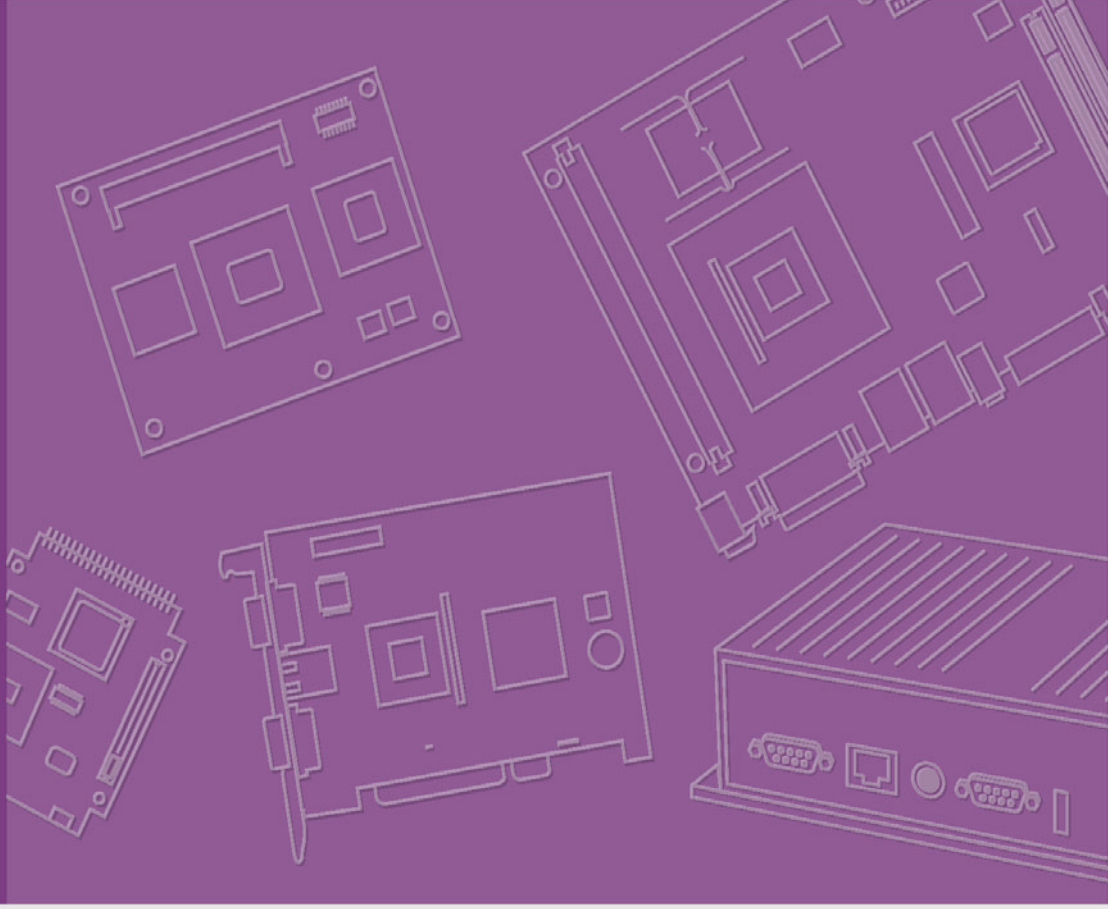


User Manual

EVA-X4300

System Design Guide

Trusted ePlatform Services

ADVANTECH

Copyright

Part No. 2006X43020 Edition 1

Printed in Taiwan August 2008

Contents

Chapter 1

Overview

1.1 Overview

The EVA-X4300 is a fully static 32-bit x86-based processor that is compatible with a wide-range of PC peripherals, applications and operating systems, such as DOS, WinCE, Linux, and most popular 32-bit RTOSs (Real Time OS). It enables maximum software re-use based on its feature of legacy compatibility. The EVA-X4300 integrates 32 KB write-through direct map L1 cache, a PCI bus interface at 33 MHz, an 8/16-bit ISA bus interface, SDR SDRAM, DDR2 SDRAM, a ROM controller, IPC (Internal Peripheral Controller) with DMA and interrupt timer/counter included, FIFO UART, SPI (Serial Peripheral Interface), LPC (low pin count), a USB 1.1/2.0 host controller, LoC (LAN on Chip), an IDE controller, and 256 KB flash within a single 581-pin BGA package to form a an SoC (System-on-Chip) processor. The EVA-X4300 integrates comprehensive features and rich I/O flexibility within a single System-on-Chip, to reduce board design complexity and shorten product development schedules. Taking advantage of ultra low power consumption, the EVA-X4300 is able to operate in a wide range of temperatures without additional thermal design. With the commitment of long term supply guaranteed for the EVA-X4300, customers can extend product life cycle and receive a maximum return on investment.

The EVA-X4300 provides an ideal solution for embedded systems and communication products (such as the thin client, NAT router, home gateway, access point and tablet PC) producing optimal performance.

This system design guide provides detailed usage information of the highly integrated EVA-X4300 SoC processor.

Chapter 2

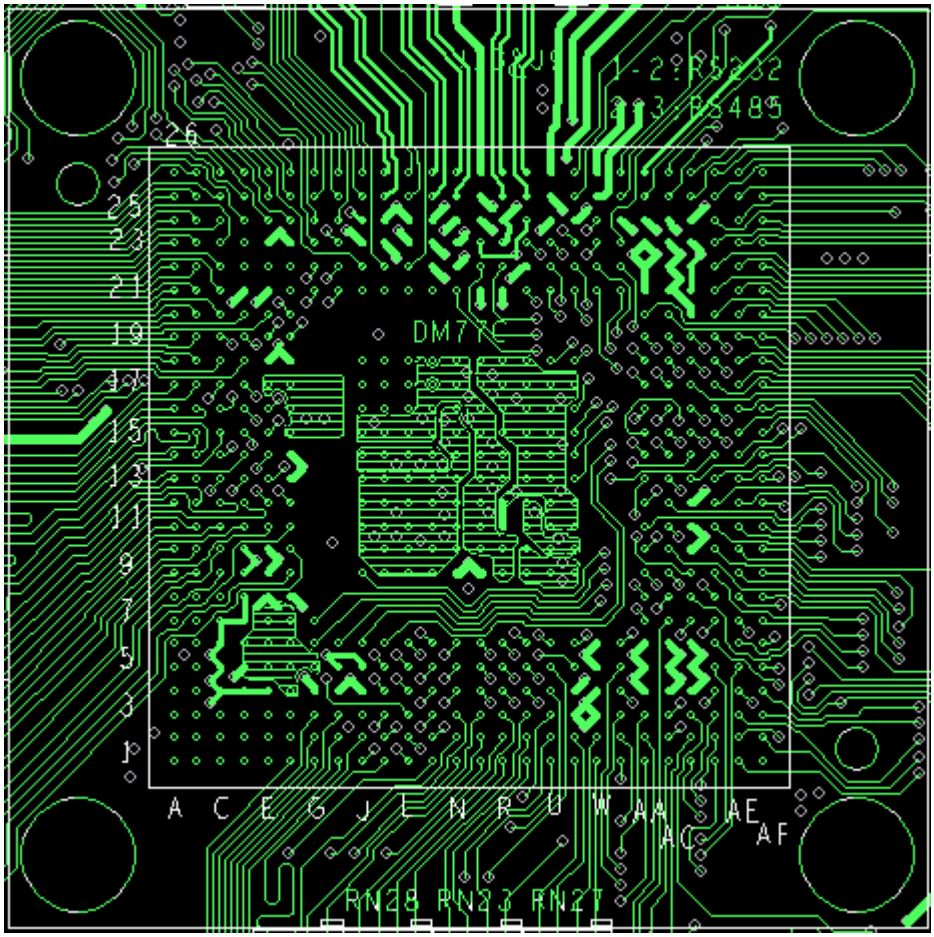
Layout Guide

2.1 BGA Layout Guideline

The package of EVA-X4300 is 581-ball PBGA. The ball diameter is 0.6 mm (24 mil); ball pitch is 1 mm (40 mil). There are 6 rows of balls arranged along the edge of the package.

