

Hardware Development Guide for i.MX7Dual and 7Solo Applications Processors

1. About this Guide

This document's purpose is to help hardware engineers design and test their i.MX 7 series processor based designs. It provides information on board layout recommendations, design checklists to ensure first-pass success and ways to avoid board bring-up problems. It also provides information on board-level testing and simulation such as using BSDL for board-level testing, using the IBIS model for electrical integrity simulation and more.

Engineers are expected to have a working understanding of board layouts and terminology, IBIS modeling, BSDL testing and common board hardware terminology.

This guide is released along with relevant device-specific hardware documentation such as datasheets, reference manuals, and application notes available on nxp.com.

Contents

1.	About this Guide	1
2.	i.MX7DS Design Checklist.....	6
3.	i.MX7 Series Layout Recommendations.....	15
4.	Avoiding Board Bring-up Problems	34
5.	Ethernet Connections.....	38
6.	IBIS Model	39
7.	Using BSDL for Board-level Testing	46
8.	Revision History	48

1.1. Devices supported

This Hardware Developer's Guide currently supports the i.MX 7D, and the i.MX7S.

1.2. Essential reference

This guide is intended as a companion to the i.MX 7 series chip reference manuals and data sheets. For reflow profile and thermal limits during soldering, see application note [AN3298](#). These documents are available on [nxp.com](#).

1.3. Suggested reading

This section lists additional reading that provides background for the information in this manual as well as general information about the architecture.

1.3.1. General information

The following documentation provides useful information about the ARM processor architecture and computer architecture in general:

For information about the ARM Cortex-A7 processor see:

<http://www.arm.com/products/processors/cortex-a/cortex-a7.php>

- **Computer Architecture: A Quantitative Approach** (Fourth Edition) - by John L. Hennessy and David A. Patterson
- **Computer Organization and Design: The Hardware/Software Interface** (Second Edition), by David A. Patterson and John L. Hennessy

The following documentation provides useful information about high-speed board design:

- **Right the First Time- A Practical Handbook on High Speed PCB and System Design - Volumes I & II** - Lee W. Ritchey (Speeding Edge) - ISBN 0-9741936- 0-72
- **Signal and Power Integrity Simplified** (2nd Edition) - Eric Bogatin (Prentice Hall)- ISBN 0-13-703502-0
- **High Speed Digital Design- A Handbook of Black Magic** - Howard W. Johnson & Martin Graham (Prentice Hall) - ISBN 0-13-395724-1
- **High Speed Signal Propagation- Advanced Black Magic** - Howard W. Johnson & Martin Graham - (Prentice Hall) - ISBN 0-13-084408-X
- **High Speed Digital System Design- A handbook of Interconnect Theory and Practice** - Hall, Hall and McCall (Wiley Interscience 2000) - ISBN 0-36090-2
- **Signal Integrity Issues and Printed Circuit Design** - Doug Brooks (Prentice Hall) ISBN 0-13-141884-X
- **PCB Design for Real-World EMI Control** - Bruce R. Archambeault (Kluwer Academic Publishers Group) - ISBN 1-4020-7130-2
- **Digital Design for Interference Specifications-** A Practical Handbook for EMI Suppression - David L. Terrell & R. Kenneth Keenan (Newnes Publishing) - ISBN 0-7506-7282-X

- **Electromagnetic Compatibility Engineering**- Henry Ott (1st Edition - John Wiley and Sons) - ISBN 0-471-85068-3
- **Introduction to Electromagnetic Compatibility** - Clayton R. Paul (John Wiley and Sons) - ISBN 978-0-470-18930-6
- **Grounding & Shielding Techniques** - Ralph Morrison (5th Edition - John Wiley & Sons) - ISBN 0-471-24518-6
- **EMC for Product Engineers** - Tim Williams (Newnes Publishing) - ISBN 0-7506- 2466-3

1.4. Related documentation

NXP documentation is available from the sources listed on the back page of this guide.

Additional literature is published as new NXP products become available. For a current list of documentation, see nxp.com.

1.5. Conventions

This document uses the following notational conventions:

Courier	Used to indicate commands, command parameters, code examples, and file and directory names.
<i>Italics</i>	<i>Italics indicates</i> command or function parameters.
Bold	Function names are written in bold.
cleared/set	When a bit takes the value zero, it is said to be cleared; when it takes a value of one, it is said to be set.
mnemonics	Instruction mnemonics are shown in lowercase bold Book titles in text are set in italics.
sig_name	Internal signals are written in all lowercase.
<i>nnnn nnnnh</i>	Denotes hexadecimal number
0b	Denotes binary number
rA, rB	Instruction syntax used to identify a source GPR
rD	Instruction syntax used to identify a destination GPR
REG[FIELD]	Abbreviations for registers are shown in uppercase text. Specific bits, fields, or ranges appear in brackets. For example, MSR[LE] refers to the little-endian mode enable bit in the machine state register.
x	In some contexts, such as signal encodings, an unitalicized x indicates a don't care.
<i>x</i>	An italicized x indicates an alphanumeric variable.
<i>n, m</i>	An italicized n indicates a numeric variable.

In this guide, notation for all logical, bit-wise, arithmetic, comparison, and assignment operations follow C Language conventions.

1.6. Signal conventions

PWR_ON_RESET An overbar indicates that a signal is active when low *_b*, *_B* Alternate notation indicating an active-low signal *signal_name* Lowercase italics is used to indicate internal signals.

1.7. Acronyms and abbreviations

The following table defines the acronyms and abbreviations used in this document.

Table 1. Definitions and acronyms

Acronym	Definitions
ARM™	Advanced RISC machines processor architecture
BGA	Ball grid array package
BOM	Bill of materials
BSDL	Boundary scan description language
CAN	Flexible Controller Area Network peripheral
CCM	Clock Controller Module
CSI	MIPI camera serial interface
DDR	Dual data rate DRAM
DDR3	DDR3 DRAM
DDR3L	Low voltage DDR3 DRAM
DDR3U	Ultra-low voltage DDR3 DRAM
DRAM	Dynamic random access memory
ECSPI	Enhanced Configurable SPI peripheral
EIM	External Interface Module
ENET	10/100/1000 Mbps Ethernet MAC peripheral
EPIT	Enhanced Periodic Interrupt Timer peripheral
ESR	Equivalent series resistance (of a crystal)
GND	Ground
GPC	General Power Controller
GPIO	General-purpose input/output
HDCP	High-bandwidth Digital Content Protection
I ² C	Inter-integrated circuit interface
IBIS	Input Output buffer information specification
IOMUX	i.MX7 chip-level I/O multiplexing
JTAG	Joint Test Action Group
KPP	Keypad Port peripheral
LDB	LVDS Display bridge
LDO	Low drop-out regulator
LPCG	Low power clock gating
LPPDR2	Low-power DDR2 DRAM
LVDS	Low-voltage differential signaling
MLB	MediaLB 150 peripheral
MMDC	Multi-Mode DDR Controller
ODT	On-die termination
OTP	One-time programmable
PCB	Printed circuit board
PCIe	PCI Express
PCISig	Peripheral Component Interconnect Special Interest Group
PMIC	Power management integrated circuit
POR	Power-on reset
RAM	Random access memory
RGMII	Reduced Gigabit Media Independent Interface (Ethernet)
RMII	Reduced Media Independent Interface (Ethernet)
ROM	Read-only memory

Table 1. Definitions and acronyms

Acronym	Definitions
SDMA	Smart Direct Memory Access Controller
UART	Universal asynchronous receiver/transmitter
USB	Universal Serial Bus
USB OTG	USB On-the-go
USB 2.0	USB version 2.0 peripheral

2. i.MX7DS Design Checklist

This document provides a design checklist for the i.MX 7DS processor. The design checklist tables contain recommendations for optimal design. Where appropriate, the checklist tables also provide an explanation of the recommendation so that users have a greater understanding of why certain techniques are recommended. All supplemental tables referenced by the checklist appear in sections following the design checklist tables.

2.1. Design checklist tables

Table 2. DDR recommendation

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. Connect ZQPAD to an external 240 Ω 1 % resistor to GND.	This is a reference used during DRAM output buffer driver calibration.
	2. Connect DRAM_VREF to a source that is 50 % of the voltage value of NVCC_DRAM.	<ul style="list-style-type: none"> The user may tie DDR_VREF to a precision external resistor divider. Shunt DDR_VREF to GND with a closely-mounted 0.1 μF capacitor. Using resistors with recommended tolerances ensures the $\pm 2\%$ DDR_VREF tolerance per the DDR3 specification. The user can use a PMIC's tracking regulator as used on NXP reference designs.
	3. Connect DRAM_RESET to a 10 k Ω 5 % pulldown resistor to GND.	<ul style="list-style-type: none"> DDR3: DRAM_RESET should be pulled down to meet the JEDEC sequence until the controller is configured and starts driving. DRAM_RESET should be kept high when DDR3 enters self-refresh mode. LPDDR2: DRAM_RESET should be left unconnected. Some NXP reference designs use a 1 % resistor simply to consolidate the BOM. DRAM_RESET is an active-low signal.
	4. DRAM_SDCKE0 and DRAM_SDCKE1 require external pull-down resistors to GND for JEDEC compliance when using LPDDR2.	<ul style="list-style-type: none"> For LPDDR2: SDCKE[1:0] must be pulled down to meet the JEDEC sequence until the controller is configured and starts driving. NXP designs use 10 kΩ. For DDR3: SDCKE[1:0] pull-down is not required to meet JEDEC.
	5. DRAM_SDCKE0 and DRAM_SDCKE1 require external resistors (such as 10 k Ω) to GND to minimize current drain during deep sleep mode (DSM).	<p>During low-power self-refresh, the BSP programs pad control register GRP_CTLDS to 0x00000000. Therefore, DRAM_SDCKE0, DRAM_SDCKE1, and other associated GRP_CTLDS I/O are forced to the high-impedance state.</p> <p>Because DRAM_SDCKE0 and DRAM_SDCKE1 are forced to high-Z, external pull-down resistors are required to avoid floating outputs during standby. In NXP designs, 10 kΩ resistors are utilized for this purpose. Any other termination on the DRAM_SDCKE0 and DRAM_SDCKE1 lines (such as 50 ohms) should not be present; simulation should be performed to ensure CKE signal integrity.</p>

