane split

The mistake shown here is where the data lines cross a plane split. This causes unpredictable return path currents and would likely cause a signal quality failure as well as creating EMI problems.

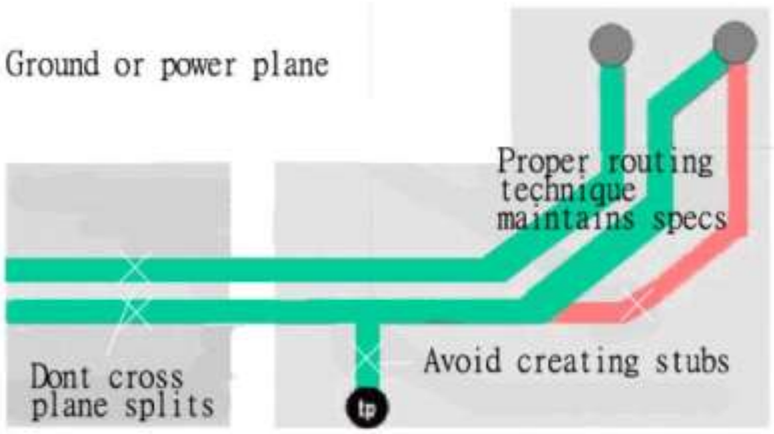


Figure 3-23 Violation of Proper Routing Techniques

3.8 Serial ATA

ROM-7420 Module provides up to four Serial ATA (SATA) interface, depending on the chipset specs of the module.

3.8.1 Signal Description

Table 3-10 shows ROM-7420 SATA signals, including pin number, signals, I/O and descriptions.

Table 3-10 SATA Signals Description

Signal	Pin	I/O	Description
SATA0_TX+ SATA0_TX-	29, 31	O	Serial ATA channel 0, Transmit Output differential pair.
SATA0_RX+ SATA0_RX-	35, 37	I	Serial ATA channel 0, Receive Input differential pair.

3.8.2 Schematic Guidelines

3.8.2.1 Serial ATA AC Coupling Requirements

Both of TX and RX SATA differential pairs require AC coupling capacitors.All AC coupling capacitorson the transmitter (TX) and receiver (RX) are placed on the ROM-7420 Module. Do not place the AC coupling capacitors on the carrier board. Figure 3-24 and Figure 3-25 show the connections.

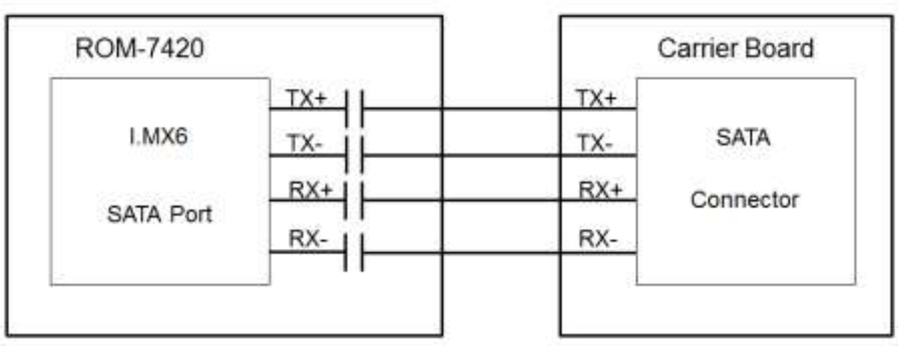


Figure 3-24 SATA interconnection example

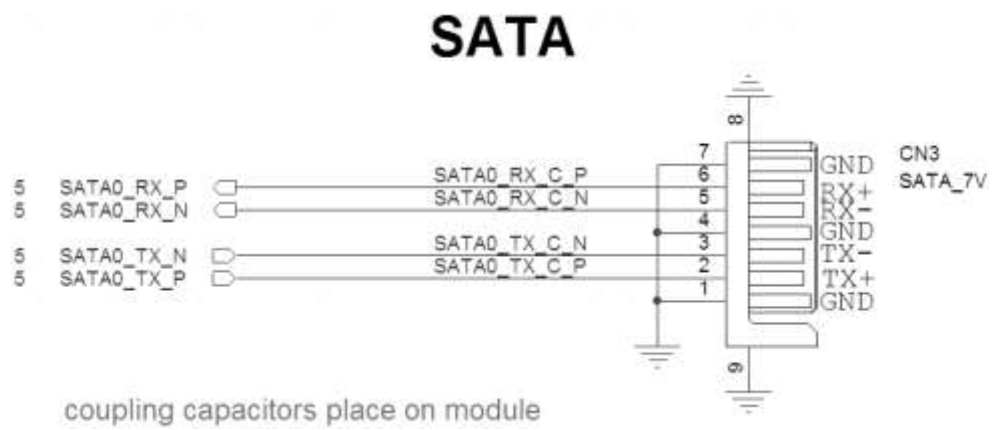


Figure 3-25 SATA Schematic Reference

3.8.3 Layout Guidelines

3.8.3.1 General routing and placement

● Place the SATA connectors as closed as possible to the ROM-7420 Module. The routing length total is recommended to be not more than 10 inches for SATA Gen1 and 5 inches for SATA Gen2. 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 ± 10%. 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 ± 10%.

3.8.3.2 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 total of SATA signals should be within 10 inches for SATA Gen1 and 5 inches for SATA Gen2 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 5 mils, and less trace mismatch is recommended.

Figure 3-26 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.

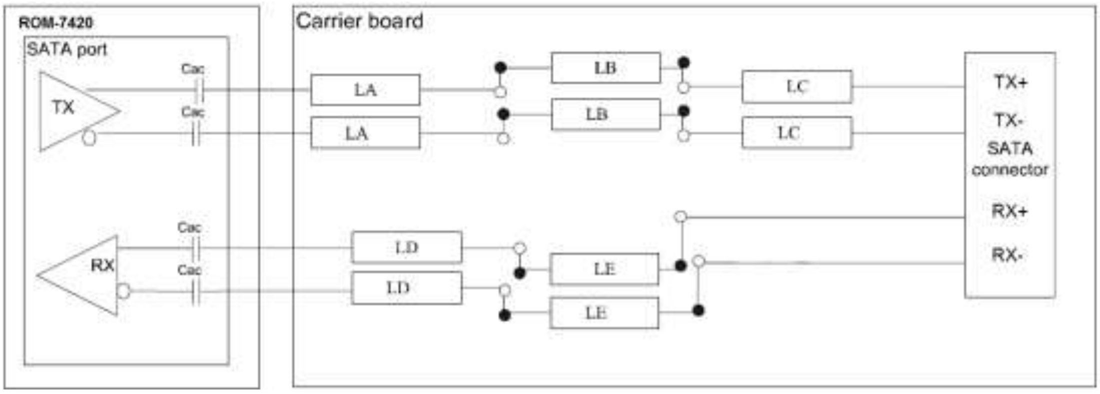


Figure 3-26 SATA traces length matching

3.9 PCI Express Bus

ROM-7420 provides a PCI Express Bus interface that is compliant with the PCI Express Base Specification, Revision 2.0. It supports one general purpose PCI Express port (x1).

3.9.1 Signal Description

Table 3-11 shows ROM-7420 PCI Express bus signals for general purpose.

Table 3-11 PCI-E Signals Description

Signal	Pin	I/O	Description
PCIE0_TX+ PCIE0_TX-	179, 181	O	PCI Express channel 0, Transmit Output differential pair.
PCIE0_RX+ PCIE0_RX-	180, 182	I	PCI Express channel 0, Receive Input differential pair.
PCIE_CLK_REF+ PCIE_CLK_REF-	155, 157	O	PCI Express reference clock output.
PCIE_WAKE#	156	I	PCI Express Wake Event.
PCIE_RST#	158	O	Reset signal for PCI Express device.

PCIE_PWR_EN	172	O	Turn on power for Express device.
PCIE_DIS#	178	O	3G/Wireless enable for external devices.