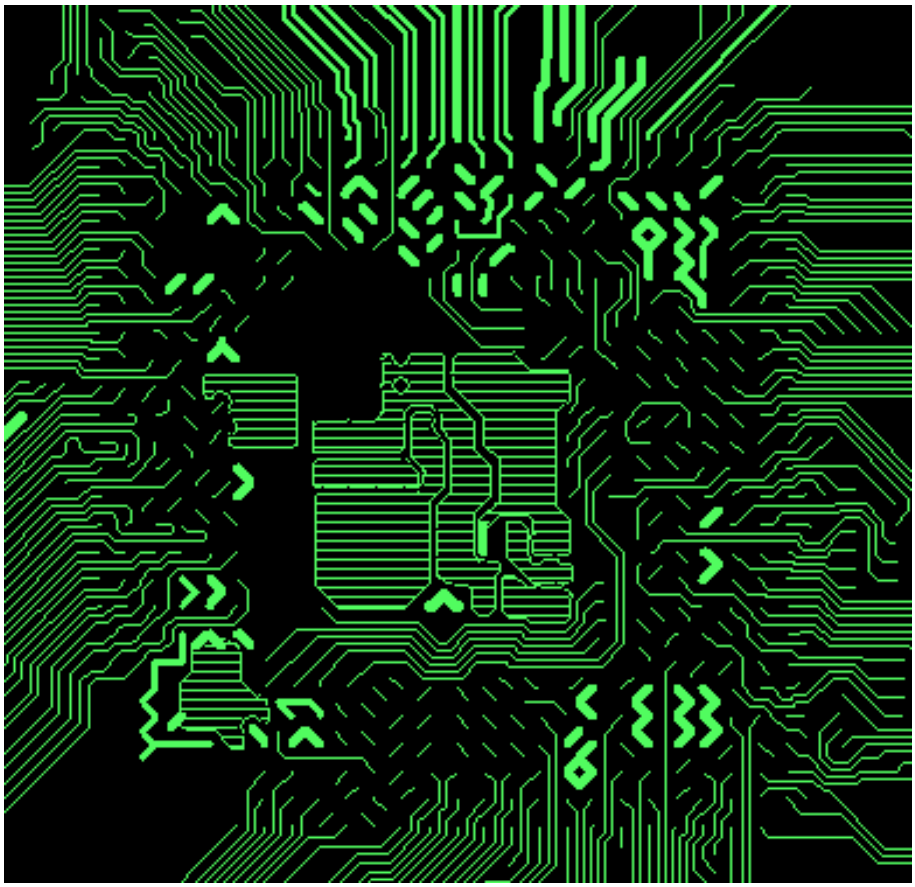
For 6-layer:

- Ball pad diameter: 24 mil
- Traces width/ spacing for Signals: 5.5/ 5 mil
- Trace width for Power and Ground: 20 mils
- Vias PAD/ Drill diameter for all Signals: 14/ 8 mil
- Via-to-via spacing (center to center): 40 mil
- Via-to-pad spacing (center to center): 30 mil



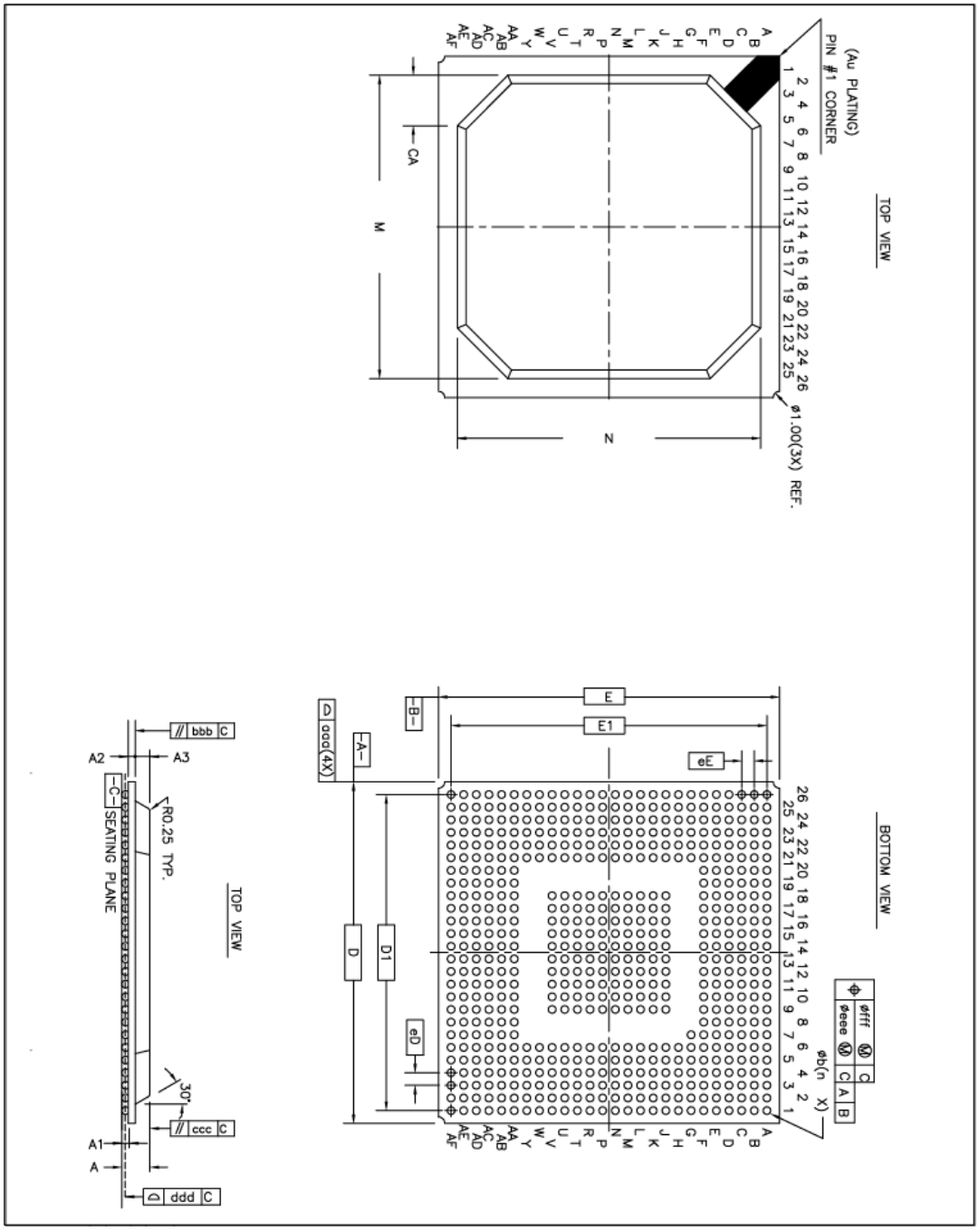
For 4-layer:

- Ball pad diameter: 14 mil
- Traces width/ spacing for Signals: 5/ 5 mil
- Trace width for Power and Ground: 20 mils
- Vias PAD/ Drill diameter for all Signals: 22/ 12 mil
- Via-to-via spacing (center to center): 40 mil
- Via-to-pad spacing (center to center): 30 mil



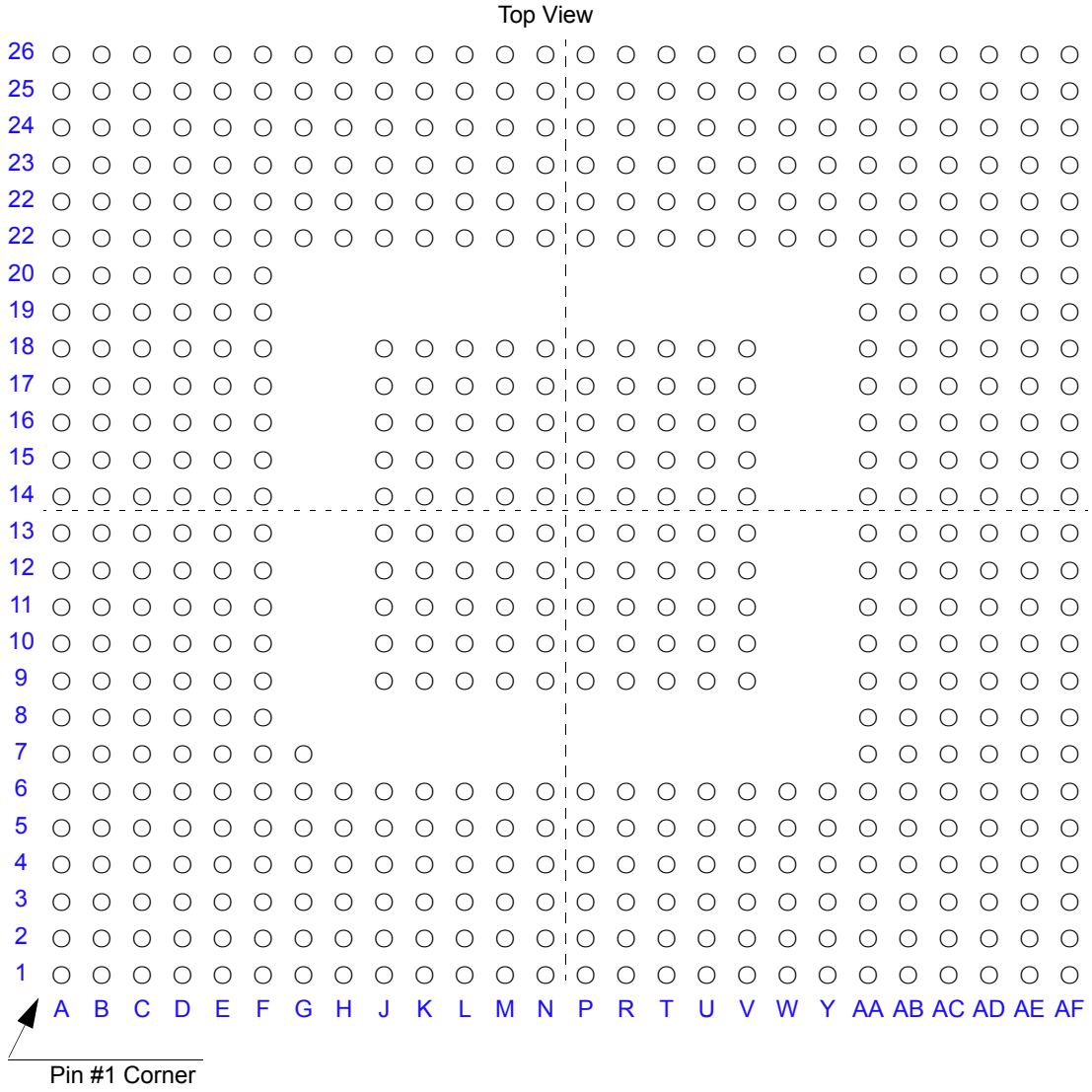
Note! *All the Vias must be covered by solder mask. To minimize inductance, power vias should be as large as possible and take good care of the inadvertently cut of the ground and power planes.*