Table 3. LCD recommendations for developer's boot modes

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. When LCD boot signals are used as the system's LCD signals, other functions, or GPIO outputs after boot, use a passive resistor network to select the desired boot mode for development boards.	Because only resistors are used, LCD bus loads can cause current drain, leading to higher (false) supply current measurements. Each LCD boot signal should connect to a series resistor to isolate the bus from the resistors and/or switchers; see Figure 1 . Each configured LCD boot signal sees either a 14.7 k Ω pulldown or a 4.7 k Ω pullup. For each switch-enabled pulled-up signal, the supply is presented with a 10 k Ω current load. An alternate approach using buffers is implemented in the SABRE-SD development board design. Either of these implementations is acceptable.
	2. To reduce incorrect boot-up mode selections, do one of the following: <ul style="list-style-type: none"> • Use LCD boot interface lines only as processor outputs. Ensure LCD boot interface lines are not loaded down such that the level is interpreted as low during power-up, when the intent is to be a high level, or vice versa. • If an LCD boot signal must be configured as an input, isolate the LCD signal from the target driving source with one analog switch and apply the logic value with a second analog switch. Alternately, peripheral devices with three-state outputs may be used; ensure the output is high-impedance during the boot up interval. 	Using LCD boot interface lines as inputs may result in a wrong boot up due to the source overcoming the pull resistor value. A peripheral device may require the LCD signal to have an external or on-chip resistor to minimize signal floating. If the usage of the LCD boot signal affects the peripheral device, then an analog switch, open collector buffer, or equivalent should isolate the path. A pullup or pulldown resistor at the peripheral device may be required to maintain the desired logic level. Review the switch or device data sheet for operating specifications.
	3. The BOOT_CFG signals are required for proper functionality and operation and should not be left floating	See the "System Boot" chapter in your chip reference manual for the correct boot configuration. Note that an incorrect setting may result from an improper booting sequence.

Table 4. Boot mode input recommendations

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. For BOOT_MODE1 and BOOT_MODE0, use one of the following options to achieve logic 0: <ul style="list-style-type: none"> • Tie to GND through any size external resistor • Tie directly to GND • Float For logic 1, use one of the following: <ul style="list-style-type: none"> • Tie directly to the VDD_SNVS_IN rail • Tie to the VDD_SNVS_IN rail through an external resistor 10 kΩ. A value of 4.7 kΩ is preferred in high-noise environments. If switch control is desired, no external pulldown resistors are necessary. Simply connect SPST switches directly to the VDD_SNVS_IN rail. If desired, a 4.7 k Ω to 10 k Ω series resistor can be used when current drain is critical.	Boot inputs BOOT_MODE1 and BOOT_MODE0 each have on-chip pulldown devices with a nominal value of 100 k Ω , a projected minimum of 60 k Ω , and a projected maximum of 140 k Ω . Be aware that when these are logic high, current is drawn from the VDD_SNVS supply. In production, when on-chip fuses determine the boot configuration, both boot mode inputs can be no connects

Table 5. I²C recommendations

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. Verify the target I ² C interface clock rates.	The bus can only operate as fast as the slowest peripheral on the bus. If faster operation is required, move the slow devices to another I ² C port.
	2. Verify that the target I ² C address range is supported and does not conflict with other peripherals. If there is an unavoidable address conflict, move the offending device to another I ² C port.	These chips support up to four I ² C ports. If it is undesirable to move a conflicting device to another I ² C port, review the peripheral operation to see if it supports remapping the address.
	3. Do not place more than one set of pullup resistors on the I ² C lines.	This can result in excessive loading. Good design practice is to place one pair of pullups only.

Table 6. JTAG Recommendations

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. Do not use external pullup or pulldown resistors on JTAG_TDO.	JTAG_TDO is configured with an on-chip keeper circuit such that the floating condition is actively eliminated if an external pull resistor is not present. An external pull resistor on JTAG_TDO is detrimental. See Table 16 for a summary of the JTAG interface.
	2. Ensure that the on-chip pullup/pulldown configuration is followed if external resistors are used with JTAG signals (with the exception of JTAG_TDO). For example, do not use an external pulldown on an input that has an on-chip pullup.	External resistors can be used with all JTAG signals except JTAG_TDO, but they are not required. See Table 16 for a summary of the JTAG interface.
	3. JTAG_MOD may be referred to as SJC_MOD in some documents. Both names refer to the same signal. JTAG_MOD should be externally connected to GND for normal operation in a system. Termination to GND through an external pulldown resistor is allowed. Use ≤ 4.7 k Ω .	When JTAG_MOD is low, the JTAG interface is configured for common software debug, adding all the system TAPs to the chain. When JTAG_MOD is high, the JTAG interface is configured to a mode compliant with the IEEE 1149.1 standard.

Table 7. Power supply decoupling recommendations

Checkbox	Supply	0.22 μ F	2.2 μ F	22 μ F	10 μ F, 16 V	Notes
	VDD_ARM	3	—	1	—	—
	VDD_SOC	5	—	1	—	—
	NVCC_DRAM	4	—	1	—	—
	VDD_SNVS_IN	1	—	—	—	—
	USB_OTG1_VBUS	—	—	—	1	—
	USB_OTG2_VBUS	—	—	—	1	—
	VDDA_1P0_CAP	1	1	—	—	—
	VDD_LPSR_1P0_CAP	—	1	—	—	—
	VDD_LPSR_IN	—	—	—	—	Combine with other 1.8 V inputs
	VDD_SNVS_1P8_CAP	—	1	—	—	Connect these 5 together Place caps at MIPI phy Place caps at PCIe phy
	VDDD_1P0_CAP	—	—	—	—	
	VDD_MIPI_1P0	1	1	—	—	
	PCIE_VP	1	1	—	—	
	PCIE_VP_RX	—	—	—	—	
	PCIE_VP_TX	—	—	—	—	

Table 7. Power supply decoupling recommendations

Checkbox	Supply	0.22 μ F	2.2 μ F	22 μ F	10 μ F, 16 V	Notes
	VDDA_1P8_IN	1	1	—	—	Connect these 2 together
	VDDA_XTAL_1P8			—	—	
	VDD_TEMPSENSOR_1P8	1	1	—	—	Connect these 3 together Ferrite bead from same source as VDDA_1P8_IN
	VDDA_ADC1_1P8	—	—	—	—	
	VDDA_ADC2_1P8	—	—	—	—	
	PCC_EPDC_LCD_CAP	—	1	—	—	Single cap to GND
	PVCC_ENET_CAP	—	1	—	—	Connect these 2 together Short PVCC_ENET_CAP to PVCC_SAI_CAP
	PVCC_SAI_SD_CAP	—		—	—	
	PVCC_I2C_SPI_UART_CAP	—	1	—	—	Single cap to GND
	PVCC_GPIO_1P8_CAP	—	1	—	—	Single cap to GND
	FUSE_FSOURCE	—	—	—	—	Connect directly to a 1.8 V that is nearby and meets current requirements
	NVCC_GPIO1	1	—	—	—	—
	NVCC_GPIO2	1	—	—	—	—
	NVCC_SD1	1	—	—	—	—
	NVCC_SD2	1	—	—	—	—
	NVCC_SD3	1	—	—	—	—
	NVCC_ENET1	1	—	—	—	—
	NVCC_EPDC1	—	—	—	—	Use 470nF
	NVCC_EPDC2	—	—	—	—	Use 470nF
	NVCC_SAI	1	—	—	—	—
	NVCC_LCD	—	1	—	—	—
	NVCC_SPI	1	—	—	—	—
	NVCC_I2C	1	—	—	—	—
	NVCC_UART	1	—	—	—	—
	VDD_USB_OTG1_3P3_IN	1	1	—	—	—
	VDD_USB_OTG2_3P3_IN	1	1	—	—	—
	DRAM_VREF	—	—	—	—	Add cap divider and resistor divider. Or source from PMIC and follow PMIC recommendations.
	NVCC_DRAM_CKE	1	—	—	—	Can short to NVCC_DRAM

NOTE

Use the smallest capacitor package size allowed with your design rules.

For 0.22 μ F use 0201 or 0402 size. 0201 package preferred

For 2.2 μ F caps, 0402 package preferred.

For 22 μ F caps, 0603 package preferred.