3.9.2 Schematic Guidelines

3.9.2.1 PCI Express AC Coupling Capacitor

Each PCI Express lane is AC coupled between its corresponding transmitter (TX) and receiver (RX). Figure 3-27 and Figure 3-28 shows the connection for ROM-7420 signals and PCI Express connector. The AC coupling capacitors of TX+/- is present on ROM-7420 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.

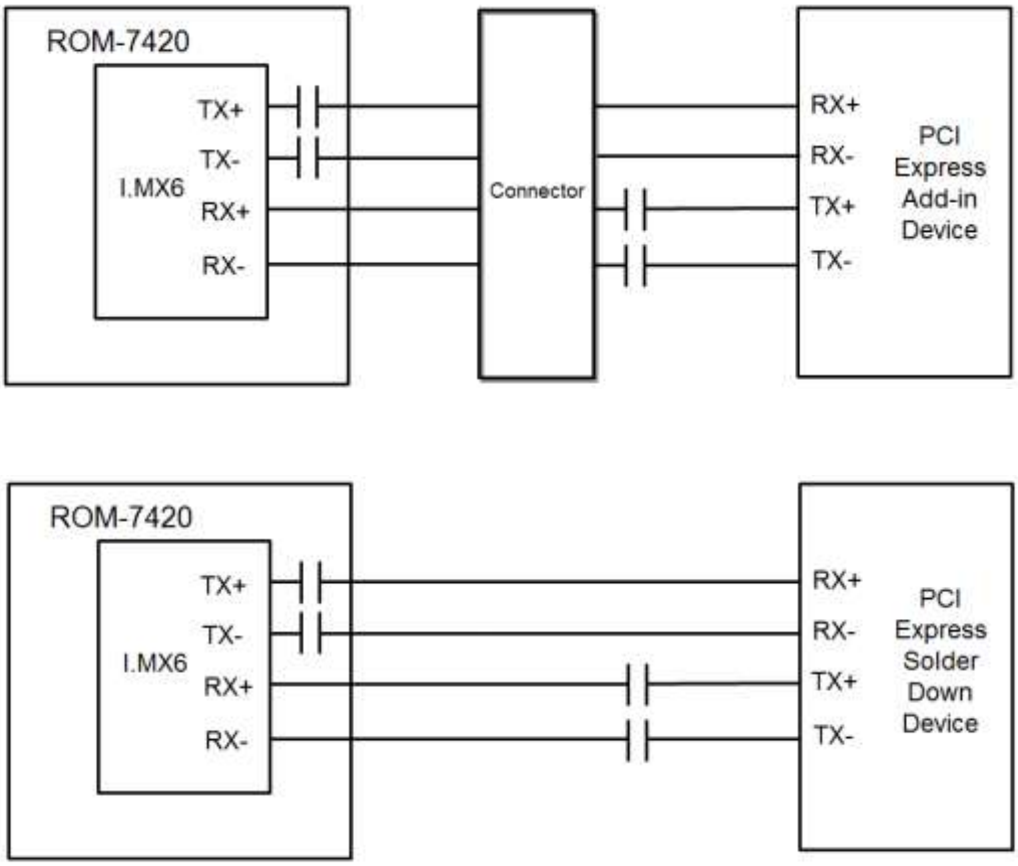


Figure 3-27 PCI Express Interconnect Example

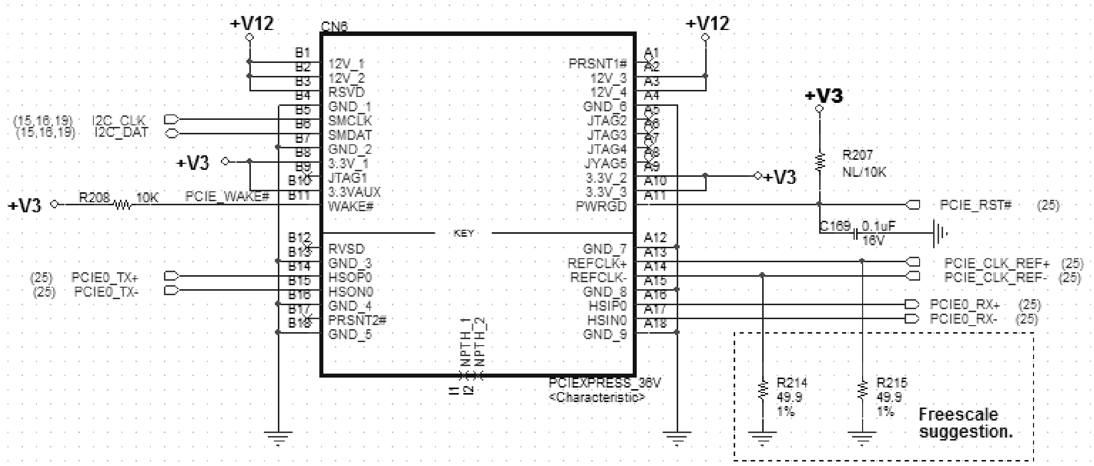


Figure 3-28 PCI Express Connector Schematic Reference

Use the exact same package size for the capacitor on each signal in a differential pair. Table 3-12 shows the PCI Express capacitor reference.

Table 3-12 PCI Express Capacitor Reference

Type	**Value**	**Tolerance**	**Placement**	**Length Matching Between Differential Pair**
AC Capacitor	100nF	20%	Recommended to place close to the transmit side	As close as possible between the differential pairs

3.9.3 Layout Guidelines

This section shows the summary of the layout routing guidelines.

3.9.3.1 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 ROM-7420 platforms the PCI Express differential trace impendence target is 85 Ω ± 10%. 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 LB‘ …, and so on.

It is preferable to route TX and RX differential pairs alternately on the same layer at outer layer and different layer at inner layer. 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.

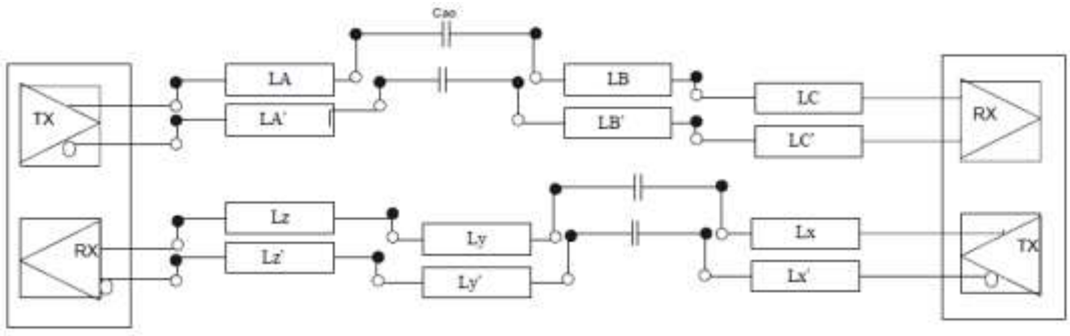


Figure 3-29 Trace Length Matching in Each Segment

3.10 HDMI

ROM-7420 provides a HDMI interface that is compliant with the HDMI 1.4. HDMI (High-Definition Multimedia Interface) is a compact audio/video interface for transmitting uncompressed digital video data and uncompressed/compressed digital audio data.

3.10.1 Signal Description

Table 3-13 shows ROM-7420 HDMI signals, including pin number, signals, I/O and descriptions.

Table 3-13 HDMI Signals Description

Signal	Pin	I/O	Description
TMDS_LANE0+ TMDS_LANE0-	143, 145	O	TMDS differential pair lines lane 0.
TMDS_LANE1+ TMDS_LANE1-	137, 139	O	TMDS differential pair lines lane 1.
TMDS_LANE2+ TMDS_LANE2-	149, 151	O	TMDS differential pair lines lane 2.
TMDS_CLK+ TMDS_CLK-	131, 133	O	HDMI Differential Pairs clock lines
HDMI_CTRL_DAT	150	I/O	DDC based control signal (data) for HDMI device.
HDMI_CTRL_CLK	152	I/O	DDC based control signal (clock) for HDMI device.
DP_HDMI_HPD#	153	I	Hot plug detection signal that serves as an interrupt request.