2.2 Package Outline



		Symbol	Common Dimensions
Package :			PBGA
Body Size:	X	D	27
	Y	E	27
Ball Pitch :	X	eD	1
	Y	eE	1
Total Thickness :		A	2.23 +/− 0.13
Mold Thickness :		A3	1.17 Ref.
Substrate Thickness :		A2	0.56 Ref.
Ball Diameter :			0.60
Stand Off :		A1	0.40~0.60
Width :		b	0.50 ~ 0.70
Mold Area :	X	M	24
	Y	N	24
Chamfer		CA	4
Package Edge Tolerance :		aaa	0.20
Substrate Flatness :		bbb	0.25
Mold Flatness :		ccc	0.35
Coplanarity:		ddd	0.20
Ball Offset (Package) :		eee	0.25
Ball Offset (Ball) :		fff	0.10
Ball Count :		n	581
Edge Ball Center to Center :	X	D1	25
	Y	E1	25

2.3 Signal Arrangement



	A	B	C	D	E	F	G	H	J	K	L	M	N
26	NC	AD24	TRDY_	PCIRST_	AD13	AD10	AD5	AD3	AD1	TXP	RXP	DP1	DP0
25	AD25	CBE_3	IRDY_	STOP_	AD14	AD9	AD7	AD4	AD2	TXN	RXN	DM1	DM0
24	AD26	CBE_2	FRAME_	DEVSEL_	AD15	CBE_0	PAR	AD0	ISET	VCCA0	VCCA1	VSSA1	AVSS0
23	AD27	AD16	AD17	TEST0	TEST4	VCC_SPI	AD11	INTA_	VCCAPLL	VSSAPLL	VSSA0	VCCABG	AVDDPLL 0
22	AD28	AD18	AD19	TEST2	TEST1	ATSTP	CBE_1	AD12	AD6	Duplex	Link/Active	VSSABG	AVDD0
21	AD29	AD20	AD21	GND_SPI	TEST3	ATSTN	ROM_CS_	AD8	TXC0	TXEN0	RXDV0	RXC0	RTC_AS_ GPIO_37
20	PCICLK_2	AD30	AD31	AD23	AD22	INTC_							
19	PCICLK_0	PREQ2	PGNT0	PGNT2	INTD_	INTB_							
18	PCICLK_1	PREQ1	PREQ0	PGNT1	VCC3V	VCC3V			TXD0_3	TXD0_0	RXD0_0	RXD0_3	Vdd_io
17	MD4	MD0	MD14	DQM1	GNDK	GND_R3			TXD0_2	TXD0_1	RXD0_1	RXD0_2	Vdd_io
16	MD3	MD1	MD9	MD15	MD15	GND_R3			MDC	MDIO	COL0	VCC3V	GND_R3
15	MD2	MD7	MD11	MD12	MD8	GND_R3			VCC3V	VCC3V	VCC3V	VCC3V	GND_R3
14	MD5	DQM0	MD13	DQS0	MD10	VCCK			VCCK	VCCK	VCCK	VCCK	GND_R3
13	MD6	CS_1	WE_	RAS_	CS_0	VCCK			GND_R3	GND_R3	GND_R3	GND_R3	GND_R3
12	MA10	MA6	BA2	BA0	CAS_	BA1			GNDK	GNDK	GNDK	GND_R3	GND_R3
11	MA1	MA5	MA7	MA9	MA11	MA13			GNDK	GNDK	GNDK	GNDK	GND_R3
10	MA0	MA3	MA4	VDLL0	GNDDLL0	MA12			GNDK	GNDK	GNDK	GNDK	GND_R3
9	SDRAM-CLKN	SDRAM-CLKP	MA2	VDLL1	GNDDLL1	MA8			TMS	GNDK	GNDK	GNDK	VCCK
8	NC	NC	NC	VCCO	GNDK	VCCK							
7	NC	NC	VCCO	VCCO	GNDK	GNDO	TCK						
6	NC	NC	VCCO	GNDO	GNDO	GNDO	TDO	TDI	SOUT9	SIN9	PE/SDD9	SOUT4	SIN4
5	NC	NC	VCCO	GNDO	GNDO	GNDO	GNDO	GNDPLL0	GNDPLL1	PD6/SDD6	PD5/SDD5	BUSY/ SDD10	SLCT/ SDD8
4	NC	NC	VCCO	VCCO	VCCO	GNDO	VCCK	VPLL0	VPLL1	PD7/SDD7	ACK_/ SDD11	PD4/SDD4	ERR_/ SDD14
3	NC	NC	NC	NC	TEST5	TEST6	SIN3	RTS3_/ SRST_	TESTCLK	PD3/SDD3	AFD_/ SDD15	SLIN_/ SDD12	PDD9
2	NC	NC	NC	NC	TEST7	TEST8	SOUT3	CTS3_/ SIOR_	DCD3_/ SDRQ	PD0/SDD0	PD1/SDD1	PD2/SDD2	PINT
1	NC	NC	NC	NC	NC	DTR3_/ SDACK_	RI3/ SIORDY	DSR3_/ SCBLID_	INIT_/ SDD13	PA2	STB_/ SCS0_	PRST_	PDD3
	A	B	C	D	E	F	G	H	J	K	L	M	N

P	R	T	U	V	W	Y	AA	AB	AC	AD	AE	AF	
AVDD33_0	DP3	DP2	AVDD33_1	RTC_Xin	AVDD3	XOUT_14.318	POWER_GOOD	CLK25MOUT	DCD2_/PWM0CLK	CTS2_/PWM1GATE	DSR2_/PWM0GATE	NC	26
AVSSPLL0	DM3	DM2	AVSSPLL1	RTC_Xout	AVSS3	XIN_14.318	MTBF	CLK24MOut	DTR2_/PWM2OUT	RTS2_/PWM1OUT	TXD_EN2/PWM2GATE	SIN2/PWM2CLK	25
REXT0	AVDD1	AVSS2	REXT1	AVDDPLL1	SERIRQ	Vss_pll_1	Vdd_pll_1	Vdd_pll_0	Vss_pll_0	RI2_/PWM1CLK	SOUT2/PWM0OUT	DCD1_/GPIO_40	24
AVSS1	AVDD2	LAD0	LAD1	LAD2	LAD3	SPEAKER	Vdd_core	Vss_io	Vdd_io	DTR1_/GPIO_45	CTS1_/GPIO_47	DSR1_/GPIO_46	23
RTC_RD_GPIO_36	RTC_IRQ8_/GPIO_34	RTC_PS	SYSFAILOut_	Ext_Switch_fail_	E_SPI_CLK/GPIOP_31	E_SPI_DI/GPIOP_33	Vdd_core	Vss_io	Vdd_io	RI1_/GPIO_43	SOUT1/GPIO_41	RTS1_/GPIO_42	22
VBat	VBatGnd	RTC_WR_GPIO_35	ExtSysFailIn_	EXT_GPCS_	E_SPI_CS/GPIOP_30	E_SPI_DO/GPIOP_32	Vdd_core	Vss_io	Vdd_io	TXD_EN1	SIN1/GPIO_44	GPIO_P2_1/SA25	21
							GPIO_P2_7/SA31	GPIO_P2_6/SA30	Vss_io	GPIO_P2_5/SA29	GPIO_P2_4/SA28	GPIO_P2_2/SA26	20
							GPIO_P1_7	GPIO_P2_0/SA24	GPIO_P1_6	GPIO_P1_5	GPIO_P1_4	GPIO_P1_0	19
Vss_io	Vss_io	Vss_core	LFRAME_	LDRQ_			GPIO_P0_7	GPIO_P1_3	GPIO_P1_2	GPIO_P2_3/SA27	GPIO_P0_5	GPIO_P0_3	18
Vss_io	Vss_io	Vss_core	Vss_io	Vss_io			GPIO_P0_6	GPIO_P1_1	GPIO_P0_1	GPIO_P0_0	GPIO_P0_4	GPIO_P0_2	17
Vdd_io	Vss_io	Vss_core	Vss_core	KBDATA / A20GATE_			SD3	IOR_	GPCS0_	GPCS1_	SD15	SD14	16
Vdd_io	Vss_io	Vss_io	Vss_core	MSDATA			LA20	LA18	LA23	LA19	DRQ7	SD12	15
VCC3V	Vdd_io	Vss_io	Vss_core	MSCLK			SD4	IRQ9	SD2	SD6	LA21	SD11	14
VCC3V	Vdd_io	Vss_io	Vss_core	KBCLK_KBRST_			IOCHCK_	SA2	SBHE_	LA22	SA5	DACK_2	13
VCC3V	Vss_io	Vdd_core	Vss_core	Vss_core			SA3	DRQ0	Vss_core	SD8	MEMW_	SA1	12
VCC3V	Vss_io	Vdd_core	Vss_core	Vss_core			DRQ5	SA4	Vss_core	DACK_6	SD10	DRQ6	11
VCC3V	Vdd_io	Vss_io	Vdd_core	Vdd_core			SMEMR_	DACK_5	Vss_core	SD13	SD9	SYSCLK	10
VCCK	Vdd_io	Vss_io	Vss_io	Vss_io			SA19	SA17	DRQ2	DACK_0	LA17	MEMR_	9
							SMEMW_	AEN	IRQ12	0WS_	IOCHRDY_	OSC14M	8
							SD0	SA10	SA8	IRQ5	IRQ7	DACK_7	7
CTS4_/SIOW_	DCD4_/SA2	DSR4_/SCS1_	RTS4_/SINT	DTR4_/SA0	Vss_io	SD7	Vdd_io	Vss_core	Vdd_core	SA7	IRQ10	REFRESH_	6
PDD12	PDD2	PDD1	PDD11	RI4/SA1	Vss_io	DRQ3	Vdd_io	Vss_core	Vdd_core	SA18	SA9	TC	5
PDD10	PDD5	PDD6	PDD7	Vss_core	Vdd_io	SD5	Vdd_io	Vss_core	Vdd_core	DRQ1	BALE	IRQ15	4
PIORDY	PIOW_	PDD0	PDD8	Vdd_io	Vdd_io	DACK_3	SA16	DACK_1	SA0	SA11	IOCS16_	IRQ11	3
PA1	PA0	PCS0_	PDD4	PDD15	PDD14	IOW_	SA12	SA14	SA6	SA13	IRQ3	IRQ14	2
PDD13	PDRQ	PDACK_	PCBLID_	PIOR_	PCS1_	SD1	SA15	RSET_DRV	IRQ6	IRQ4	MEMCS16_	NC	1
P	R	T	U	V	W	Y	AA	AB	AC	AD	AE	AF	