Table 8. Power supply decoupling recommendations

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. Comply with the power-up sequence guidelines as described in the data sheet to guarantee reliable operation of the device.	Any deviation from these sequences may result in the following situations: <ul style="list-style-type: none"> • Excessive current during power-up phase • Prevention of the device from booting • Irreversible damage to the processor (worst-case scenario)
	2. Do not overload coin cell backup power rail VDD_SNVS_IN. Note that the following I/Os are associated with VDD_SNVS_IN; most inputs have on-chip pull resistors and do not require external resistors: <ul style="list-style-type: none"> • POR_B – on-chip pullup • ONOFF – on-chip pullup • BOOT_MODE0 – on-chip pulldown • BOOT_MODE1 – on-chip pulldown • TAMPER – on-chip pulldown • PMIC_STBY_REQ – push-pull output • PMIC_ON_REQ – push-pull output • TEST_MODE – on-chip pulldown 	NXP PMIC PMPF3000 VSNVS regulator is rated to supply 400 μ A output current under worst-case operating conditions. The VDD_SNVS_IN regulator can supply larger current in transient situations without damaging the regulator. Concerning i.MX7: <ul style="list-style-type: none"> • When VDD_SNVS_IN = VDD_HIGH_IN, SNVS domain current is drawn from both equally. • When VDD_HIGH_IN > VDD_SNVS_IN, VDD_HIGH_IN supplies all SNVS domain current and current flows into VDD_SNVS_IN to charge a coin cell battery. • When VDD_SNVS_IN > VDD_HIGH_IN, VDD_SNVS_IN supplies current to SNVS, and some current flows into VDD_HIGH_IN. Note: VDD_HIGH_IN must be valid (above the internal detector threshold, 2.4 V typ) for the current flow to occur. Thus, current flow only happens when VDD_HIGH_IN is powered to a level below VDD_SNVS_IN. If VDD_HIGH_IN is off or low, no extra current is drawn from VDD_SNVS_IN. The whole circuit assumes it is charging a coin cell and starts charging when VDD_HIGH_IN is valid. If you are driving VDD_SNVS_IN with a non-battery power source, it must be at the same level as VDD_HIGH_IN or current will flow between them. <ul style="list-style-type: none"> • When VDD_SNVS_IN is not powered by a battery, it is recommended that VDD_SNVS_IN = VDD_HIGH_IN. If VDD_SNVS_IN is tied to a battery, the battery eventually discharges to a value equal to that of VDD_HIGH_IN and never subsequently charges above VDD_HIGH_IN. The battery chemistry may add restrictions to VDD_HIGH_IN's voltage range. External charging components should be based on the battery manufacturer's specifications.
	3. Maximum ripple voltage requirements.	Common requirement for ripple noise should be less than 5 % Vp-p of supply voltage average value. Related power rails affected: all VDD_XXX_IN and VDD_XXX_CAP.
	4. If VDD_SNVS_IN is directly supplied by a coin cell, a schottky diode is required between VDD_HIGH_IN and VDD_SNVS_IN. The cathode is connected to VDD_SNVS_IN. Alternately, VDD_HIGH_IN and VDD_SNVS_IN can be tied together if the real-time clock function	When no power is supplied to VDD_VSNVS_IN, the diode limits the voltage difference between the two on-chip SNVS power domains to approximately 0.3 V. The processor is designed to allow current flow between the two SNVS power domains proportional to the voltage difference.
	5. If boundary scan test (BSDL) will be used, the following supplies must be powered: PCIE_VP,	The boundary scan test scan chain runs through the PCIe PHY. If the PCIe PHY supplies are not powered,

Table 8. Power supply decoupling recommendations

Checkbox	Recommendation	Explanation/Supplemental recommendation
	PCIE_VPH, PCIE_VPTX	the scan chain will not function.

Table 9. Oscillator and clock recommendations

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. Precision 32.768 kHz oscillator Connect a crystal between RTC_XTALI and RTC_XTALO. Choose a crystal with a maximum of 100 k Ω ESR (equivalent series resistance) and follow the manufacturer's recommendation for loading capacitance.	The integrated oscillation amplifier has an on-chip self-biasing scheme, but is high-impedance (relatively weak) to minimize power consumption. Care must be taken to limit parasitic leakage from RTC_XTALI and RTC_XTALO to either power or ground (> 100 M Ω) as this negatively affects the amplifier bias and causes a reduction of startup margin. Use short traces between the crystal and the processor, with a ground plane under the crystal, load capacitors, and associated traces.
	2. External kilohertz source If feeding an external clock into the device, RTC_XTALI can be driven DC-coupled with RTC_XTALO floated or driven with a complimentary signal.	The voltage level of this driving clock should not exceed the voltage of VDD_SNVS_CAP and the frequency should be <100 kHz under typical conditions. Do not exceed VDD_SNVS_CAP or damage/malfunction may occur. The RTC_XTALI signal should not be driven if the VDD_SNVS_CAP supply is off. This can lead to damage or malfunction. For RTC_XTALI VIL and VIH voltage levels, see the latest i.MX 7 series datasheet available at nxp.com . Note that if this external clock is stopped, the internal ring oscillator starts automatically.

Table 10. Reset and ON/OFF recommendations

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. If the external SRC_POR_B signal is used to control the processor POR, then SRC_POR_B must be immediately asserted at power-up and remain asserted until the VDD_ARM_CAP and VDD_SOC_CAP supplies are stable. VDD_ARM_IN and VDD_SOC_IN may be applied in either order with no restrictions. In the absence of an external reset feeding the SRC_POR_B input, the internal POR module takes control.	A reset switch may be wired to the chip's POR_B, which is a cold-reset negative-logic input that resets all modules and logic in the IC. POR_B may be used in addition to internally generated power-on reset signal (logical AND, both internal and external signals are considered active low).
	2. For portable applications, the ONOFF input may be connected to an ON/OFF SPST push-button switch. On-chip debouncing is provided, and this input has an on-chip pullup. If not used, ONOFF should be a no connect.	A brief connection to GND in OFF mode causes the internal power management state machine to change state to ON. In ON mode, a brief connection to GND generates an interrupt (intended to be a software-controllable power-down). An approximate 5 second or more connection to GND causes a forced OFF.

Table 11. PCIe recommendations

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. Appropriate PCIe reference clock generator is suggested. i.MX differential clock is not compliance with PCIe standard.	—
	2. The differential transmitter must be ac coupled. Use a 0.1 μ F-series capacitor on PCIE_TXP and a second 0.1 μ F on PCIE_TXM.	To ensure PCIe specification compliance, ac coupling is required at each transmitter. The receiver must be DC coupled.
	3. PCIe Jitter compliance NXP recommends including an external clock source that meets the PCIe jitter specification until the i.MX 7DS PCIe jitter compliance can be assessed.	—

Table 12. USB recommendations

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. USB OTG To comply with the USB OTG specification, the VBUS supply on the OTG connector should default to <i>off</i> when the boards power up.	The processor should turn VBUS <i>on</i> as required.
	2. VDD_USB_CAP Adopters should consider adding a zener diode or clamp to VDDUSB_CAP, in parallel with the required capacitors. Clamp at approximately 3.3-3.5 V.	This aids the absorption of reflection-induced transients when long (~5 meters) USB cables are used. Also, this is good practice if the product will be interfaced to a test fixture via USB.

Table 13. Reference resistor recommendations

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. PCIE_REXT – Connect an external 4.7 K Ω 1% resistor to GND.	The impedance calibration process requires connection of this reference resistor. If PCIe is unused, the reference resistor may be populated if desired for manufacturability purposes, or left no-connect for cost savings.

Table 14. Miscellaneous recommendations

Checkbox	Recommendation	Explanation/Supplemental recommendation
	1. The TEST_MODE input is internally connected to an on-chip pulldown device. The user must tie this signal to GND.	This input is reserved for NXP manufacturing use.
	2. For termination of unused analog interfaces, see Datasheet for this part.	—
	3. GPANAIO must be a no connect. The user must leave this connection floating.	This output is reserved for NXP manufacturing use.
	4. NC contacts are no connect and should be floated.	Depending on the feature set, some versions of the IC may have NC contacts connected inside the BGA.

2.2. Bus isolation circuit

The following figure provides supporting information for [Table 3](#), recommendation # 1.

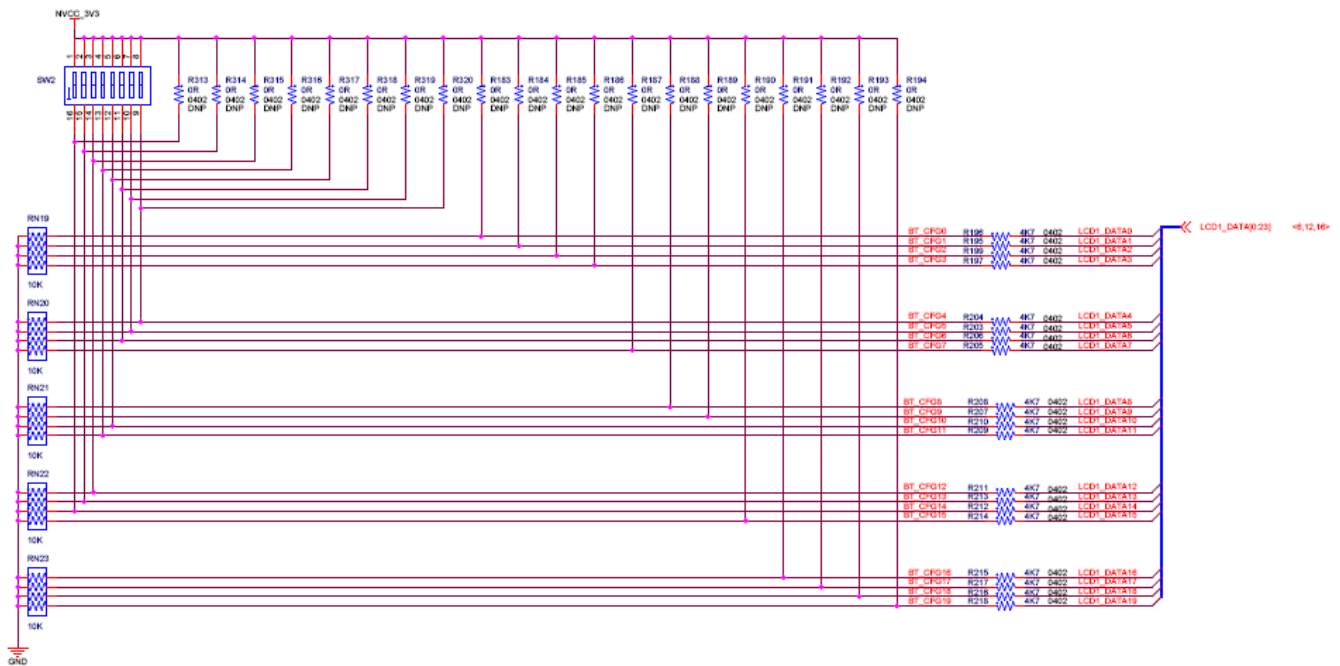


Figure 1. Boot configuration for development mode

2.3. DDR reference circuit

The following table is a resistor chart (See [Table 2](#) recommendation #2). The recommendations are appropriate for designs with DDR memory chips with a maximum Vref input current of 2 uA each.

Table 15. DDR Vref resistor sizing guideline

Number or DRAM with 2uA Vref input current	Resistor divider value (2 resistors)
2	$\leq 1.2 \text{ k}\Omega \text{ } 1 \%$
2	$\leq 1.254 \text{ k}\Omega \text{ } 0.5 \%$
2	$\leq 2.32 \text{ k}\Omega \text{ } 0.1 \%$

2.4. JTAG signal termination

The following table is a JTAG termination chart (see recommendations in [Table 6](#)).