3.10.2 Schematic Guidelines

3.10.2.1 EMI

The HDMI signals can be routed directly from the ROM-7420 module to the HDMI connectors.Figure 3-30 shows one pair of HDMI connections. Differential pair can use the common-mode choke for EMI compliance if needed.

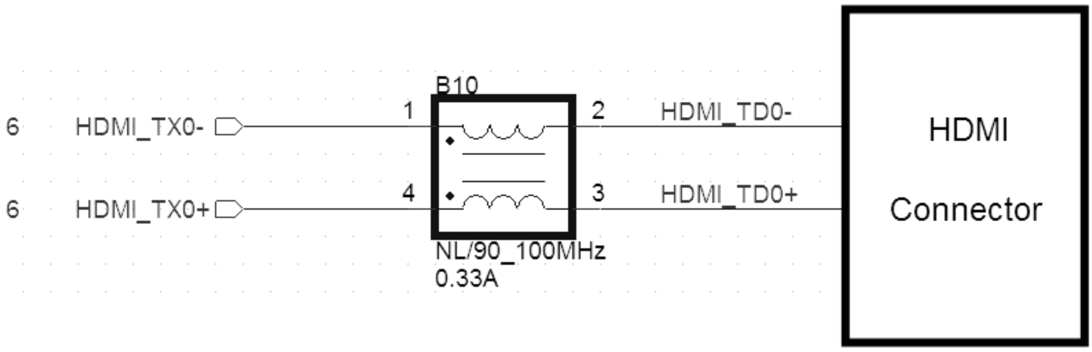


Figure 3-30 One HDMI Differential Pair with common-mode Choke Design

3.10.2.2 DDC level shift

The HDMI_CTRL_DAT and HDMI_CTRL_CLK signals must connect to the HDMI monitor to detect the monitor-type info, the DDC pulled-up voltage is 3.3V on module board , DO NOT pull up DDC signal on carrier board, you can through level shift circuit to change pull up voltage from 3.3V to 5V. ESD protection voltage could be 3.3V or 5V depends on the power map of the carrier board

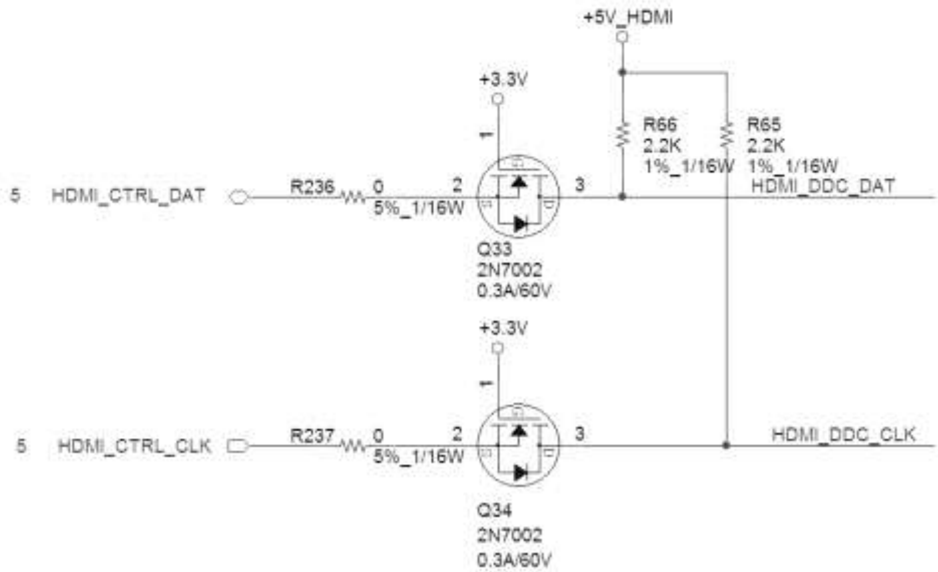


Figure 3-31 HDMI DDC level-shift circuit Design

3.10.2.2 Reference schematic

The reference schematic of HDMI is shown in FIigure 3-32. The HDMI_HPD must have a buffer IC to change input voltage from 5V to 3.3V.

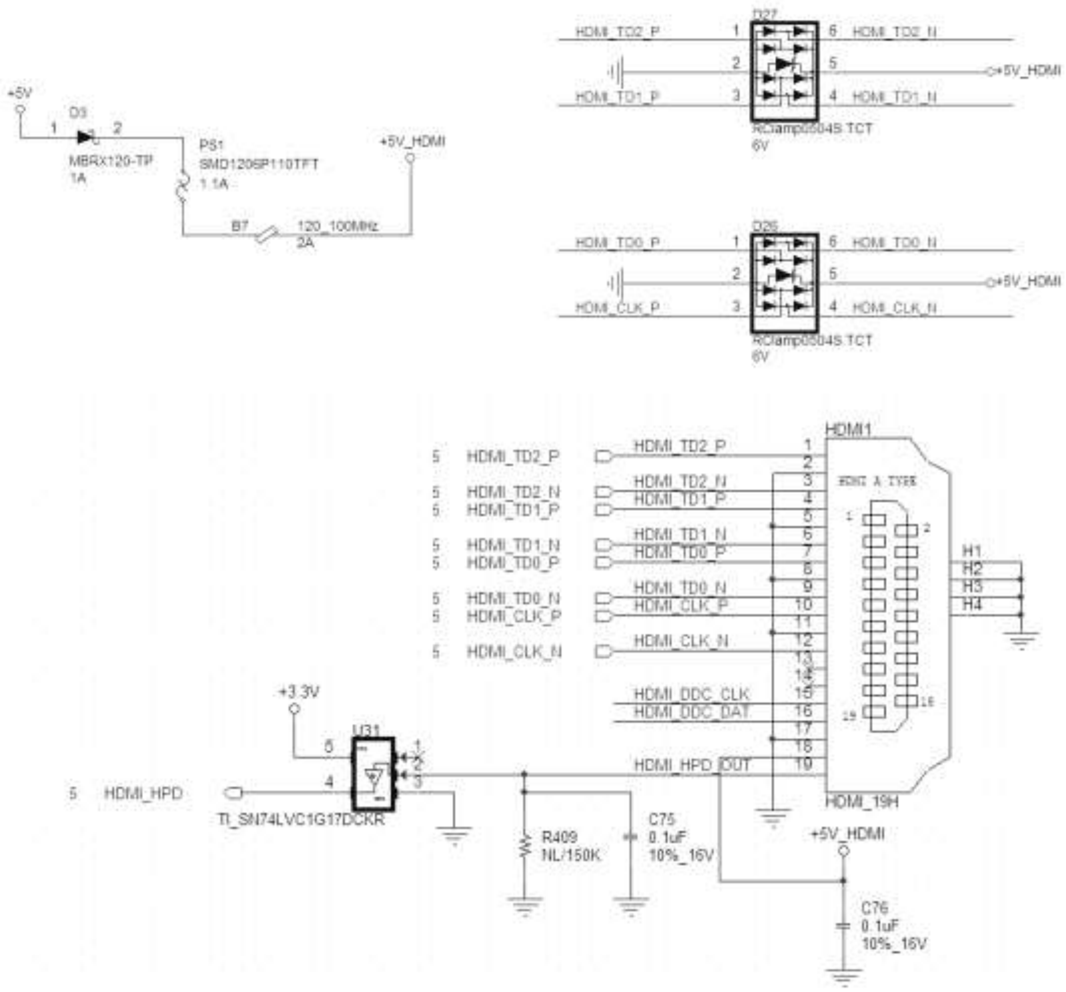


Figure 3-32 HDMI reference schematic

3.11 SPI Interface

Advantech ROM-7420 provides additional SPI interface for extended functions, for example, user can attach a SPI flash device for storage purpose as well as ADC for analog data acquirement, audio codec for audio applications, or I/O devices for specific purposes such as a capacitive touch screen controller.