2.4 EVA-X4300 Power Requirement

DC Volt	Tolerance	Ripple & Noise	Max. Current
3.3V for VCCP	± 0.30 V	< 100 mV	185 mA
3.3V for VCCO 1.8V for VCCO	± 0.15 V ± 0.09 V	< 50 mA	< 245 mA

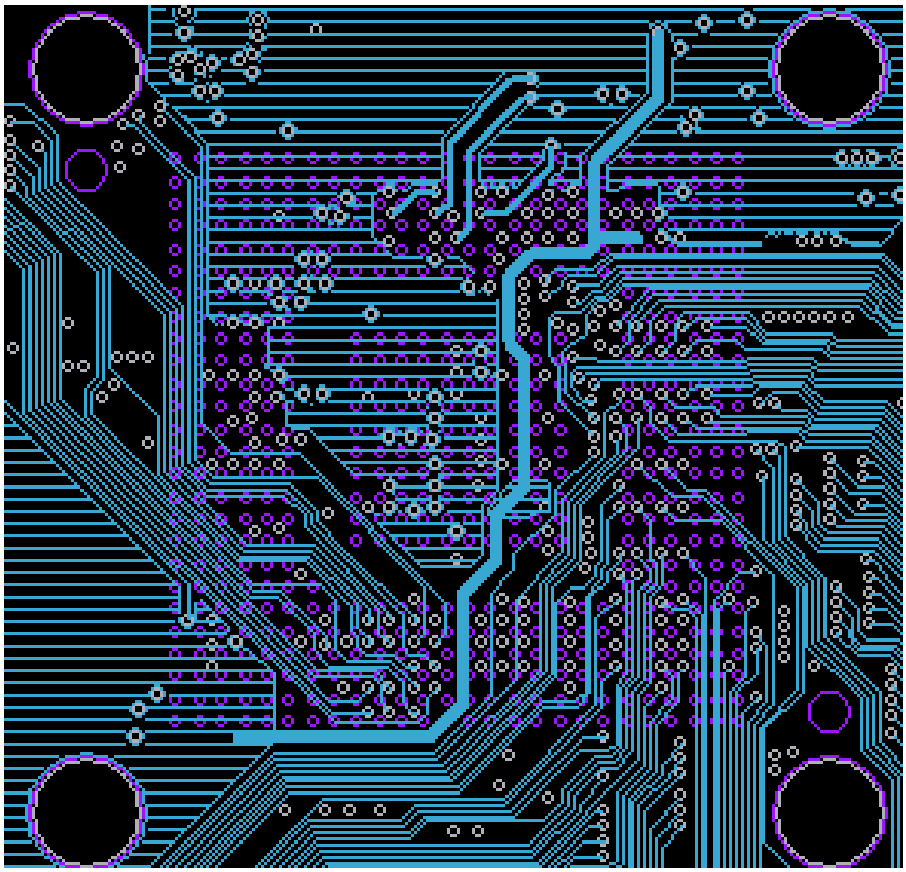
DC Volt	Min. Volt	Max. Volt	Ripple & Noise	Max. Current
1.3V for VCCK	1.30 V	1.40 V	< 50 mA	240 mA

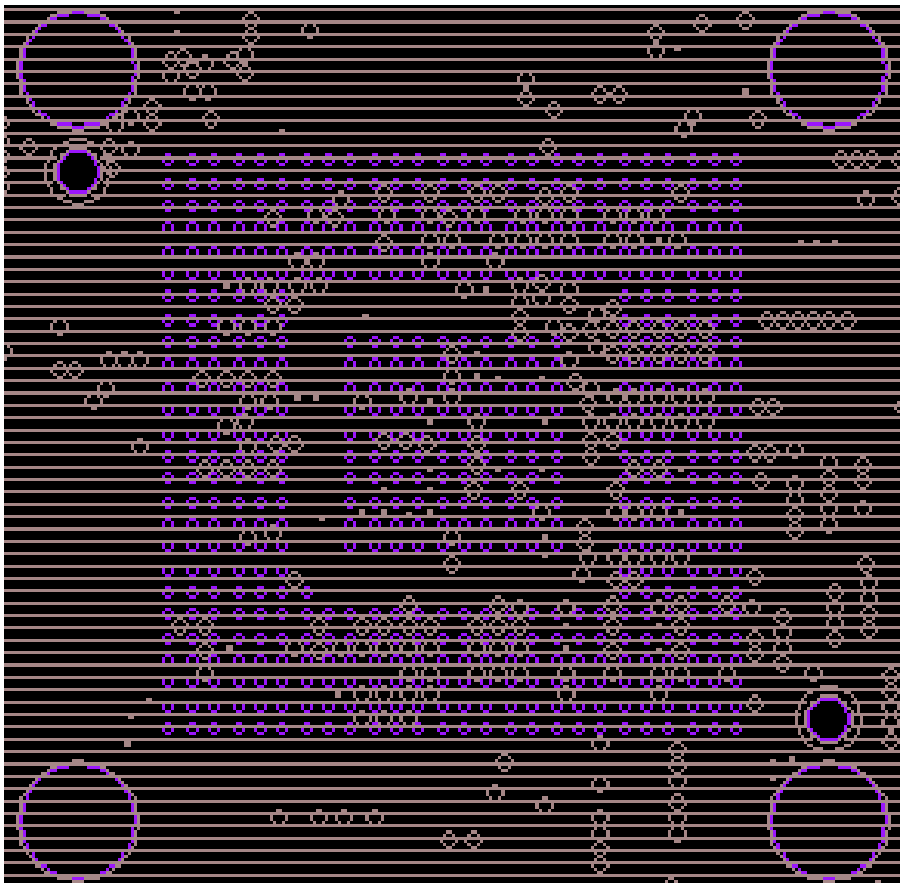
Note! *CPU clock 300 MHz, DDR2 clock 133 MHz and PCI clock 33 MHz*