Table 16. JTAG interface summary

JTAG signal	I/O type	On-chip termination	External termination
JTAG_TCK	Input	47 K Ω pullup	Not required: can use 10K Ω pullup
JTAG_TMS	Input	47 K Ω pullup	Not required: can use 10K Ω pullup
JTAG_TDI	Input	47 K Ω pullup	Not required: can use 10K Ω pullup
JTAG_TDO	3-state output	Keeper	Do Not use pullup or pulldown
JTAG_TRSTB	Input	47 K Ω pullup	Not required: can use 10K Ω pullup
JTAG_MOD	Input	100 K Ω pullup	Use 1K Ω pulldown or tie to GND

2.5. Oscillator tolerance

The following table provides 24 MHz oscillator tolerance guidelines (see [Table 9](#), recommendations #4 and #5). Because these are guidelines, the designer must verify all tolerances per the official specifications.

Table 17. 24 MHz crystal tolerance guidelines

Interface	Tolerance (\pm ppm)
Ethernet	50
USB 2.0	500
PCIe	300

2.6. Unused analog interfaces

For recommendations for unused analog interfaces see appropriate i.MX7 Datasheet section “3.2 Recommended connections for unused analog interfaces.”

3. i.MX7 Series Layout Recommendations

3.1. Introduction

This chapter provides recommendations to assist design engineers with the layout of an i.MX7 series-based system.

3.2. Basic design recommendations

The i.MX 7DS processor comes in multiple packages. For detailed information, see the i.MX 7 datasheets.

When using the Allegro tool, optimal practice is to use the footprint as created by NXP. When not using the Allegro tool, use the Allegro footprint export feature (supported by many tools). If export is not possible, create the footprint per the package mechanical dimensions outlined in the product data sheet.

Native Allegro layout and gerber files are available on nxp.com.

3.2.1. Placing decoupling capacitors

Place small decoupling and larger bulk capacitors on the bottom side of the PCB.

The 0201 decoupling and 0603 bulk capacitors should be mounted as close as possible to the power vias. The distance should be less than 50 mils. Additional bulk capacitors can be placed near the edge of the BGA via array. Placing the decoupling capacitors close to the power balls is critical to minimize inductance and ensure high-speed transient current demand by the processor.

A correct via size is critical for preserving adequate routing space. The recommended geometry for the via pads is: pad size 18 mils and drill 8 mils.

The following list provides the main recommendations for choosing the correct decoupling scheme:

- Place the largest capacitance in the smallest package that budget and manufacturing can support.
- For high speed bypassing, select the required capacitance with the smallest package (for example, 0.22 μ F and package 0201).
- Minimize trace length (inductance) to small caps.
- Series inductance cancels out capacitance.
- Tie caps to GND plane directly with a via.
- Place capacitors close to the power contact of the associate package designed from the schematic.
- A preferred BGA power decoupling design is the layout is available through nxp.com. Customers should use the NXP design strategy for power and decoupling.

3.3. Stack-up recommendations

High-speed design requires a good stack-up in order have the right impedance for the critical traces. The constraints for the trace width may depend on a number of factors, such as the board stack-up and

associated dielectric and copper thickness, required impedance, and required current (for power traces). The NXP reference design uses a minimum trace width of 3 mils for the DDR routing. The stack-up also determines the constraints for routing and spacing.

Consider the following when designing the stack-up and selecting the material for your board.

- Board stack-up is critical for high-speed signal quality.
- You must preplan impedance of critical traces
- High-speed signals must have reference planes on adjacent layers to minimize cross-talk.
- FSL reference design equals Isola 370HR.
- FSL validation boards equals Isola FR408.

The recommended stack-up is 8-layers, with the layer stack as shown in the following figure. The left-hand image shows the detail provided by NXP inside the fabrication detail as a part of the Gerber files. The right-and side shows the solution suggested by the PCB fabrication company for our requirements.

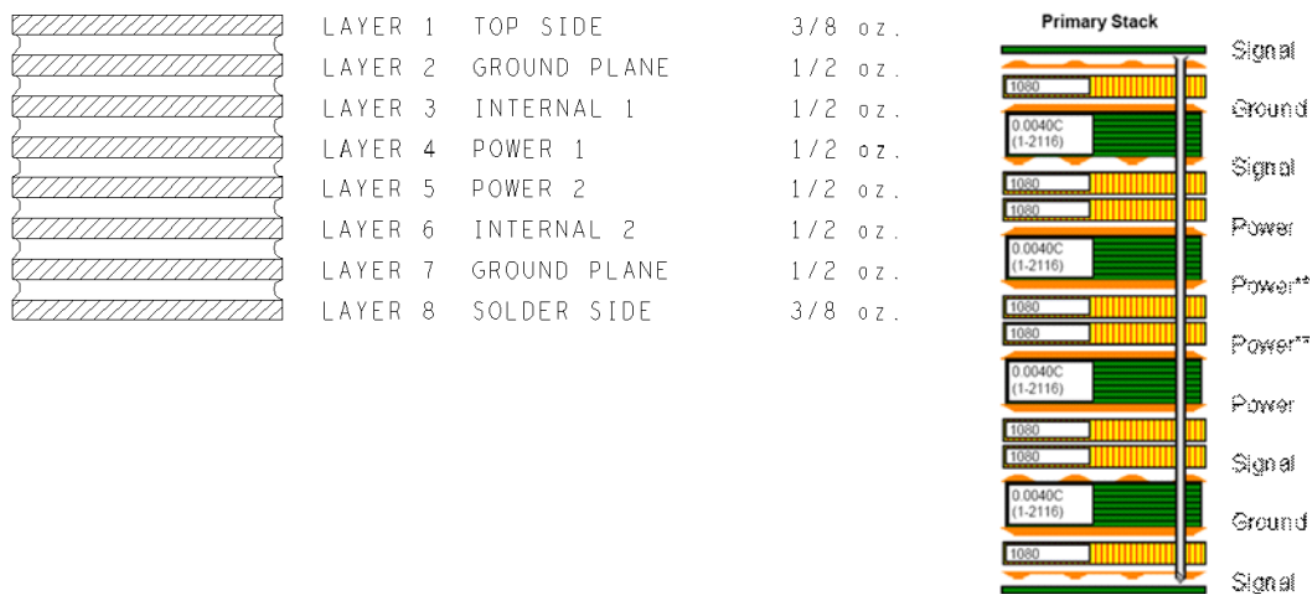


Figure 2. Layer stack SABRE SD board

The following table shows a working stack-up implementation:

Table 18. Stackup implementation

Layers	Single Ended		Differential					
	Trace width (Mils)	Impedance (Ωs)	Trace width (Mils)	Trace spacing 'Airgap' (Mils)	Impedance (Ωs)	Trace width (Mils)	Tracespacing 'Airgap' (Mils)	Impedance (Ωs)
TOP	4.7	50	4.3	5.8	90	3.7	5.3	100
INT1	4.5	50	4.2	5.8	90	3.8	5.2	
INT2	4.5	50	4.2	5.8	90	3.8	5.2	
BOT	4.7	50	4.3	5.7	90	3.7	533	100




Impedance Type	Layer	Design	Actual	Pitch	Plane	Target	Tol (ohms)	Predict
1  Surface MS	L1	0.00470	0.0047	-	-	50	5.0	49.96
	-	-	-	-	L2			
2  EC Microstrip	L1	-	0.0043	0.0100	-	90	9.0	90.62
	-	-	0.0043	-	L2			
3  EC Microstrip	L1	0.00370	0.0034	0.0090	-	100	10.0	98.88
	-	0.00370	0.0034	-	L2			

Figure 3. Example top layer impedance from PCB fabricator

3.4. DDR Connection information

The following figure show the block diagram from the reference design board for DDR3 interface with the i.MX 7DS.

The DDR interface is one of the most critical interfaces for chip routing. It must have the controlled impedance for the single ended traces be equal to 50 Ω and differential pairs be equal to 100 Ω .

The following figure shows the physical connection scheme for both top and bottom placement of the DDR chips, showing the final placement of the DDR3 memory and the decoupling capacitors. The blue figure shows the top layer and the red figure shows the bottom layer. It is very important to place the memory as close to the processor as possible to reduce trace capacitance and keep the propagation delay to the minimum. Follow the reference board layout as a guideline for memory placement and routing.

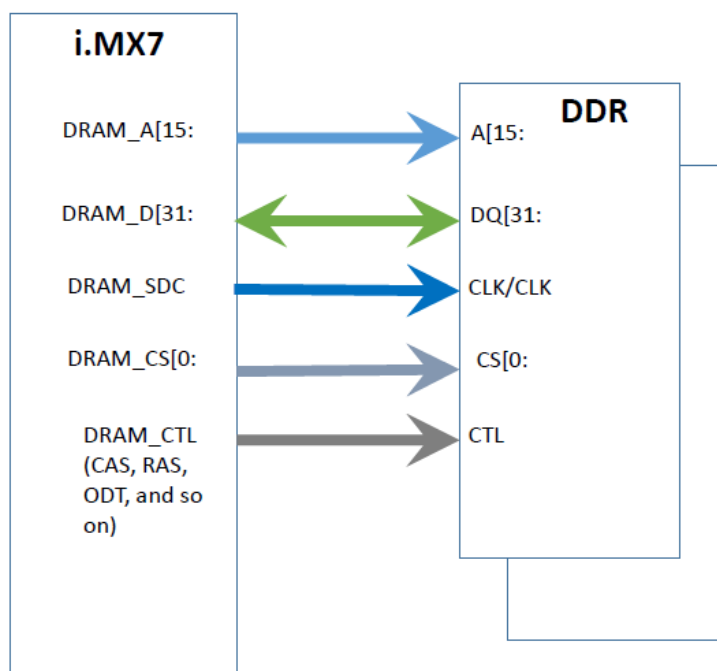


Figure 4. i.MX to DDR3 block diagram

3.4.1. DDR Routing Rules

The i.MX7D processor is not capable of automatically calibrating each DQS strobe to arrive at the DRAM in phase with the SDCLK signal. Therefore, Fly-By topologies are not recommended or supported for designs i.MX7D. Ideally, all signals should be routed at the same length. Nevertheless, it could be difficult because of the large number of connections in the tight space between the DDR and the processor. The following table explains the rules for routing the signals by the same length.