Please note that user CANNOT use this SPI interface for booting function.

3.11.1 Signal Description

Table 3-14 shows ROM-7420 SPI signals, including pin number, signals, I/O and descriptions.

Table 3-14 SPI Signals Description			
Signal	**Pin**	**I/O**	**Description**
SPI_MOSI	199	O	Data out from module to carrie SPI
SPI_MISO	201	I	Data in to module from carrier SPI
SPI_SCK	203	O	Clock from module to carrier SPI
SPI_CS0	200	O	Chip select for carrier board SPI – may be sourced from chipset SPI0
SPI_CS1	202	O	Chip select for carrier board SPI – may be sourced from chipset SPI1

3.11.2 Schematic Guidelines

3.11.2.1 Reference schematic (Single Device)

SPI Point-to-Point Topology is shown in Figure 3-33.

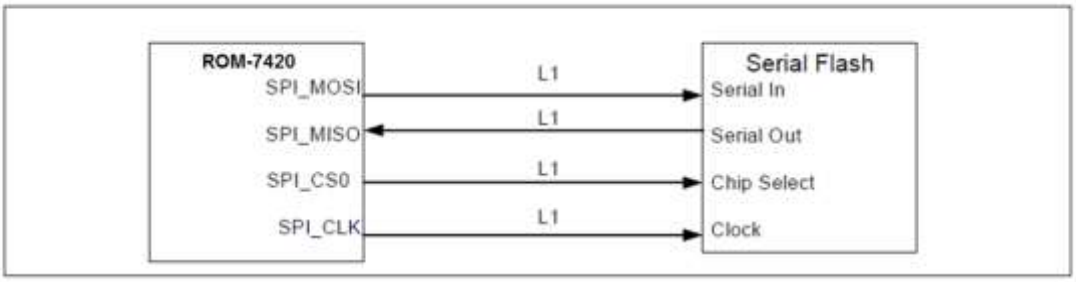


Figure 3-33 SPI Point-to-Point Topology

3.11.2.2 Reference schematic (2 Flash Device)

Dual SPI Devices - Daisy Chain Topology is shown in Figure 3-34.

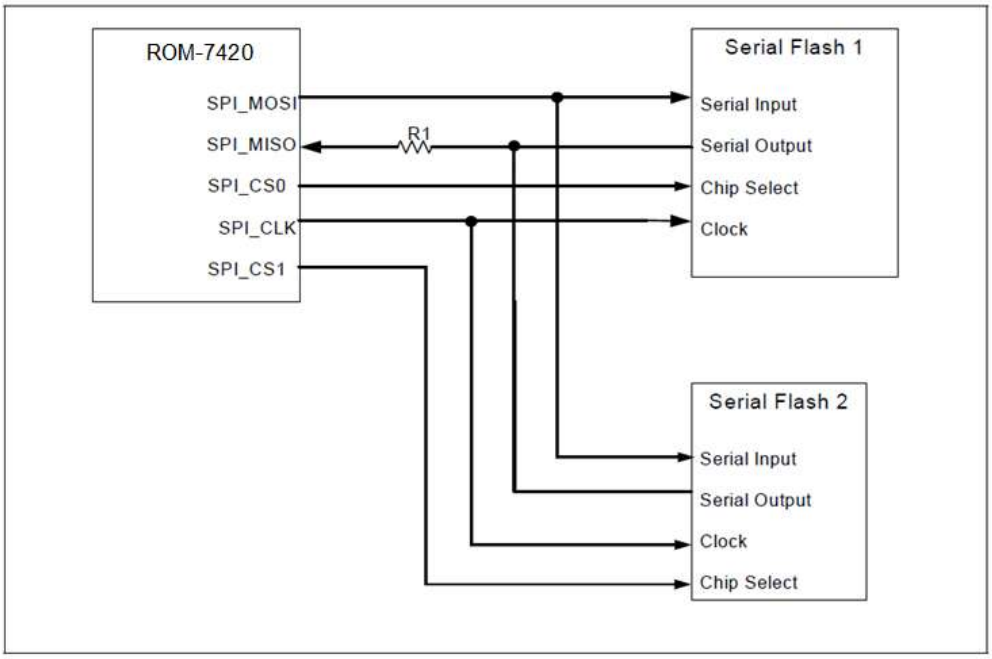


Figure 3-34 SPI Daisy Chain Topology

3.11.3 Layout Guidelines

3.11.3.1 Layout guidelines for single device

Table 3-15 lists guidelines of SPI signals which is for single device in PCB routing stage, note that trace width of single-ended signals differ from PCB stack and copper thickness; even differ from each layer inside one PCB.

Table 3-15	
Parameter	**Trace Routing**
Maximum trace length allowance for all signals on carrier board	8 inches
Single-ended impedance	50Ω ± 10%

3.11.3.2 Layout guidelines for dual device

Table 3-16 lists guidelines of SPI signals which is for dual device in PCB routing stage, do not routing L2 or L3 trace length too long.

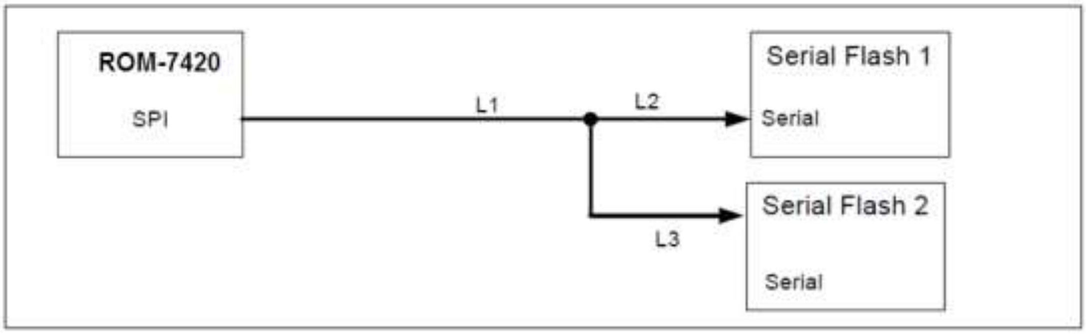


Figure 3-35 Dual SPI ROM routing

Table 3-16	
Parameter	**Trace Routing**
L1	1.5 – 5.5”
L2	0.5 – 2.5”
L3	L2 ± 0.1”
Single-ended impedance	50Ω ± 10%

3.12 General Purpose Input/Output (GPIO)

ROM-7420 provides maximum 8 general purpose input / output signals to carrier board. All signals operate at 3.3V level, those GPIO can not change its direction dynamically.

3.13 CAN Bus

Controller Area Network (CAN or CAN-bus) is a message based protocol designed specifically for automotive applications but now is also used in other areas such as industrial automation and medical equipment.
ROM-7420 modules can optionally support two CAN bus ports.

3.13.1 Signal Description

Table 3-17 shows ROM-7420 CAN bus signals, including pin number, signals, I/O and descriptions.

Table 3-17 CAN Signals Description			
Signal	**Pin**	**I/O**	**Description**
CAN0_TX	129	O	CAN1 Transmit output
CAN0_RX	130	I	CAN1 Receive input
CAN1_TX	144	O	CAN2 Transmit output
CAN1_RX	146	I	CAN2 Receive input

3.13.2 Schematic Guidelines

it is necessary to add transceiver hardware on the carrier board. Please pay attention, if use 5V transceiver, it must has a buffer IC to transfer voltage from 5V to 3.3V with CAN0_RX signal, cause of CAN bus signal from CPU is 3.3V tolerance.