2.5 General Layout Rule

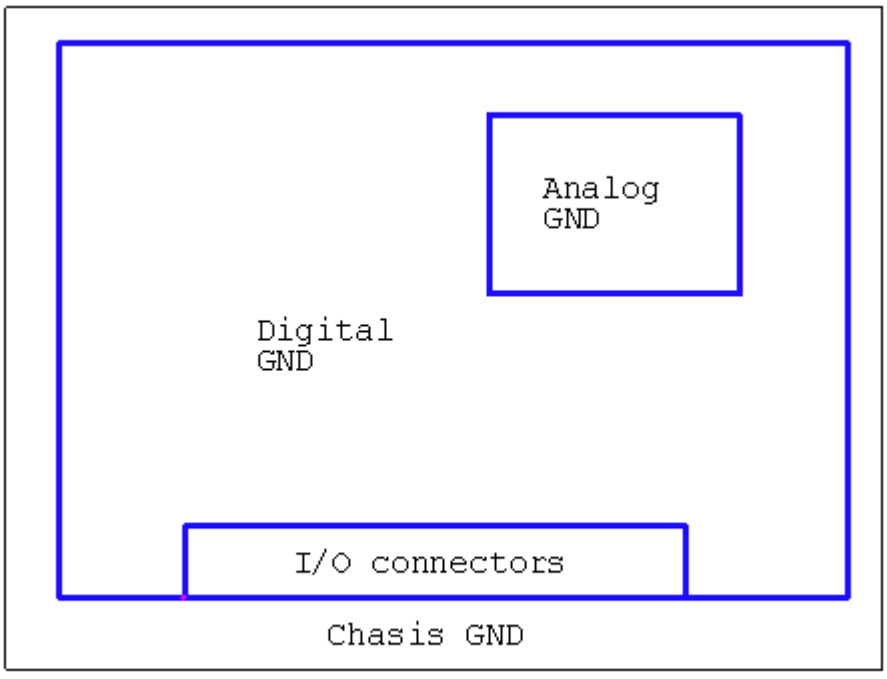
1. Keep all traces as SHORT as possible.
2. **Decoupling** is to remove the RF energy injected into the power distribution network from high-speed switching devices.
3. **Bypassing** is to divert unwanted common-mode RF noise from components or cables coupling from one area to another.
4. Place 0.01 µF~0.1 µF decoupling capacitors across each pair of VCC and GND pins. Use the shortest and thickest trace between decoupling capacitor and the VCC/GND pins.
5. Bulk capacitors ensure that a sufficient amount of DC voltage and current is available for digital components. At least one bulk capacitor should be located:
   - Every VLSI device
   - Power connector(s)
   - Daughter card slots
   - Furthest location from the power connector
   - Clock generation circuitry
6. The voltage rating of the bulk capacitors should be 50% higher than the actual voltage level to prevent self-destruction and/ or voltage surge.
7. For decoupling of DDR2 interface, please check the DDR2 section for more details.
8. In digital circuits, conductors may be treated as transmission lines if the propagation time (T_{delay}) is equal to or greater than the pulse transition time (T_{rise} or T_{fall}).
9. The propagation speed on typical PCB is about 170 pS/inch (50 Ω Stripline), 150pS/inch (50 Ω Microstrip).
10. There are 4 properties affect the performance of the transmission line:
    - Impedance: reflection/ distortion
    - Time delay
    - High-Frequency Loss: limit signal bandwidth and transmission distance
    - Crosstalk: coupling

11. It is recommended to add serial damping resistors to all high-speed signals, especially the clock signals, to reduce high frequency energy and EMI. **These resistors should be placed as close to the driving source as possible**.
12. The clock signals should be the first routed trace in any PCB design. Adjacently routed (guard) ground traces provide shielding and signal return path. These ground shielding traces should be connect to the ground plane by vias at both ends. It is best to run all clock signals on the signal plane above a solid ground plane (on a multi-layer board).
13. If clock/ high-speed signals must make a layer jump, route ground trace adjacent to the signals and connect both ends of the ground trace to the GND plane, to form a RF return path of the signals.
14. The split power planes may cause serious EMI and signal integrity problems for high-speed signals which run adjacent to the power planes. To minimize these problems, some decoupling capacitors should be placed between these split power planes, to enhance RF return current path.
15. Any unused area of the top and bottom signal layers of the PCB can be filled with copper that is connected to the ground plane through stitches of vias.
16. Example for BGA Power/ Ground routing:





17. Every I/O connector must be isolated from the digital ground and power planes. A clean or quiet ground must be located at the point where interconnects leave the system. Connect (“Bridge”) those grounds with only one connection. Only those signals required for operation or interconnect can run into the isolated area.



Note: the interconnects between GNDs were not shown.

18. Ensure that any signal passing between those sections runs ONLY through the “bridge”, and run the signals on a layer adjacent to the bridge to maintain RF return path.
19. If analog or digital power is not required in the isolated area, the unused power plane can be redefined as a second ground plane, referenced to the main ground plane by stitches of vias within the isolated area.
20. Connect system Power-on and H/W Reset to “Power-good” input of EVA-X4300, then use “/PCI_RST” output of EVA-X4300 to reset ALL peripherals.
21. The power-on-strap pins are combined with the memory address bus, which belong to DRAM power category. So the pull-high resistors should connect to the same supply (VCCO) of DRAM interface. That is:

DRAM type	Supply (V)	Note
SDR	3.3	
DDR2	1.8	

For detail of power-on-straps function, please refer to the EVA-X4300 data sheet.

22. Do follow the power-on sequence to prevent excessive current from the power supplies during power-up and power-down periods:
    - Power-up core supply (VDD_CORE), and then power-up the I/O supply (DVDD).
    - Power-down I/O supply (DVDD), and then power-down the core supply (VDD_CORE).
23. Recommend Termination:

Signal	Zs (Ω)	Termination(Ω)	Trace impedance (Ω)
Clocks 24M	26	22	60
PCI clock	7.2	22	60
PCI signal	13.2	51	60
DDR Clocks	19	33	90, Differential
DDR signal	19	33	60

24. Recommended PCB stack up:

a. 4 layers

Layer	Type	Description	Material/ Thickness
1	Signal	Top Routing	Copper/ 0.5~1oz
FR4/ 5mil			
2	Plane	**Ground**	Copper/ 1oz
FR4/ 40mil			
3	Plane	Power	Copper/ 1oz
FR4/ 5mil			
4	Signal	Bottom Routing	Copper/ 0.5~1oz

b. 6 layers

Layer	Type	Description	Material/ Thickness
1	Signal	Top Routing	Copper/ 0.5~1oz
FR4/ 5mil			
2	Plane	**Ground**	Copper/ 1oz
FR4/ 5mil			
3	Plane	**Power**	Copper/ 1oz
FR4/ 40mil			
4	Signal	Internal Routing	Copper/ 1oz
FR4/ 5mil			
5	Plane	**Ground**	Copper/ 1oz
FR4/ 5mil			
6	Signal	Bottom Routing	Copper/ 0.5~1oz

2.6 Crystal / External Oscillator

The EVA-X4300 requires a 14.318 MHz clock to generate all the internal and external clocks. Connect an external 14.318 MHz crystal (Parallel-resonant) between Xtal_I and Xtal_o pins to operate as a Pierce oscillator. The EVA-X4300 specification requires a frequency tolerance of ±30 parts pre million (PPM).

The other way is to use an external 14.318MHz oscillator, and directly route the oscillator is output into XTALi pin of EVA-X4300. A serial damping resistor and a bypass capacitor are recommended for EMI reduction. The XTALo pin should leave unconnected. The requirement of the clock is duty cycle should be between 40% ~60%.

Recommended Layout:

- Route the signal traces of the crystal to EVA-X4300 as SHORT as possible.
- When use external crystal with EVA-X4300 internal oscillation circuitry to work as clock oscillator, be sure to put all the related components close to EVA-X4300 chip. The ground area under the crystal circuitry should be physically insolated from system ground plane with only a small bridge between them for signals crossing. This isolation prevents noise located elsewhere on the PCB from corrupting the oscillator circuitry.
- The power plane under the crystal/ oscillator area should be void, when no use, or insolated with exactly the same pattern of ground plane.
- Do NOT run any signals under this area, for EMI and interference enhancement.