Table 19. LPDDR2/3 routing by the same length

Chip Signals	Group	Length		Recommendations
		Min	Max	
DRAM_SDCLK DRAM_SDCLK_B	Clock	Short as possible	1400mils	Match the signals ± 5 mils. (750 mil length is easily achievable)
DRAM_A[15:0] DRAM_SDBA[2:0] DRAM_RAS DRAM_CAS DRAM_SDWE	Address and Command	Clock (min) – 55 mils	Clock (min) + 55 mils	Best practice is to match the signals ± 55 mils.
DRAM_D[7:0] DRAM_DQM0 DRAM_SDQS0 DRAM_SDQS0_B	Byte Group 0			Best Practice: DQS strobe should have maximum length of Clock -10 mils Match the differential signals of DQS ± 5 mils. Match the signals of each byte group ± 55 mils to the strobe. Limit minimum DQS length to Clock (min) – 200 mils. If the DQS strobe is more than 200 mils shorter than Clock (min), consider manually adjusting each field of register DDR_PHY_LVL_CON0. Increment +1 for each 100 mils that the DQS trace is shorter than Clock (min).
DRAM_D[15:8] DRAM_DQM1 DRAM_SDQS1 DRAM_SDQS1_B	Byte Group 1			
DRAM_D[23:16] DRAM_DQM2 DRAM_SDQS2 DRAM_SDQS2_B	Byte Group 2			
DRAM_D[31:24] DRAM_DQM3 DRAM_SDQS3 DRAM_SDQS3_B	Byte Group 3			

Routing by byte group requires better control of the signals of each group. It is also more difficult for analysis and constraint settings. However, its advantage is that the constraint to match lengths can be applied to a smaller group of signals. This is often more achievable once the constraints are properly set.

The following table explains the rules for routing the signals by byte group.

Table 20. DDR3 Routing by byte group

Chip Signals	Group	Length		Recommendations
		Min	Max	
DRAM_SDCLK[1:0] DRAM_SDCLK_B[1:0]	Clock	Short as possible	1900 mils	Match the differential signals ± 5 mils.
DRAM_A[15:0] DRAM_SDBA[2:0] DRAM_RAS DRAM_CAS DRAM_SDWE	Address and Command	Clock (min) – 200	Clock (min)1	Match the signals ± 10 mils. Match each signal to within ± 5 mils when comparing IC1 to IC2 for dual chip memory
DRAM_D[7:0] DRAM_DQM0 DRAM_SDQS0 DRAM_SDQS0_B	Byte Group 1	—	—	Best Practice: DQS strobe should have maximum length of Clock -10 mils Match the differential signals of DQS ± 5 mils. Match the signals of each byte group ± 55 mils to the strobe. Limit minimum DQS length (lesser of): Clock (min) – 10 mils or 1000mils
DRAM_D[15:8] DRAM_DQM1 DRAM_SDQS1 DRAM_SDQS1_B	Byte Group 2	—	—	—
DRAM_D[23:16] DRAM_DQM2 DRAM_SDQS2 DRAM_SDQS2_B	Byte Group 3	—	—	—
DRAM_D[31:24] DRAM_DQM3 DRAM_SDQS3 DRAM_SDQS3_B	Byte Group 4	—	—	—

3.4.2. Two chip Topology routing examples

The figures in this section show examples for the routing of the 2 GB DDR3 memories. The following figures are a guideline of the T configuration routing with eight layers PCB.

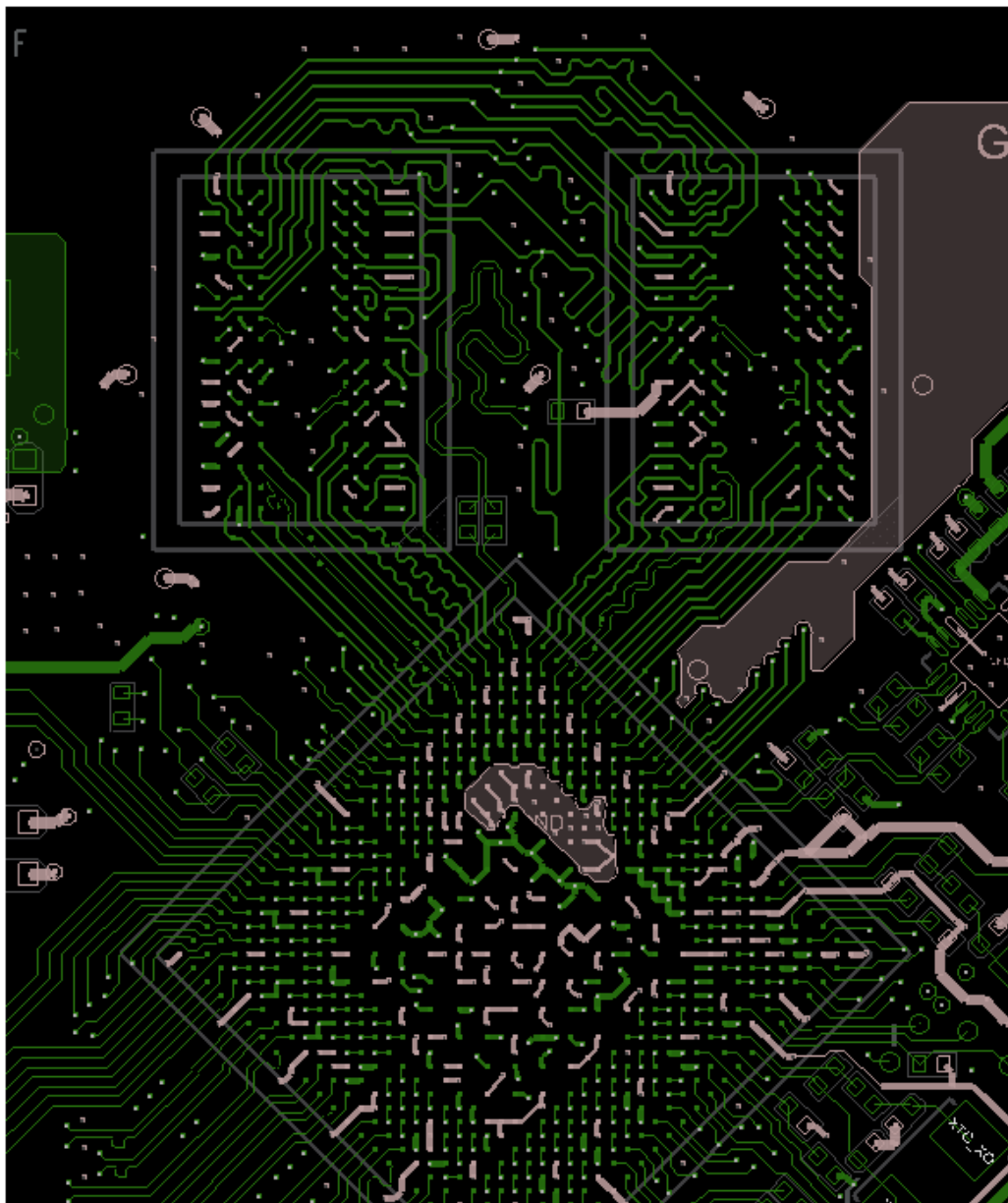


Figure 5. Top Layer



Figure 6. Signal layer 1



Figure 7. Signal layer 2

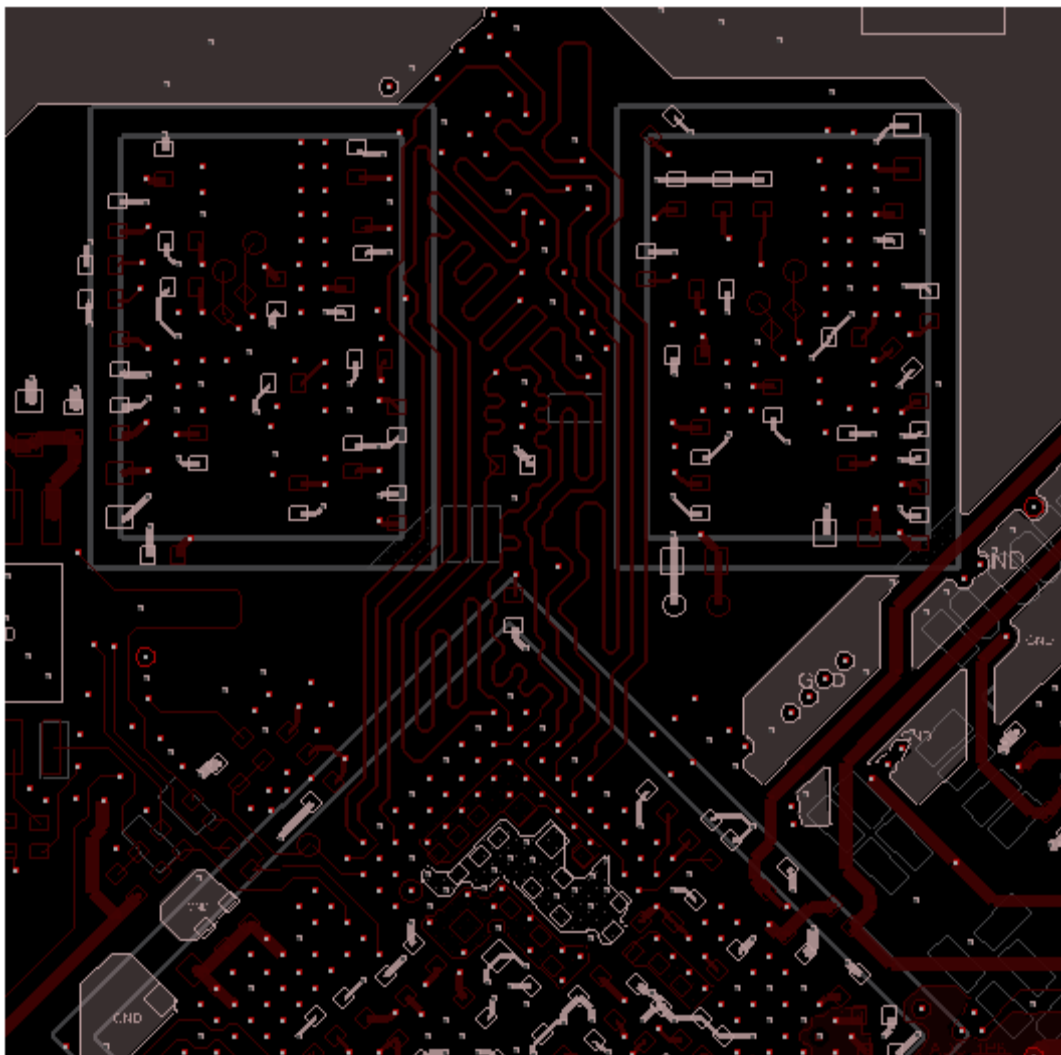


Figure 8. Bottom layer

3.4.3. High speed routing recommendations

The following list provides recommendations for routing traces for high speed signals. Note that the propagation delay and the impedance control should match in order to have the correct communication with the devices.