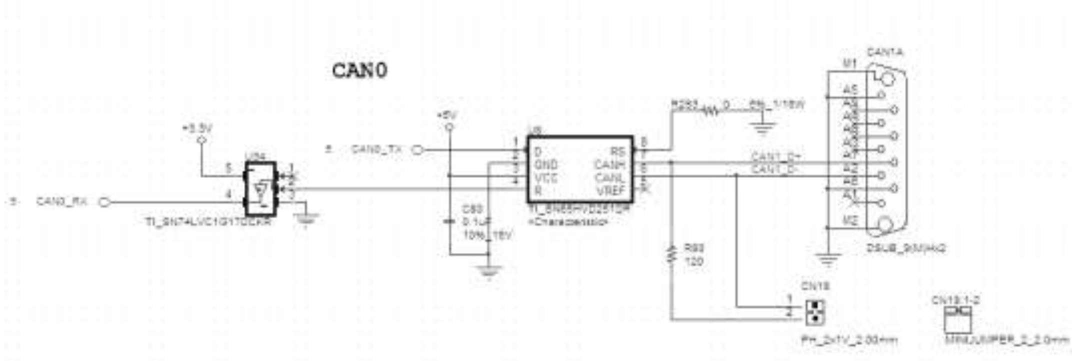


Figure 3-36 CAN bus reference schematic

3.14 I2C Bus Bus

Due to the simple two-wire serial bus protocol and the high availability of devices, the I2C Bus is a frequently used low speed bus interface for connecting embedded devices such as sensors, converters or data storage. ROM-7420 modules provide one I2C bus on the module's connectors.

3.14.1 Signal Description

Table 3-18 shows ROM-7420 I2C bus signals, including pin number, signals, I/O and descriptions.

Table 3-18 I2C Signals Description

Signal	Pin	I/O	Description
I2C_CLK	66	I/O	Clock line of I2C bus.
I2C_DAT	68	I/O	Data line of I2C bus.

3.14.2 Design Guidelines

The maximum amount of capacitance allowed on I2C bus line (CLK, DAT) is 200pF (determined by the 10KΩ pull-up resistor value on ROM-7420 module). A general guideline of I2C standard mentioned that an IC input has 12pF capacitance and PCB trace has 3.8pF capacitance per inch of trace length. Hence it is suggested that customer place less than 12 devices for each I2C channel on carrier board. If device number needs to be greater than 12, customer could add another 4.7KΩ pull-up resistor on carrier board to give more driving strength and extend recommended device number from 12 to 27.

3.15 Watchdog control signals

The Watchdog on a ROM-7420 module can be hardware-triggered by an external control circuitry. When generating a low level pulse on the ROM-7420 module's *'WDTRIG#'* (Watchdog trigger signal) signal, the Watchdog timer will be reset and restarted. If the Watchdog timer has expired without a software or hardware trigger occurrence, the ROM-7420 module will do a internal reset by itself.

3.16 Power management signals

Table 3-19 Power Management Signals Description

Signal	Pin	I/O	Description
PWGIN	26	I	Power OK from main power supply. A high value indicates that the power for module domain is good (Not for carrier board circuits) Pulled up on module.
CB_PWR_EN	18	O	Carrier board circuits should not be powered up until the module asserts the CB_PWR_EN signal.
RSTBTN#	28	I	Reset input from carrier board. Carrier drives low to force a Module reset. Pulled up on module.
SLP_BTN#	21	I	Sleep indicator from carrier board. May be sourced from user Sleep button or carrier logic.

3.16.1 Power Good Input Signal PWGIN

This signal is generated by the power supply circuitry on carrier board and indicates that the power for module domain is ready. The power is not include carrier board power domain.

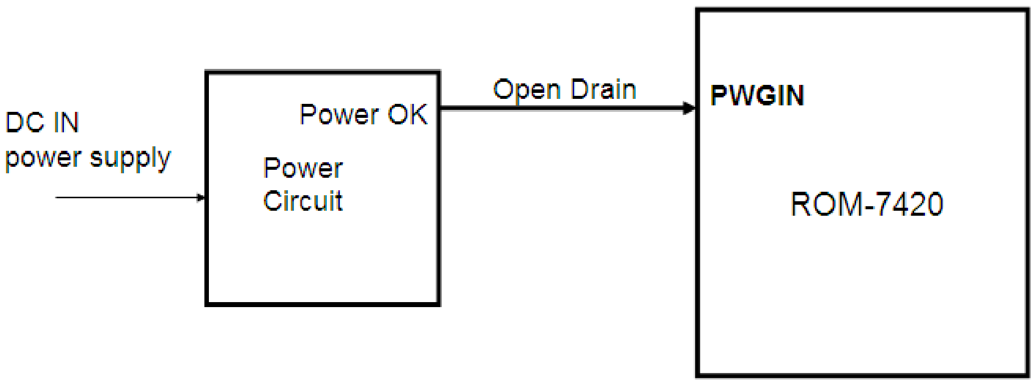


Figure 3-37 PWGIN generation

3.16.2 Carrier board power enable signal CB_PWR_EN

This signal is generated from CPU on ROM-7420 module board, it indicates that the carrier board circuits should not be powered up until the module assert the CB_PWR_EN signal.

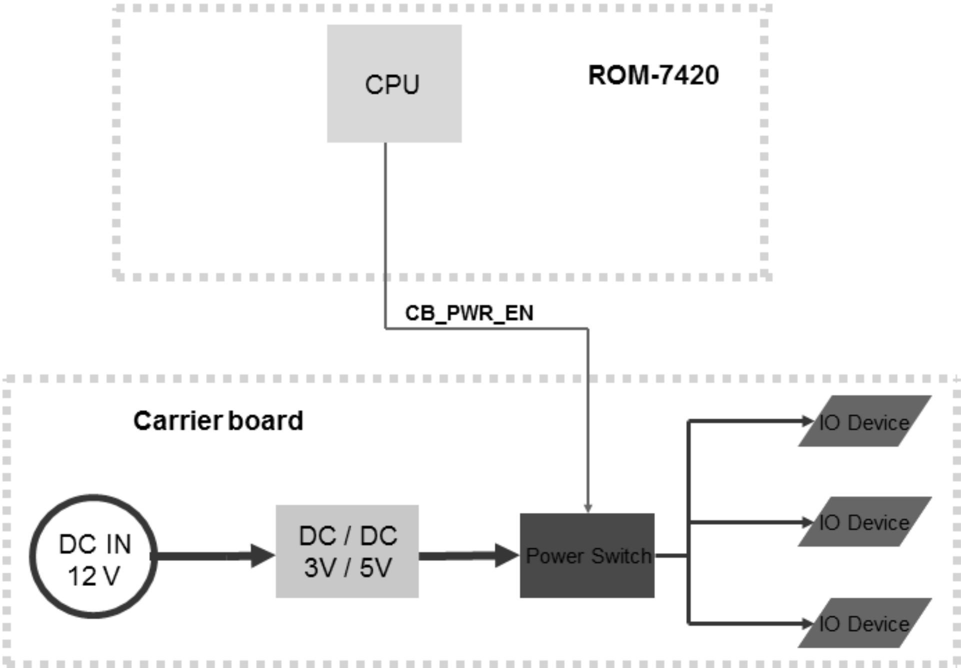


Figure 3-37 CB_PWR_EN implement

3.16.3 Reset signal RSTBTN#

Reset button input. This input may be driven active low by an external circuitry to reset the ROM-7420 module. The signal is pulled up on module.

Chapter 4 Input Power

This chapter provides the power supply design recommendations for ROM-7420 and provide customer's carrier board design reference.