2.7 DDR2 Interface

1. It is assumed that the reader is familiar with the specification and the basic electrical operation of the DDR2 interface.
2. EVA-X4300 DDR2 interface substrate conductors' length for signal integrality:

EVA-X4300 Substrate Conductor Length	
Net Name	Length in microns
MA[0]	13283.31
MA[1]	12989.16
MA[2]	8745.03
MA[3]	10321.07
MA[4]	10173.85
MA[5]	11728.8
MA[6]	12466.29
MA[7]	11480.64
MA[8]	7638.44
MA[9]	10753.84
MA[10]	13659.62
MA[11]	9367.04
MA[12]	7999.4
MA[13]	8948.06
MD[0]	12939.2
MD[1]	12379.26
MD[2]	12218.38
MD[3]	12868.28
MD[4]	13327.88
MD[5]	11731.55
MD[6]	11817.28
MD[7]	11681.94
MD[8]	11494.07
MD[9]	10379.54
MD[10]	10577.86
MD[11]	11902.14
MD[12]	11610.42
MD[13]	10048.32
MD[14]	10981.68
MD[15]	12062.12
BA0	10484.24
BA1	9914.61
BA2	9542.11
/CAS	9668.98
/CS0	10088.02
/CS1	10430.52
DQM0	12436.31
DQM1	10402.02
DQS0	10560.44
DQS1	9752.32

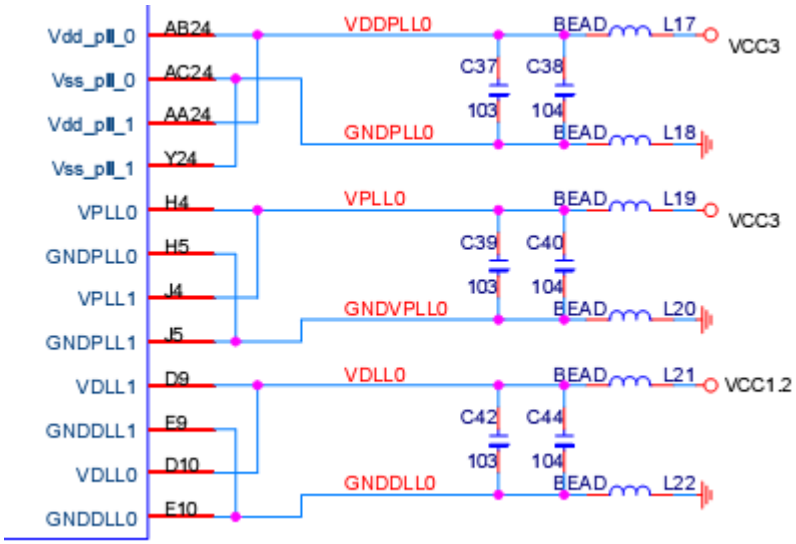
MCLK	6700.98
/MCLK	6678.9
/RAS	11307.98
/WE	10909.61

3. Supported DDR2 Devices:

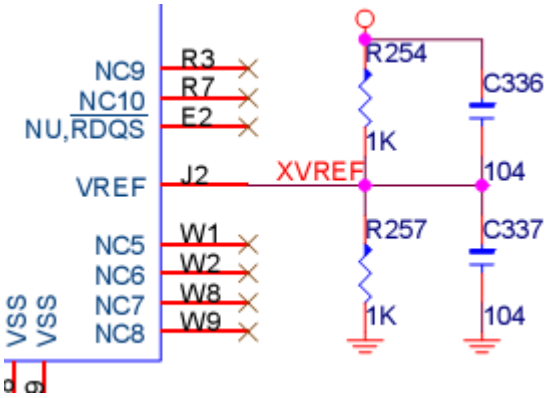
Vendor	Part Number	Note
Hynix	HY5PS121621C	84-ball
Hynix	HY5PS12821C	60-ball
SiS	DDRII6408-5C	60-ball
Elpida	EDE1108ABSE	68-ball

4. In order to meet the maximum interface speed (200 MHz/400 Mbps), memory device drive strength should be set 100% strength. This requires the series terminations to avoid excessive Under/Over-shoot. **We highly recommend reserve terminations for ALL high speed signals. It is much easier to remove terminations than adding them after the PCB has been found to fail EMI.**
5. It is critical that all signal routing layers have a ground reference plane, meaning that there is a full, contiguous ground plane next to every DDR2 routing layers. The purpose is to provide a path for return currents to minimize crosstalk and EMI.
6. The DDR2 devices should be placed as close to EVA-X4300 as possible. The distance between DDR2 devices and EVA-X4300 should less than 3 inches.
7. Other devices should be kept away to ensure other signals do not interfere with the DDR2 interface.
8. The 1.8V power partial plane (island) should encompass at least the entire DDR2 region.
9. All signals avoid crossing over an unrelated plane or different power plane. Six or more layers of PCB could eliminate these problems by routing DDR2 signals in the layer that is adjacent to the ground plane(s).
10. Rout traces with minimal layer transitions and minimize the total number of turns and vias.
11. Decoupling capacitors are critical to the reliable operation of the DDR2 interface. The decoupling capacitors should be **0402** size or smaller.
12. Bulk capacitors ensure that a sufficient amount of DC voltage and current is available for DDR2 devices. At least one bulk capacitor should be located for each DDR2 device.
13. DO NOT share the vias for decoupling capacitors, due to the inductance of the vias.
14. To minimize inductance, power vias should be as large as possible, but take good care of the inadvertently cut of the ground and power planes.
15. The PLL and DLL power supply pins draw small currents, but they are noise sensitive. Each supply should be filtered by **π-filter** networks. Use Ferrite Bead, NOT inductor.

i. EVA-X4300:



ii. DDR2 SDRAM:



16. VREF is not a high current supply, but it is important to keep it as quiet as possible with minimum inductance. For VREF, the minimum trace width is **20 mil**, and keeps overall trace length as short as possible.
17. The VREF divider resistors can be placed close to the DDR2 devices.
18. The decoupling capacitors for VREF are intended to reduce AC noise. Place one each at the divider and every VREF input of the DDR2(s).
19. Recommended Terminations for DDR2 interface:

Signal	Impedance (Ω)	Value (Ω)	Note
Clock±	Differential, 90	33	Near EVA-X4300
DQS±			Near DDR2
A[0..13], BA[0..2], Controls	60		Near EVA-X4300
DQM			
DQ[0..15]			Near DDR2
MD[0..15]			Near EVA-X4300

Note! *Termination value may have to be adjusted according to manufacturing condition.*

20. Recommended layout:

Trace length (include the substrate & PCB trace length) routing:

(a) The following signals of DDR2 SDRAM in each group must route near by in the same plane:

Group (1): LDQS, /LDQS, LDQM, DQ [0...7]

Group (2): UDQS, /UDQS, UDQM, DQ [8...15]

Group (3): MA [0...13], BA [0...2], /RAS, /CAS, /WE, MCLK, /MCLK

i. The intra-signals trace length in each group better keeps the same, otherwise keeps the mismatch less than 250 mils.

ii. The mismatch trace length between the inter-groups must be less than 400 mils.

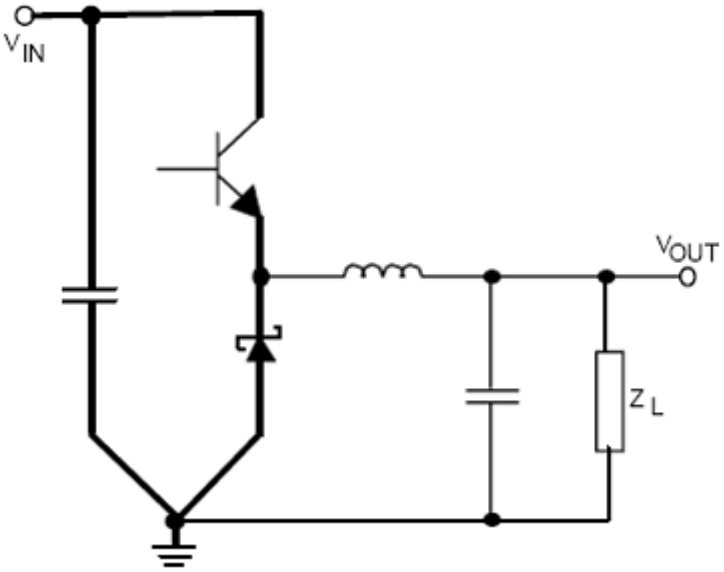
(b) All signals avoid crossing over an unrelated plane or different power plane.

(c) Route traces with minimal layer transitions and minimize the total number of turns & vias.

Signal	Width/ Spacing/ Isolation	Maximum Length	Note
Data Strobe	5 mil/ >20 mil/ 20 mil	2 inches	
Data Mask	5 mil/ >5 mil/ 15 mil		
Data Bus	5 mil/ >5 mil/ 5 mil		
Clock	5 mil/ >20 mil/ 20 mil	3 inches	The mismatch of the differential pair is ±10 mil
Address	5 mil/ >5 mil/ 5 mil		
Command			
VREF	Width > 20 mil	Kept to a minimum	
Power/ GND			

2.8 Switching Power

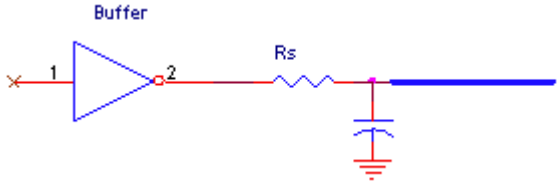
1. Please check section 2.3 and DDR2 data sheet for the detailed power requirements of the DDR2 system.
2. In a step-down switching regulator, the input bypass capacitor, the main power switch and the freewheeling diode carry discontinuous currents with high dt/di. For jitter-free operation, the size of the loop formed by these components should be minimized.