- High-speed signals (DDR, RGMII, display) must not cross gaps in the reference plane
- Avoid creating slots, voids, and splits in reference planes. Review via voids to ensure they do not create splits (space out vias)
- Provide ground return vias within 100 mils distance from signal layer-transition vias when transitioning between different reference ground planes
- A solid GND plane must be directly under crystal, associated components, and traces
- Clocks or strobes that are on the same layer need at least $2.5\times$ spacing from an adjacent trace ($2.5\times$ height from reference plane) to reduce cross-talk
- Provide ground return vias within 100 mils distance from signal layer-transition vias when

transitioning between different reference ground planes

- All synchronous modules should have bus length matching and relative clock length control

For SD module interfaces:

- Match data and CMD trace lengths (length delta depends on bus rates)
- CLK should be longer than the longest signal in the Data/CMD group (+5 mils)
- Similar DDR rules must be followed for data, address and control as for SD module interfaces

3.4.4. Ground plane recommendations

This section provides examples of good practices and how to avoid common user mistakes when flowing the ground planes layers.

The following two figures show common examples of poor GND planes.

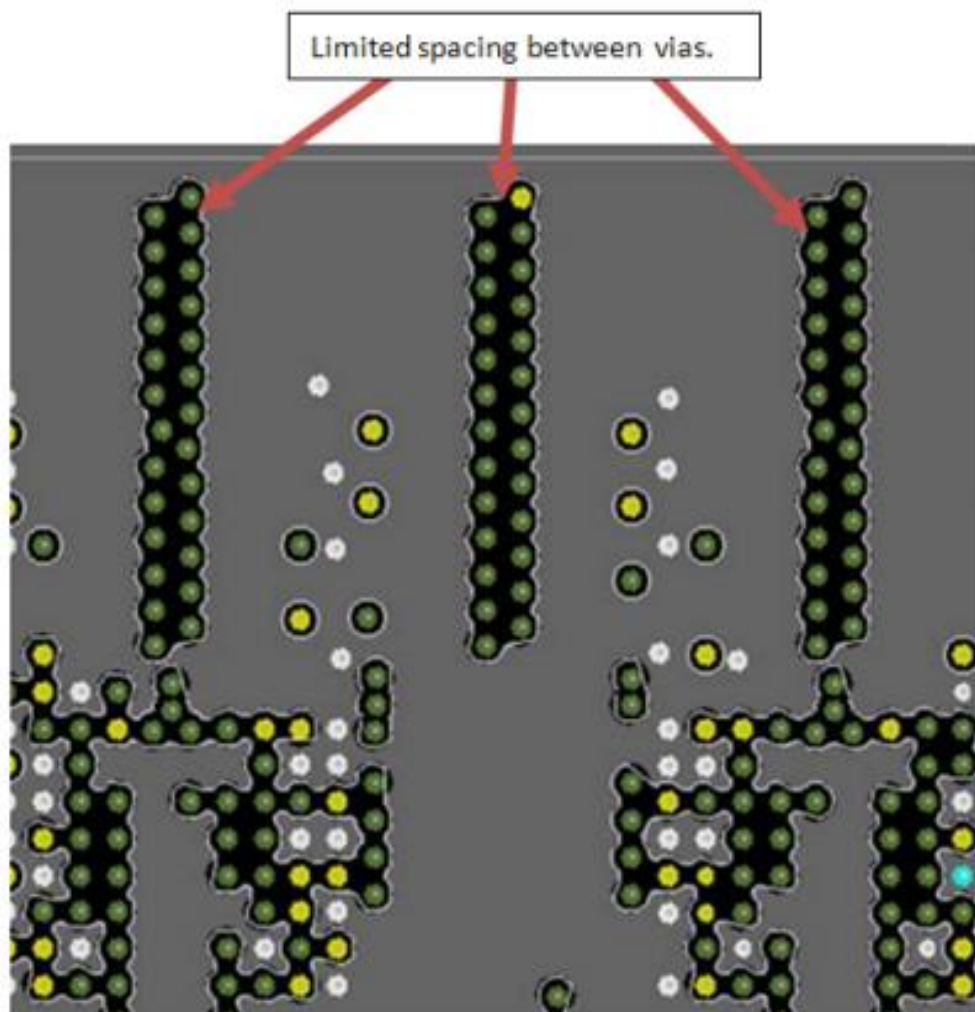


Figure 9. Poor GND plane 1

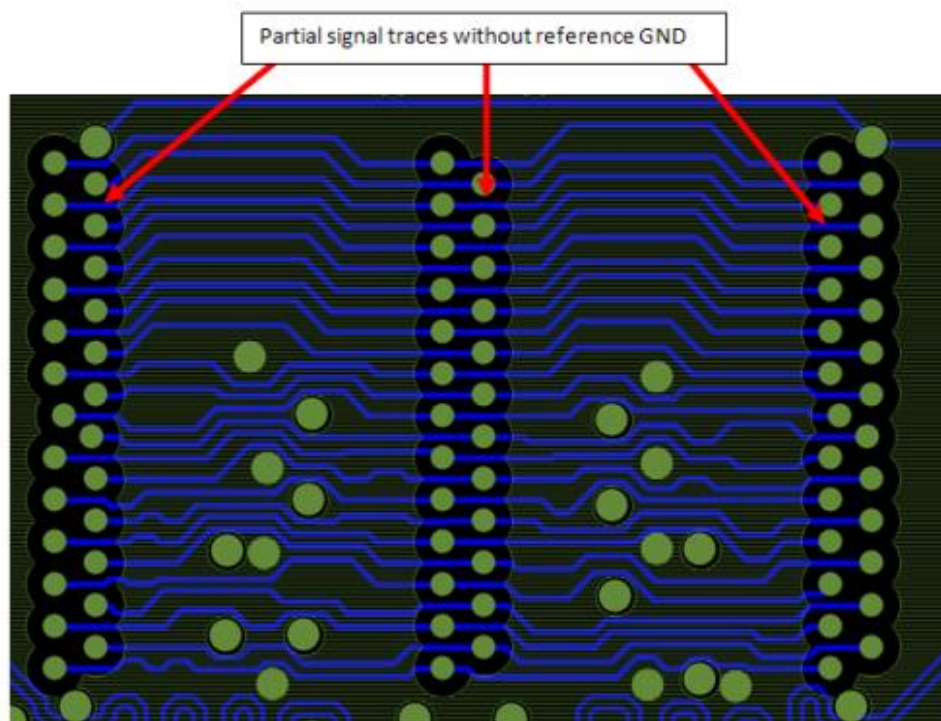


Figure 10. Poor GND plane 2

Spacing the vias some mils apart facilitates the GND copper flowing in the plane. The following figures show good practices of ground planes.