4.1 Module power supply range

Table 4-1 Power supply range

Power Supply	Minimun	Nominal	Maximum
VCC	4.75V	5.00V	5.25V

4.2 Module power DC Characteristics

Table 4-2 Power consumption

Power Supply	Voltage (V)	Standby Mode (A)	Typical (A)	Maximum (A)
VCC	5.00V	N/A	N/A	0.6306

4.3 Power Delivery Block Diagram

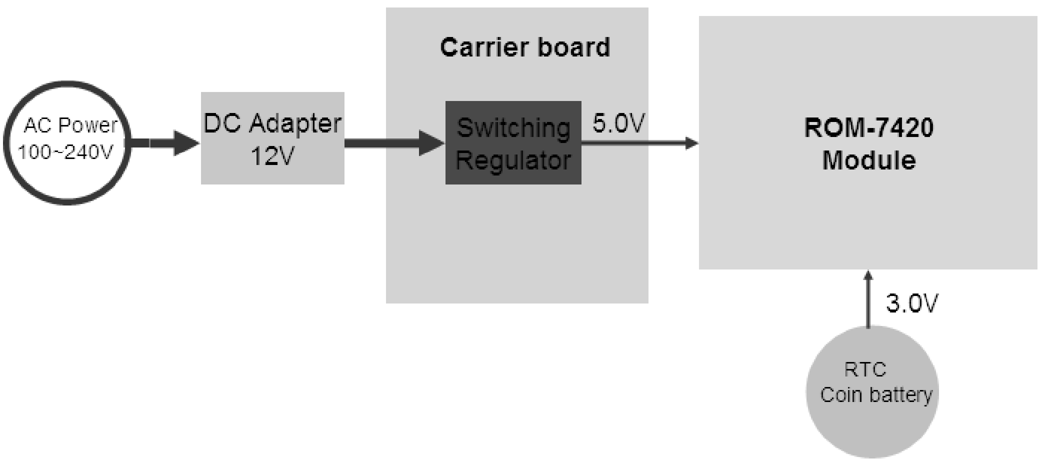


Figure 4-1 Carrier Board Reference Power Delivery Block Diagram

4.3 System Power Domains

It is useful to describe an system as being divided into a hierarchy of two power domains:

(1) ROM-7420 Module power domain

(2) Carrier Board Circuits power domain

The ROM-7420 Module domain includes the ROM-7420 module and ***may*** include a serial EEPROM on the carrier board, connected to the I2C_PM I2C bus in the Module power domain, allowing Module software to read carrier board parameters. The carrier circuits domain includes “every thing else” (and does not include items from the Module domain, even though they may be mounted on the carrier).

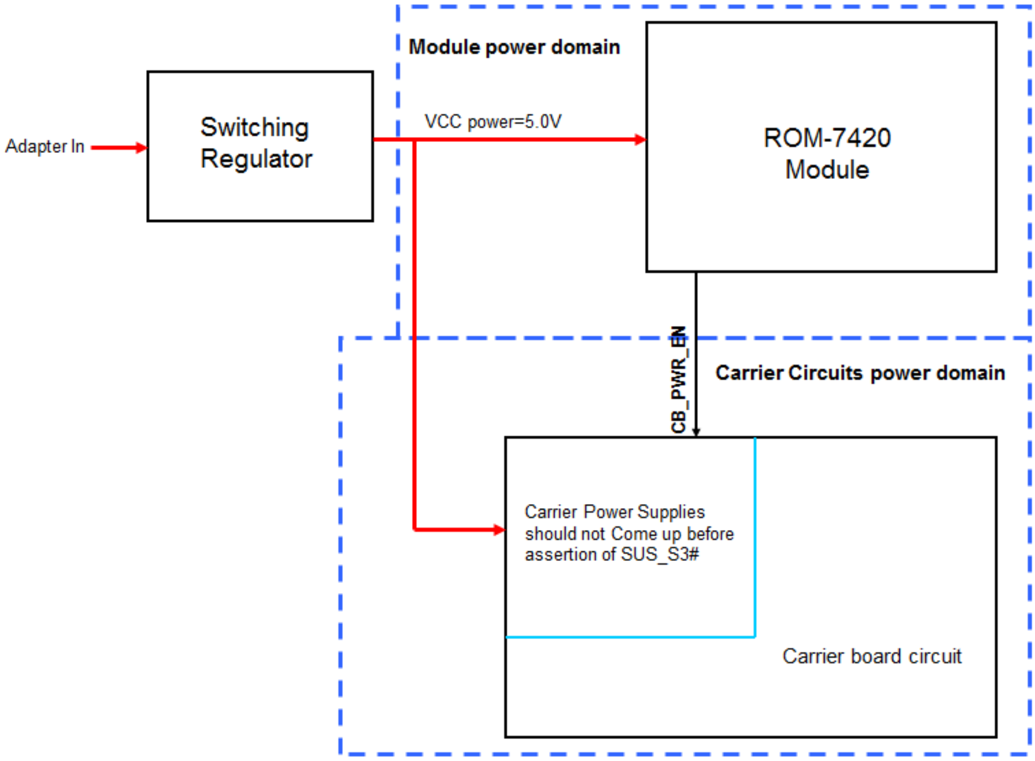


Figure 4-2 System power domain

4.4 RTC Battery Implementation

If customer needs to keep timer while system power off, a coin battery is required. Figure 4-3 gives an example of designing coin battery circuits. Using battery is Rayovac® BR2032 non-rechargeable coin battery, with battery capacity 195mAh and working temperature -40~85℃. R199 in this figure is an easy current limit circuit to protect over discharge situation.

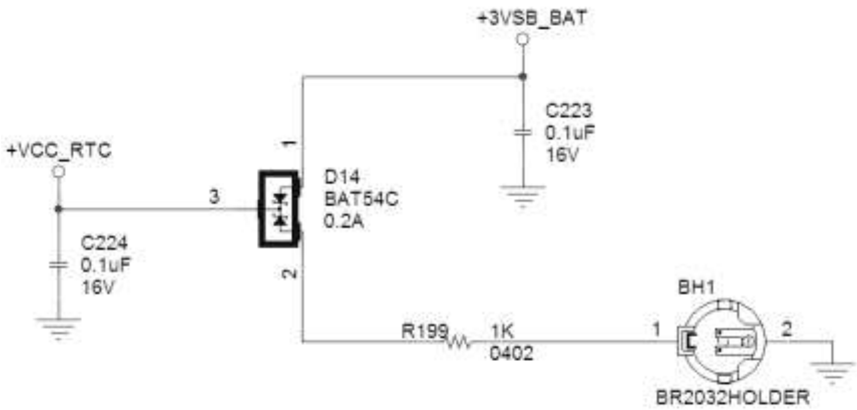


Figure 4-3 RTC Coin Battery Circuits

Chapter 5 PCB Layout Guidelines

A brief description of the Printed Circuit Board (PCB) for ROM-7420 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 eight-layer board is preferred.

5.1 Nominal Board Stack-Up

The trace impedance typically noted (50 Ω ± 10%) is the “nominal” trace impedance for a 5-mil wide external trace and a 4-mil wide internal trace. However, some stackups
may lead to narrower or wider traces on internal or external layers in order to meet the 50 Ω ± 10% 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. Also, all high speed signals should have continuous GND referenced planes and cannot be routed over or under plane splits.

5.1.1 Four layer board stack-up

Figure 5-1 illustrates an example of a four-layer stack-up with 2 signal layers and 2 power planes. The two power planes are the power layer and the ground layer. The layer sequence of component-ground-power-solder is the most common stack-up arrangement from top to bottom.