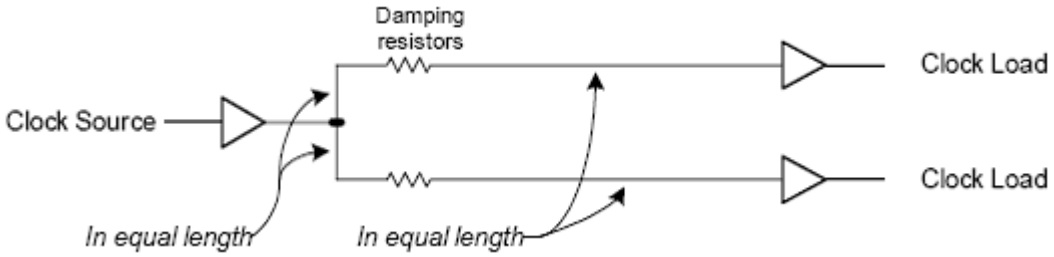
3. The input bypass capacitors should be placed close to the VIN pins. Shortening the traces of the SW node reduces the parasitic trace inductance at these nodes. This not only reduces EMI but also decreases switching voltage spikes at these nodes.
4. Some switching devices have exposed pad which should be soldered to a large analog ground plane, as the analog ground copper acts as a heat sink. To ensure proper adhesion to the ground plane, avoid using vias directly under the device.
5. For more detailed design information, please refer to the switching regulatorís data sheet.

2.9 PCI Interface

1. **We highly recommend reserve terminations for ALL PCI signals. It is much easier to remove terminations than adding them after the PCB has been found to fail EMI.**
2. The trace length for all PCI signals must be limited to 7 inches.
3. The trace length for PCI Clocks must be short for on-board PCI devices to minimize the clock skew.
4. The PCI clock traces should be parallel to their reference plane, usually ground planes. That means the clock traces should be right beneath or on top of their reference plane.
5. There is NO board impedance specified in the PCI bus specification. We recommend the trace impedance to be **60 Ω ± 10%**, with a **trace width/ spacing** design of **5 mil/10 mil**.
6. Add serial termination resistor and bypassing capacitor (tens of pF) to PCI clock signals to match the trace impedance and enhance EMI.



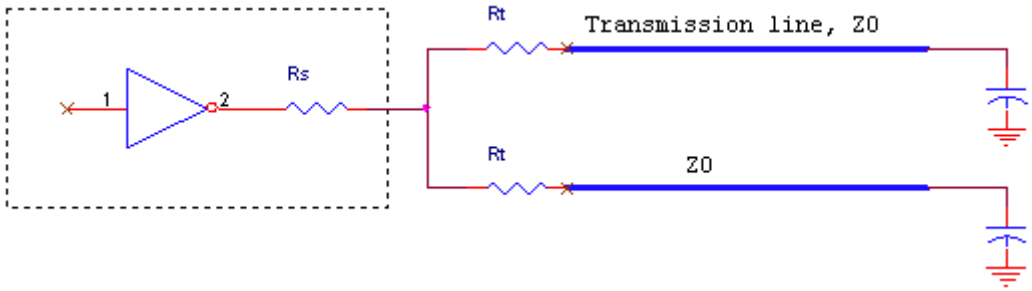
7. Avoid running PCI clocks in parallel with other signals for a long distance, for interference and coupling.
8. The user may need to drive two or more source-terminated clock lines with single output. However, the following condition must be achieved:
   i. The clock traces length must be as equal as possible, to guarantee the arriving time of the reflected pulses.
   ii. The loads must be balanced, to guarantee the same shape of the reflected pulses.



iii. The termination value must be calculated according to:

$$R_t = Z_0 - R_s * N$$

R_t = termination resistor, Ω
R_s = output impedance of the driver
Z_0 = line impedance
N = number of lines

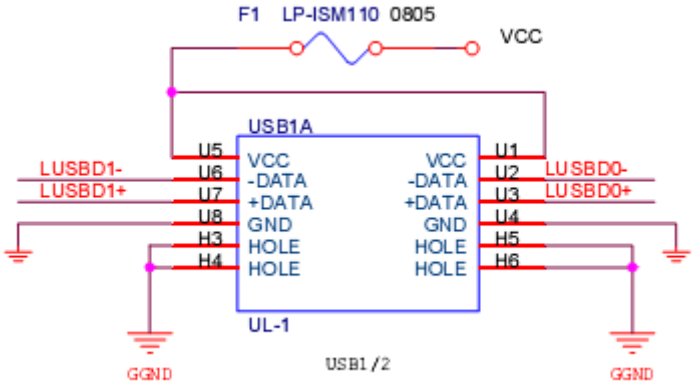


2.10 USB 2.0

1. **Route the Hi-Speed USB differential pairs over continuous ground or power planes. Avoid crossing anti-etch areas and any breaks in the internal planes (plane splits).**
2. Avoid placing a series of vias near the DP and DM lines, as these will create “break areas” in the ground plane below.
3. Avoid routing the USB differential pairs near I/O connectors, signal headers, crystals/ oscillators, magnetic and power connectors.
4. Maintain parallelism between USB differential signals, with the trace spacing needed to achieve **90 Ω differential impedance**.
5. We recommend the trace **Width/ Spacing/ Isolation** of USB differential pairs to be **8 mil/ 8 mil/ >20 mil**. And the mismatch of the differential pairs should be **less than ±70 mil**. These values may vary depending on the actual PCB parameters.
6. The maximum trace **length** of USB differential pairs should be **less than 2”**.

7. The common-mode choke used (**if really necessary**) on the DP and DM lines must be placed as close as possible to the USB connector and must have Z_{com}< 8 Ω @ 100 MHz and Z_{diff} < 300 Ω @ 100 MHz.
8. The analog power pins of EVA-X4300, AVDD0/1/2/3 and AVDDPLL0/1, need to be properly filtered for USB performance.

9. It is recommended to connect the analog ground pins of EVA-X4300, AVSS0/1/2/3 and AVSSPLL0/1, to an isolated (quiet) analog ground plane, which is connected to the digital ground plane with a single “Bridge” or Ferrite Bead. Please refer to section 2.4 for more details.
10. Place the external resistors of REXT0/1, pin U26 & P26, as close to the EVA-X4300 as possible. And connect another ends of these external resistors to the isolated analog ground plane described above.
11. Do not run any high-speed signal close to the external resistors of REXT0/1 to keep from interference and coupling.
12. Provide a good path from the USB connector shell to the chassis ground.
13. Maintain the maximum possible distance between Hi-Speed USB differential pairs, high-speed or lows-peed clock, and non-periodic signals. The minimum recommended distances are as follows:
    - 20 mils between the DP and DM traces and low-speed non-periodic signal traces.
    - 50 mils between the DP and DM traces and clock/high-speed periodic signal traces.
    - 20 mils between two pairs of the DP and DM traces.
14. USB data lines must be routed as “**critical signals**”. Locate the USB connector close to EVA-X4300. The DP and DM signals in a pair must be routed in parallel to each other. Do not route these traces near high frequency signals. Guard ground traces on each side of the signal pair can minimize the induced common mode noise.
15. Ferrite beads and decoupling capacitor placed on VBUS are for EMI purposes, thus these components should be placed close to the USB connectors. A value of about 150 µF/10 V for the decoupling capacitor is recommended on each port.

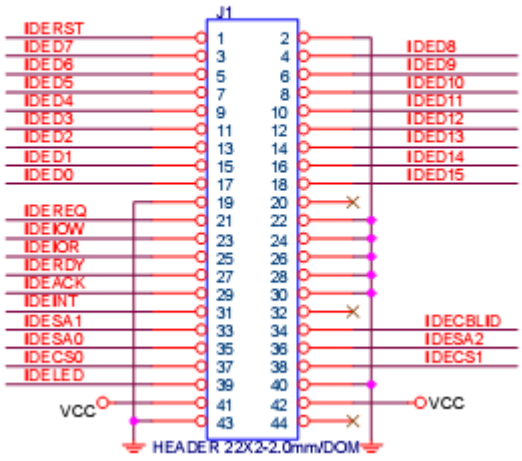


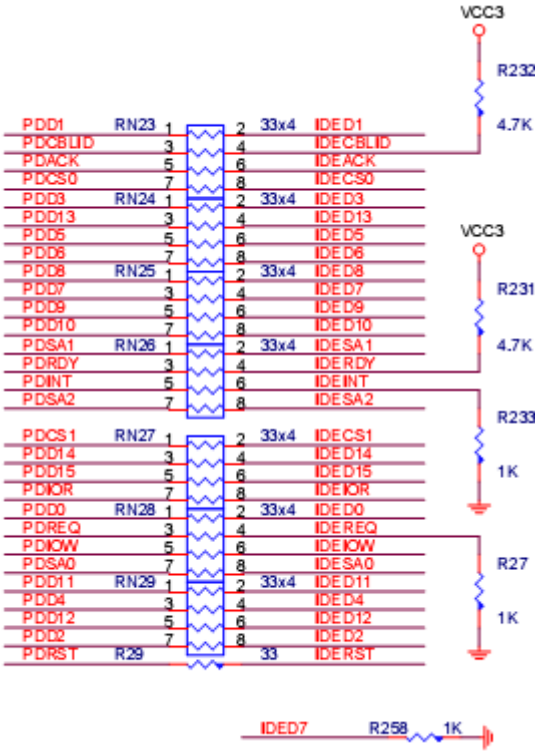
2.11 IDE

The trace-length and impedance-match must be considered for IDE signals to enhance the EMI and signal integrity:

1. The pull-up resistors on open-collector signals, such as IORDY, should be more than 1 KΩ.
2. All IDE signals need to be serial terminated. Place the termination resistors for A[0...2], CS[0..1], /IOR, /IOW and /DACK near EVA-X4300. Place the termination resistors for D[0..15], DRQ, IORDY, and IRQ near IDE connector.
3. The termination value should be optimized to compensate for transceiver and trace impedance to match the characteristic cable impedance.
4. The 28th pin of IDE connectors (CSEL) should be pulled low.
5. Signals in the same channel should have the same length, the mismatch must be less than 1”.
6. Reference circuitry:

IDE 44PIN/DOM





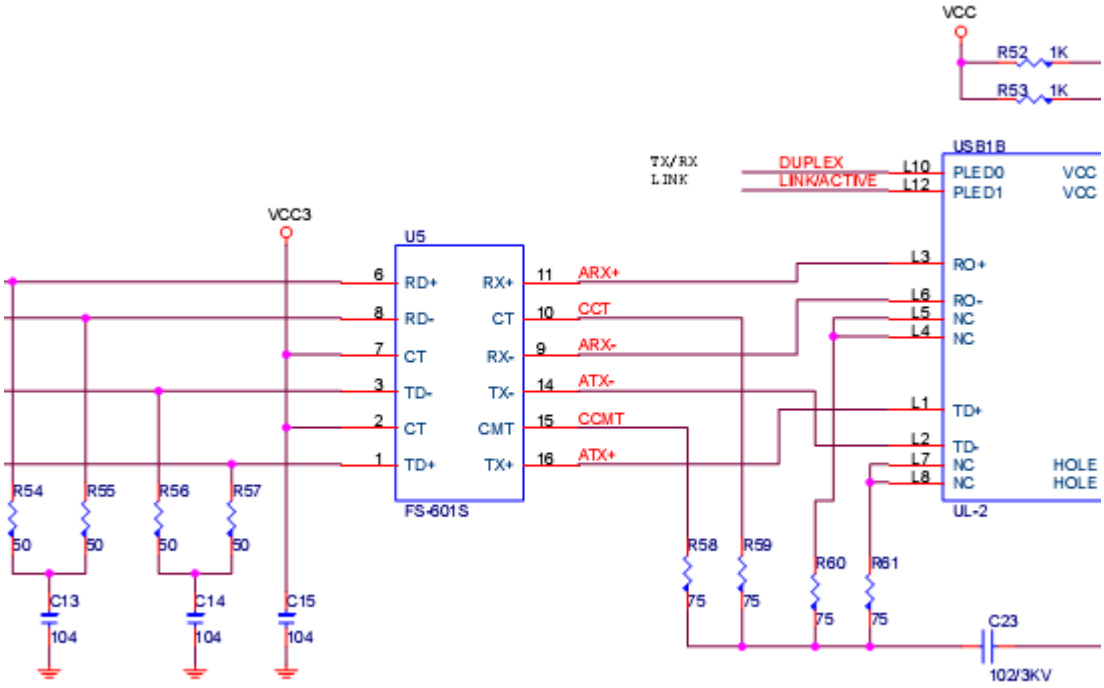
2.12 LPC Interface

A Low Pin Count (LPC) controller is integrated in EVA-X4300. The LPC interface is to replace ISA interface serving as a bus interface between the system processor and peripherals (e.g. LPC super I/O chip). Many of the signals are the same as signals found on the PCI interface. Data transfer on the LPC bus is serialized over a 4-bit bus.

Route the LPC signals with the PCI design rule.

2.13 10/100 LAN

1. Recommended Schematics:



2. Magnetics Specification:

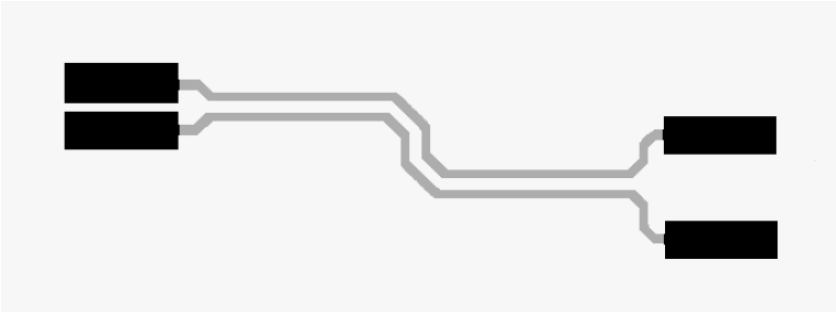
Parameters	Specification		Note
	Tx	Rx	
Turns Ratio	1CT:1 CT	1CT:1CT	
Inductance, μH	350	350	
Capacitance, pF	15	15	
DC Resistance, Ω	0.9	0.9	

3. Magnetics Selection Guide:

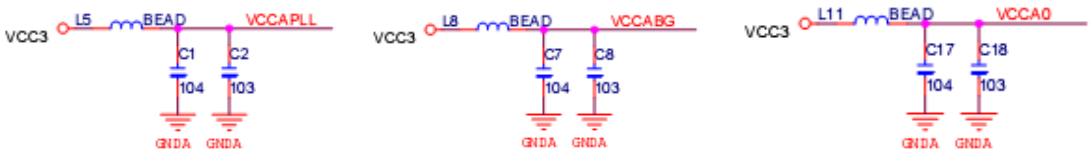
Manufacturer	Part Number	Note
Mingtek	**HN16005CG**	

4. Keep the distance between the EVA-X4300 and the RJ-45 connector short (under 4”).
5. Route the differential traces, TX±, RX±, TD± & RD±, for **100 Ω** differential impedance.

6. Keep the differential traces lengths equal to reduce signal skew.
7. Keep the differential traces close and symmetric to reduce noise.

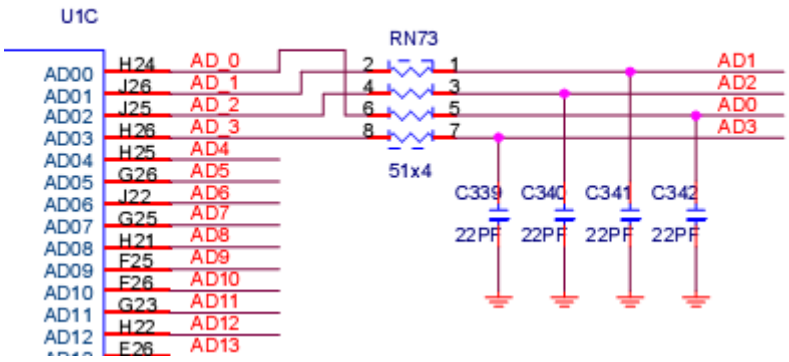


8. The parallel termination resistors of TD± should be placed near the magnetic. The parallel termination series resistors of RD± should be placed near the EVA-X4300.
9. It is recommended that route the differential traces turn with arcs, and avoid any vias and corners.
10. The traces of TX±/ TD± pair should be kept away from RX±/ RD± pair. It is best to place ground guard traces between these two pair of traces.
11. We recommend the trace **Width/Spacing/Isolation** of TX±/ TD± and RX±/ RD± pairs to be **8 mil/ 10 mil/ >20 mil**. And the mismatch of the differential pairs should be **less than ±100 mil**.
12. Do NOT run any digital trace close to and parallel to the differential pairs.
13. The RJ-45 and output side of transformer should reference to quiet ground plane (chassis ground) which is isolated from the ground plane of the input side of transformer and EVA-X4300. Connect this quiet ground plane to system ground with only one connection (bridge). Do NOT run any signal into this isolated area.
14. The moat to isolate the quiet ground should be at least **100 mil**.
15. Avoid laying power and ground planes underneath the magnetic to enhance EMI.
16. Provide a good path from the RJ-45 connector shell to the chassis ground.
17. The analog power pins of EVA-X4300, VCCAPLL, VCCABG, VCCA0/1 and AVDD33_0/1, need to be properly filtered for LAN performance.



18. It is recommended to connect the analog ground pins of EVA-X4300, VSSAPLL, VSSABG and VSSA0/1, to an isolated (quiet) analog ground plane, which is connected to the digital ground plane with a single “Bridge” or Ferrite Bead. Please refer to section 2.4 for more details.
19. **Place the external resistor of ISET, pin J24, as close to the EVA-X4300 as possible. And connect another end of the external resistor to the isolated analog ground plane described above.**
20. Do not run any high-speed signal close to the external resistor of ISET to keep from interference and coupling.

21. Avoid over-damping which affects LAN stability. It is recommend to place 51 ohm and 22 pF from PCI AD0~AD3.



Appendix A

References

A.1 References

1. “EVA-X4300 Technical Manual”
2. “PCI Local Bus Specification” rev. 2.2
3. “EMC and the Printed Circuit Board”, Mark I. Montrose
4. “High-speed Digital Design”, Howard W. Johnson, PH.D.
5. “Noise reduction Techniques in the Electronic System”, Henry W. Ott
6. “Printed Circuit Board design Techniques for EMC Compliance”, Mark I. Montrose.

Trusted ePlatform Services

ADVANTECH

www.advantech.com