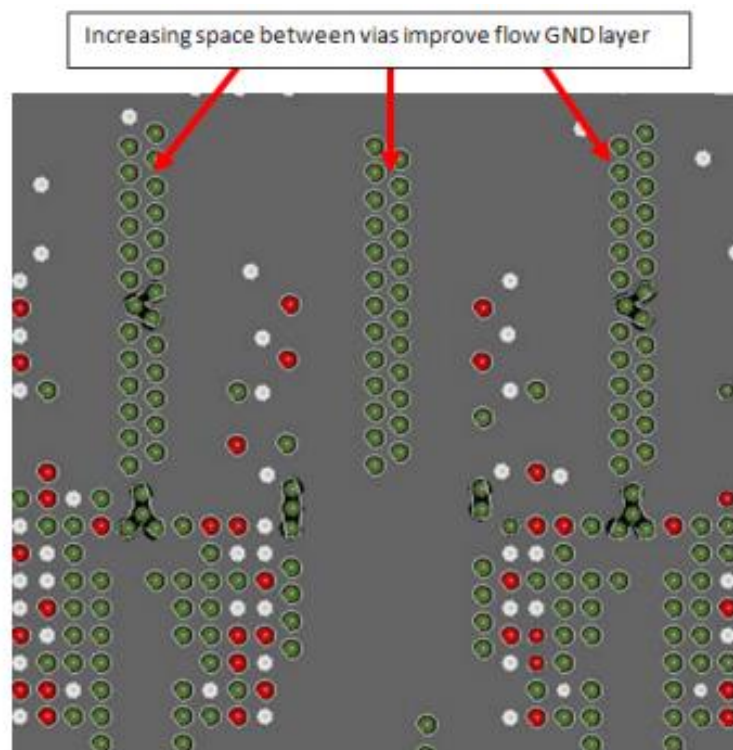


Figure 11. Good layout GND plane detail

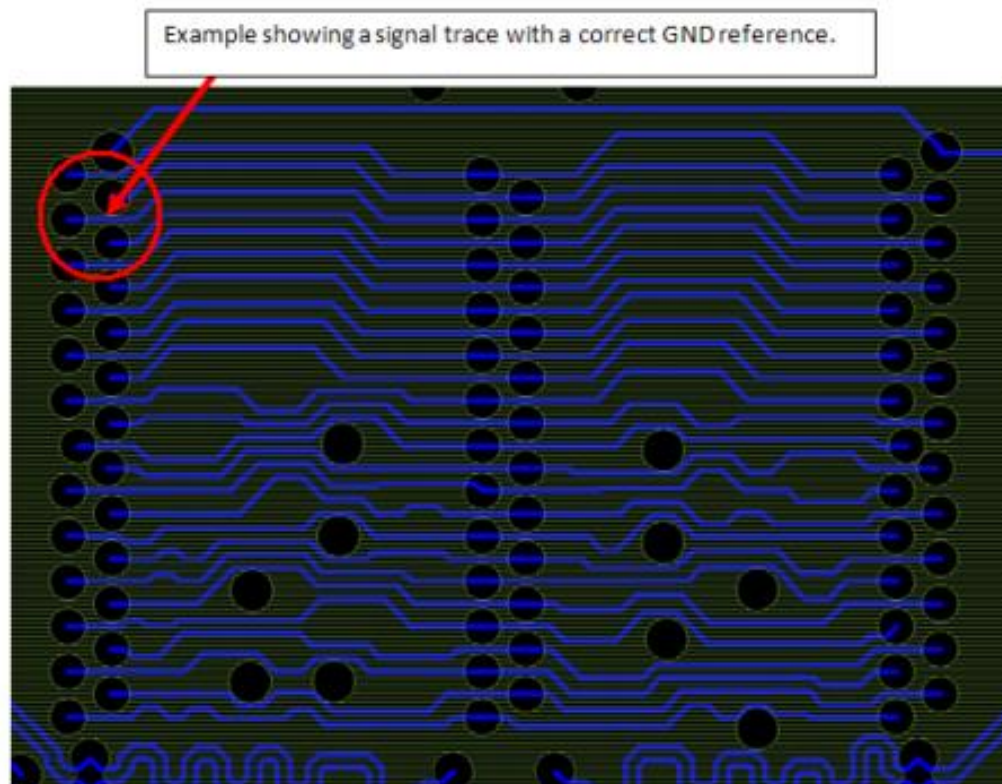


Figure 12. Good layout GND plane detail

3.5. DDR power recommendations

The following recommendations apply to the VREF (P0V675_REFDDR) voltage reference plane:

- Use 30 mils trace between decoupling cap and destination
- Maintain a 25 mils clearance from other nets
- Isolate VREF and/or shield with ground

Decouple using distributed 0.22 μ F capacitors by the regulator, controller, and devices.

- Place one 1.0 μ F near the source of VREF: one near the VREF pin on the controller and two between the controller and the devices

The following recommendations apply to the VTT (DDR_VTT) voltage reference plane. The figures are examples from the evaluation board for the VTT reference schematic.

- Place the VTT island on the component side layer at the end of the bus behind the DRAM devices
- Use a wide-island trace for current capacity
- Place the VTT generator as close to termination resistors as possible to minimize impedance (inductance)
- Place one or two 0.1 μ F decoupling capacitors by each termination RPACK on the VTT island to minimize the noise on VTT. Other bulk (10–22 pF) decoupling is also recommended to be placed on the VTT island

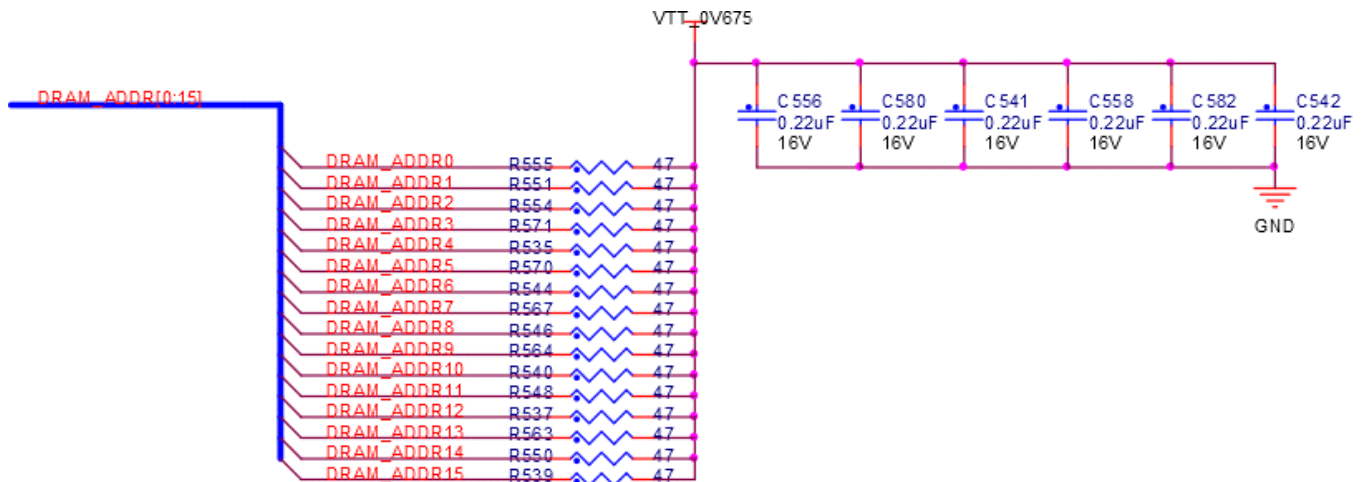


Figure 13. DDR_VTT validation board example

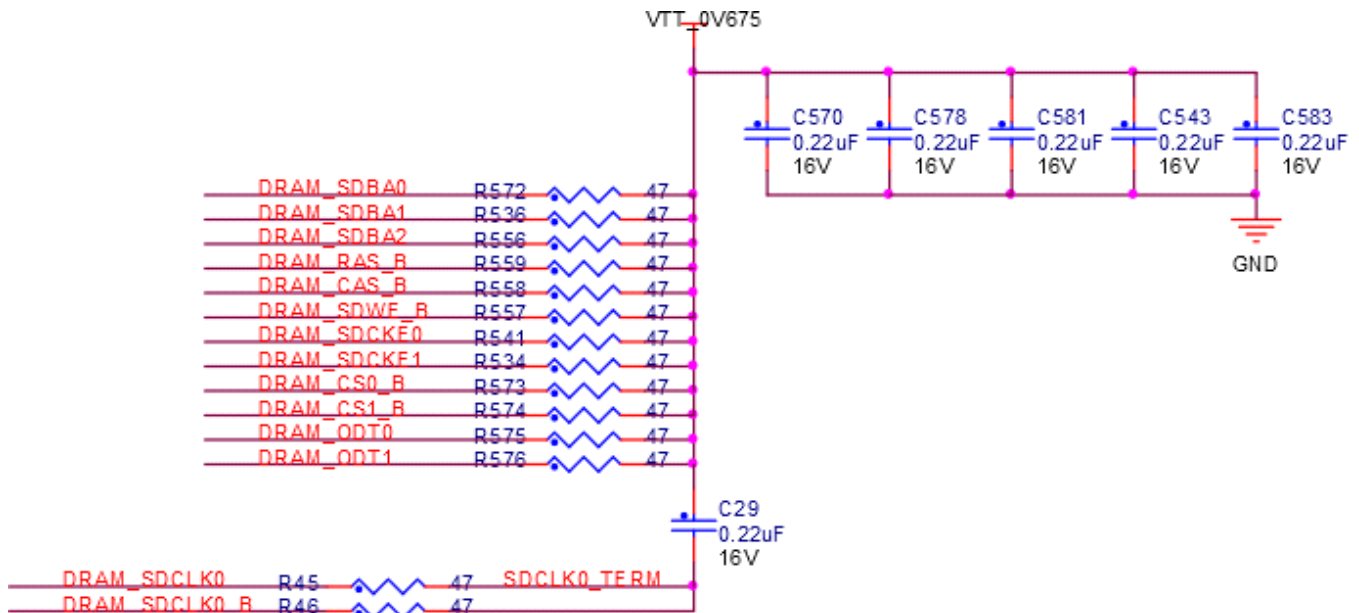


Figure 14. DDR_VTT validation board examples

3.6. PCI Express interface recommendations

This chip provides a $\times 1$ PCIe lane. The PCIe module supports PCI Express Gen 2.0 interfaces at 5 GB/s. It is also backwards compatible to Gen 1.1 interfaces at 2.5 GB/s.

NOTE

Lane $\times 1$ is composed of two differential signals pairs: one TXD signal pair and one RXD signal pair.

Table 21. PCI Express signal description

Signal Name	Signal Group	Description
PCIE_TXP, PCIE_TXN	Data	PCI Express transmit differential pair
PCIE_RXP, PCIE_RXN	Data	PCI Express receive differential pair

3.6.1. PCI Express General routing guidelines

Use the following recommendations for PCI Express general routing:

- The trace width and spacing of the lanes $\times 1$ signals should be such that the differential impedance is $85\ \Omega \pm 10\%$
- The PCIE_REXT contacts should be connected to a $200 \pm 1\%$ resistor to ground. The trace length between the pin and the resistor should be minimized. The resistor value is defined within the data sheet and should determine the exact resistor value
- Route traces over continuous planes (power and ground):
 - They should not pass over any power/GND plane slots or anti-etch
 - When placing connectors, make sure the ground plane clear-outs around each pin have ground continuity between all pins
- Maintain the parallelism (skew matched) between DP and DM, and match overall differential length difference to less than 5 mils
- Maintain symmetric routing for each differential pair
- Do not route DP and DM traces under oscillators or parallel to clock traces and/or data buses.
- Minimize the lengths of high speed signals that run parallel to the DP and DM pair
- Keep DP and DM traces as short as possible
- Route DP and DM signals with a minimum amount of corners. Use 45-degree turns instead of 90-degree turns
- Avoid layer changes (vias) on DP and DM signals. Do not create stubs or branches
- Provide ground return vias within 50 mils distance from signal layer-transition vias when transitioning between different reference ground planes

3.6.2. PCI Express coupling lane

Based on our development design, we have the following coupling signal schema. Consult the PCISig documentation for detailed information.

- DC-coupled Rx signals with $0\ \Omega$ resistors
- AC-coupled Tx signals with $0.1\ \mu\text{F}$ capacitors

3.6.3. Additional resources for PCI Express signal routing recommendations

For more information about PCI Express signal routing recommendations see the following.