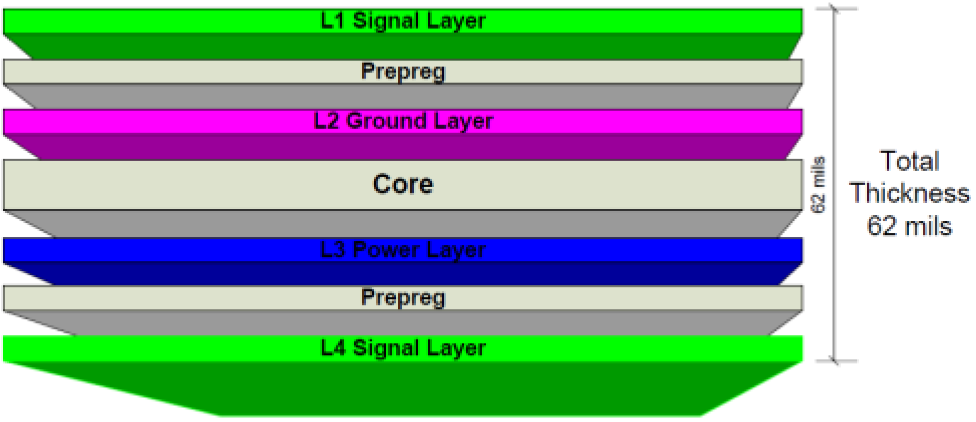


Figure 5-1 Four-Layer Stack-up

Table 5.1: Recommended Four-Layer Stack-Up Dimensions

Dielectric Thickness (mil)	Layer No	Layer Type	Single-End Signals		Differential Signals		USB differential Signals	
			Width (mil)	Impedance (ohm)	Width (mil)	Impedance (ohm	Width (mil)	Impedance (ohm)
1.7	L1	Signals	5/5	50+/-10%	5/10	100+/-10%	5/6	90+/-10%
4		Prepreg						
1.2	L2	Ground						
47.6		Core						
1.2	L3	Power						
4		Prepreg						
1.7	L4	Signals	5/5	50+/-10%	5/10	100+/-10%	5/6	90+/-10%

Notes:

Target PCB Thickness totals 62mil+/-10%

5.1.2 Six layer board stack-up

Figure 5-2 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.

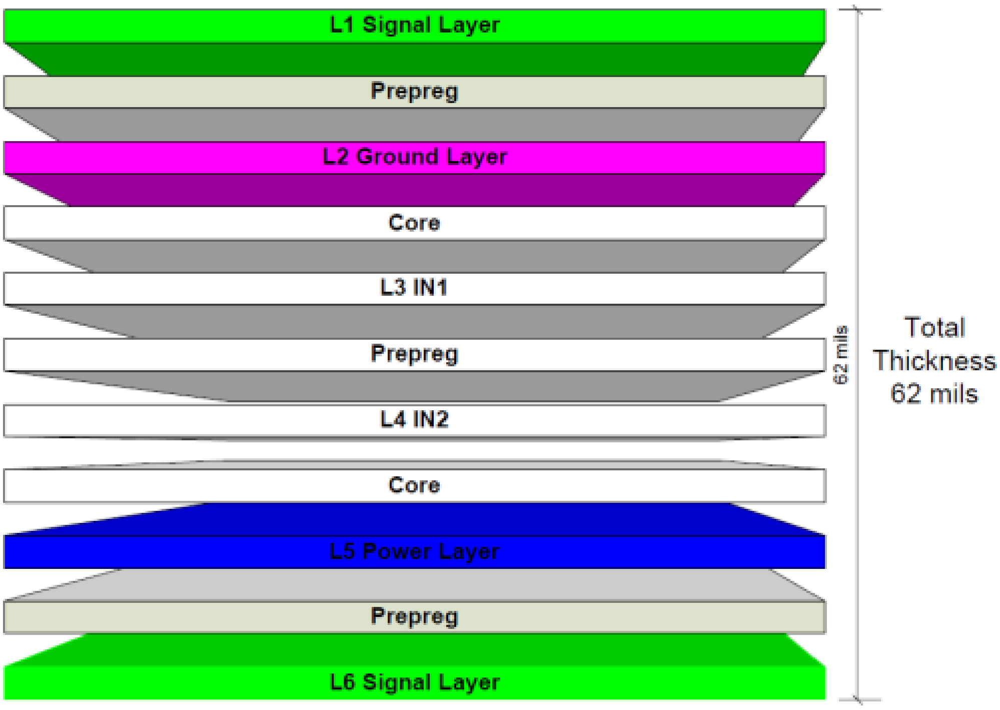


Figure 5-2 Six-Layer Stack-up

Table 5.2: Recommended Six-Layer Stack-Up Dimensions

Dielectric Thickness (mil)	Layer	Layer	Single-End Signals		Differential Signals		USB differential Signals	
	No	Type	Width (mil)	Impedance (ohm)	Width (mil)	Impedance (ohm	Width (mil)	Impedance (ohm)
1.7	L1	Signals	5/5	50+/-10%	**5/10**	100+/-10%	5/6	90+/-10%
4		Prepreg						
1.2	L2	Ground						
4		Core						
1.2	L3	IN1	5/5	50+/-10%	**5/10**	100+/-10%	5/6	90+/-10%
38		Prepreg						
1.2	L4	IN2	5/5	50+/-10%	**5/10**	100+/-10%	5/6	90+/-10%
4		Core						
1.2	L5	Power						
4		Prepreg						
1.7	L6	Signals	5/5	50+/-10%	**5/10**	100+/-10%	5/6	90+/-10%

Notes:

Target PCB Thickness totals 62mil+/-10%

5.2 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.

5.3 Differential Impedance Targets for Trace Routing

Table 5.3 shows the target impedance of the differential signals. The carrier board should follow the required impedance in this table.

Table 5.3 Differential Signals Impedance Requirement

Signal Type	Impedance
HDMI	100 Ω ± 10%
LVDS	100 Ω ± 10%
SATA	100 Ω ± 10%
USB	90 Ω ± 10%
PCI Express	85 Ω ± 10%
LAN	100 Ω ± 10%

Chapter 6 Mechanical Characteristics Guidelines

6.1 ROM-7420 Mechanical Dimensions

The PCB size of the ROM-7420 module is 70mm x 70mm. The PCB thickness is 1.2mm to allow high layer count stack-ups. Figure 6-1 shows the ROM-7420 module board mechanical dimensions, unit is millimeter. The 230 pin connector pair shall be mounted on the backside of the PCB and is seen “through” the board in this view.

The holes shown in this drawing are intended for mounting the module / heatspreader 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 6-1 shows the ROM-7420 Module board mechanical dimensions. The unit is millimeters.

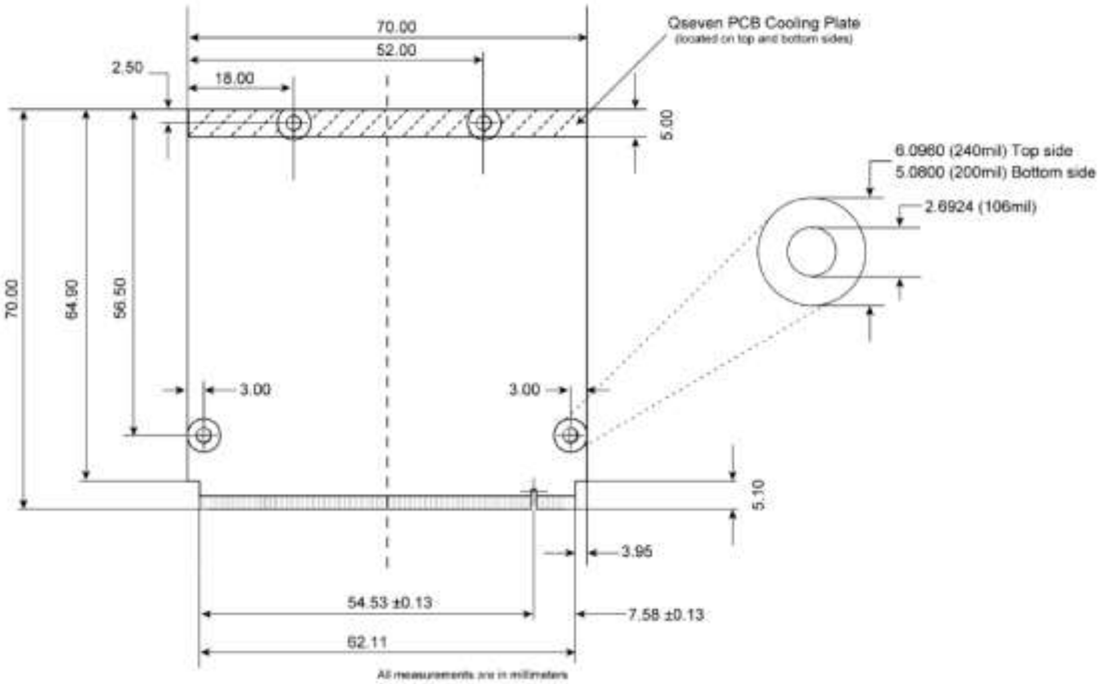


Figure 6-1 ROM-7420 Module Board Mechanical Dimensions

6.2 Carrier Board Connector

The carrier board utilizes a 230-pin card-edge connector to connect the ROM-7420 module. Originally, this card-edge connector was designed for MXM graphics modules that are used for PCI Express capable notebook graphics cards. The card-edge connector is following the MXM specification and therefore this connector type is also known as a MXM connector.

The MXM edge connector is the result of an extensive collaborative design effort with the industry's leading notebook manufacturers. This collaboration has produced a robust, low-cost edge connector that is capable of handling high-speed serialized signals.

The MXM connector accommodates various connector heights for different carrier board applications needs. This specification suggests two connector heights, 7.8mm and 5.5mm.

6.2.1 MXM Connector Dimensions

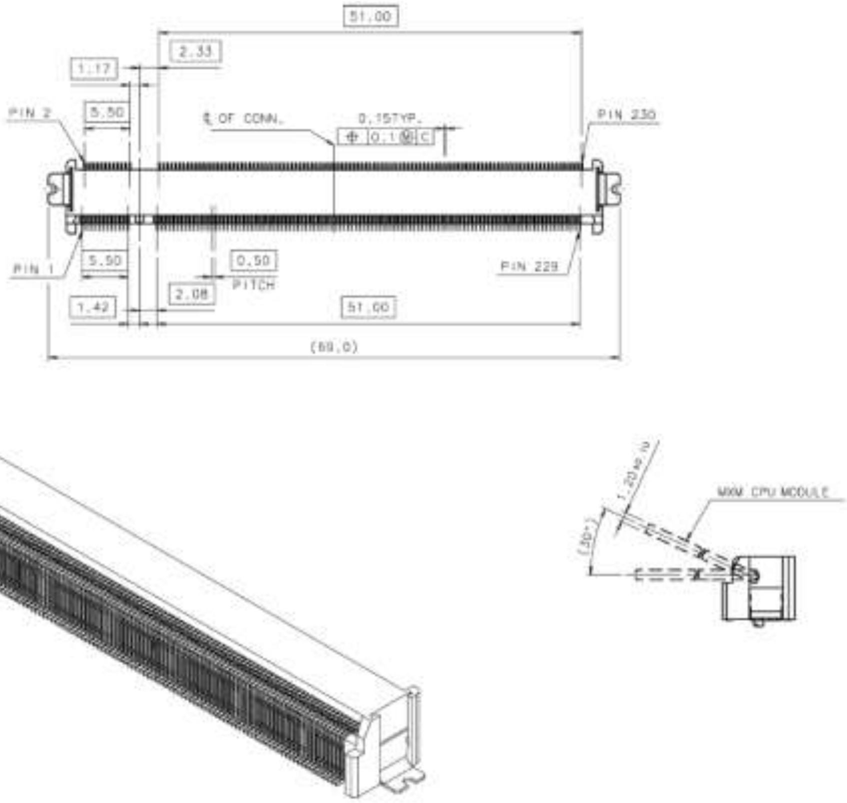


Figure 6-2 MXM Connector

6.2.2 MXM Connector Footprint

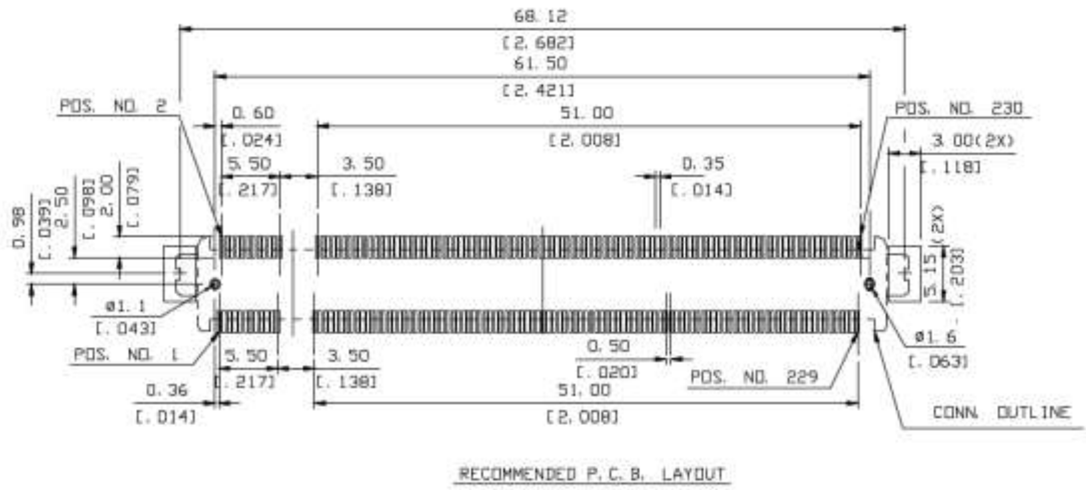


Figure 6-3 Carrier Board PCB Footprint for the MXM connector

6.2.3 ROM-7420 Module and MXM Connector PCB Footprint

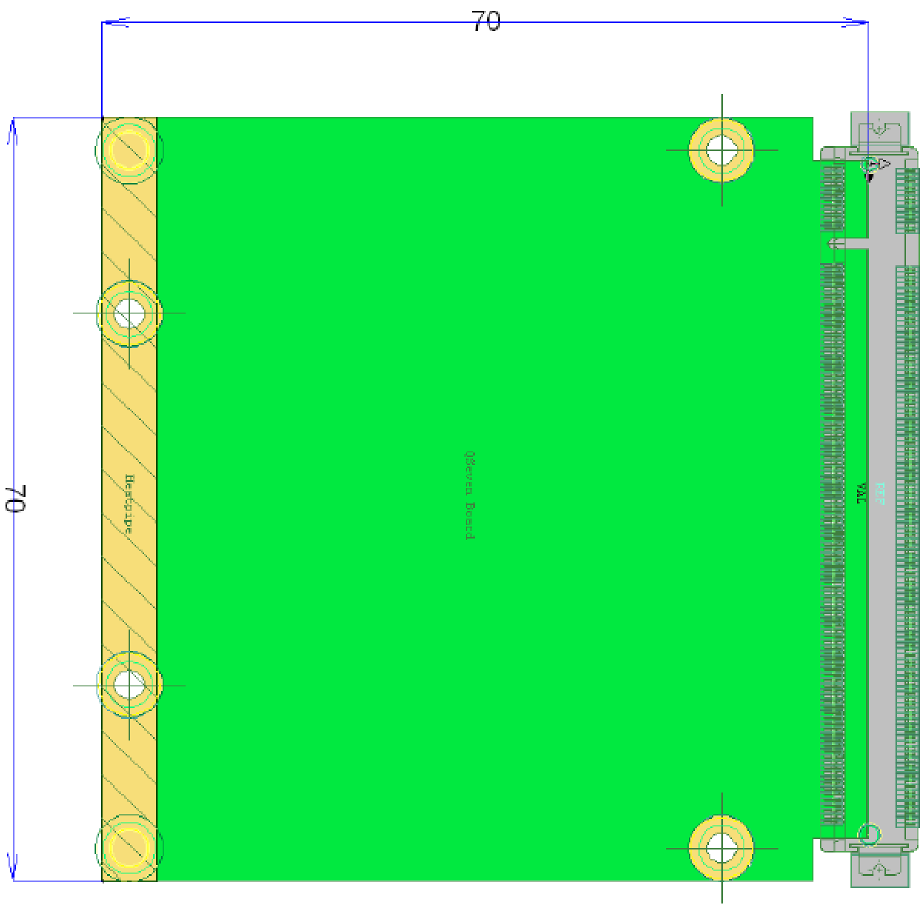


Figure 6-4 PCB footprint for The ROM-7420 Module inserted in the MXM carrier board connector