- NXP Hardware Design Considerations for PCI Express® and SGMII
(http://www.nxp.com/support/online-academy/hardware-design-considerations-for-pci-express-and-sgmii:TP_HARDWARE_DESIGN_PCI_SMGIII?srch=1&sr=1&pageNum=1)
- PCISig, PCI Express Base Specification
- PCISig, PCI Express Card Electromechanical Specification
- PCISig, PCSIG Board Design Guidelines for PCI Express™ Architecture

- PCI Express Basics: Developing Physical Design Rules for PCIe (<http://www.mentor.com/products/pcb-system-design/multimedia/pcie-basics-webinar>)

3.6.4. USB recommendations

Use the following recommendations for the USB:

- Route the high speed clocks and the DP and DM differential pair first
- Route DP and DM signals on the top or bottom layer of the board
- The trace width and spacing of the DP and DM signals should meet the differential impedance requirement of 90 Ω
- Route traces over continuous planes (power and ground):
 - They should not pass over any power/GND plane slots or anti-etch
 - When placing connectors, make sure the ground plane clearouts around each pin have ground continuity between all pins
- Maintain the parallelism (skew matched) between DP and DM, and match overall differential length difference to less than 5 mils
- Maintain symmetric routing for each differential pair
- Do not route DP and DM traces under oscillators or parallel to clock traces and/or data buses
- Minimize the lengths of high speed signals that run parallel to the DP and DM pair
- Keep DP and DM traces as short as possible
- Route DP and DM signals with a minimum amount of corners. Use 45-degree turns instead of 90-degree turns
- Avoid layer changes (vias) on DP and DM signals. Do not create stubs or branches
- Provide ground return vias within 50 mils distance from signal layer-transition vias when transitioning between different reference ground planes

3.6.5. Impedance signal recommendations

Use the following table as a reference when you are updating or creating constraints in your software PCB tool to set up the impedance and the correct trace width.

Table 22. Impedance signal recommendations

Signal Group	Impedance	Layout Tolerance (\pm)
All signals, unless specified	50 Ω SE	10 %
PCIe Diff signals	85 Ω Diff	10 %
USB Diff signals	90 Ω Diff	10 %
Diff signals: DDR, Phy IC to Ethernet Connector	100 Ω Diff	10 %

The following figure shows the dimensions of a stripline and microstrip pair. [Figure 16](#) shows the differential pair routing.

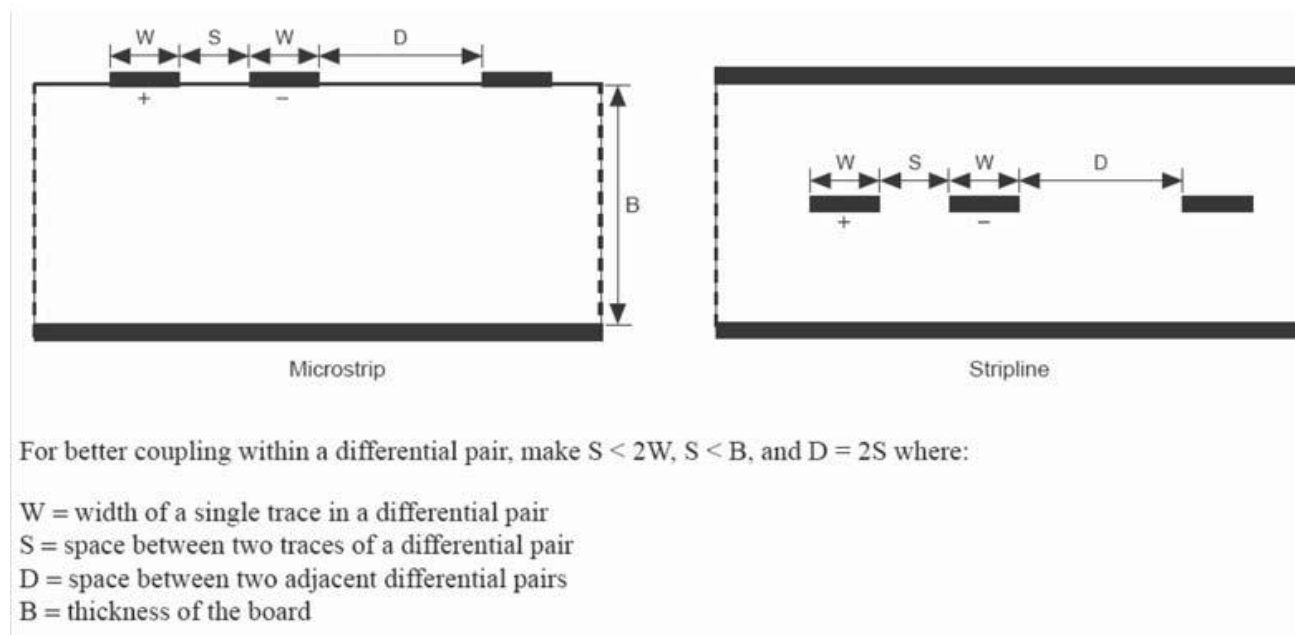


Figure 15. Microstrip and stripline differential pair dimensions

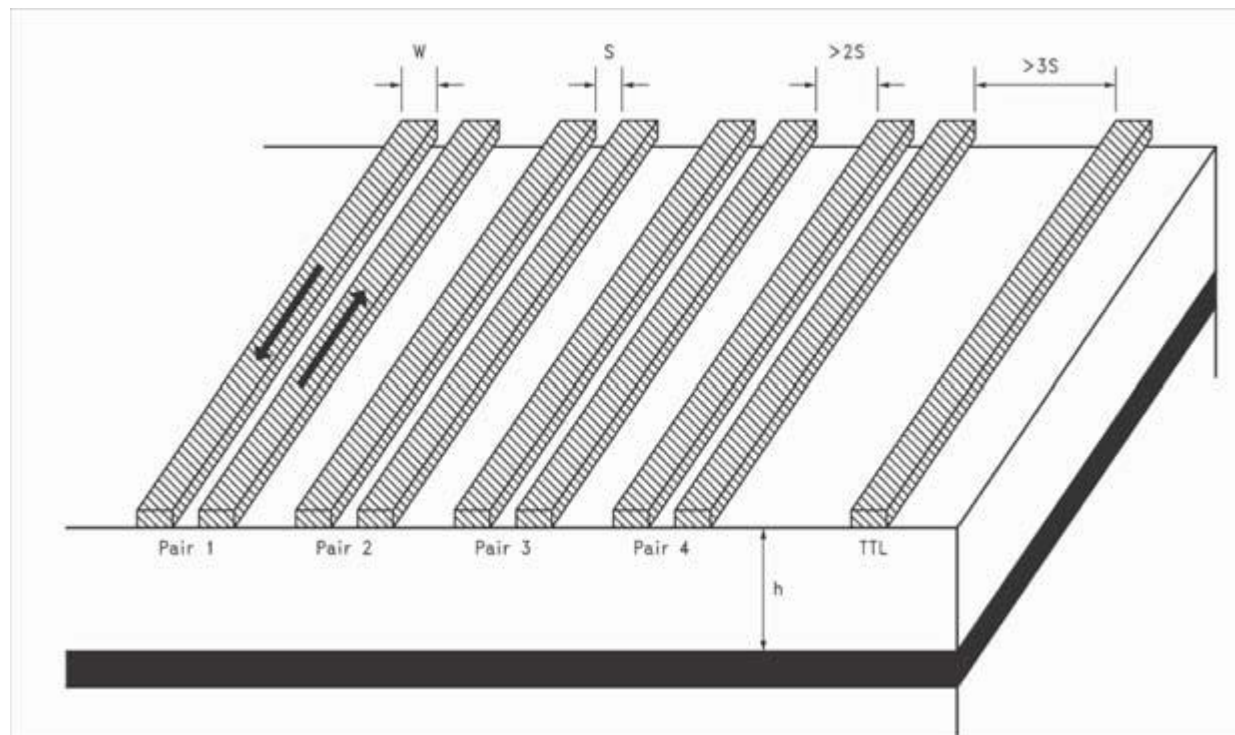


Figure 16. Differential pair routing

- The space between two adjacent differential pairs should be greater than or equal to twice the space between the two individual conductors.
- The skew between LVDS pairs should be within the minimum recommendation (± 100 mil).

3.7. Reference Resistors

NOTE

The reference resistor and the connection should be placed away from noisy regions. Noise induced on it may impact the internal circuit and degrade the interface signals.

3.8. ESD and radiated emissions recommendations

The PCB design should use six or more layers, with solid power and ground planes. The recommendations for ESD immunity and radiated emissions performance are as follows:

- All components with ground chassis shields (USB jack, buttons, and so on) should connect the shield to the PCB chassis ground ring.
- Ferrite beads should be placed on each signal line connecting to an external cable. These ferrite beads must be placed as close to the PCB jack as possible.

NOTE

Ferrite beads should have a minimum impedance of 500Ω at 100 MHz with the exception of the ferrite on USB_5V.

- Ferrite beads should NOT be placed on the USB D+/D– signal lines as this can cause USB signal integrity problems. For radiated emissions problems due to USB, a common mode choke may be placed on the D+/D– signal lines. However, in most cases, it should not be required if the PCB layout is satisfactory. Ideally, the common mode choke should be approved for high speed USB use or tested thoroughly to verify there are no signal integrity issues created.
- It is highly recommended that ESD protection devices be used on ports connecting to external connectors. See the reference schematic (available at nxp.com) for detailed information about ESD protection implementation on the USB interfaces.
- If possible, stitch all around the board with vias with 100 mils spacing between them connected to GND planes with exposed solder mask to improve EMI.

3.9. Component placement recommendations

Adhere to the following recommendations when placing components:

- Place components such that short and/or critical routes can be easily laid out.
 - Critical routes determine component location
 - Orient devices to facilitate routes (minimize length and crossovers)

Consider placing the following pairings adjacent:

- i.MX and DDR
- PHY and associated jack
- Jack and CODEC input
- Bluetooth® (or other RF) and antenna

3.10. Reducing skew and phase problems in deferential pairs traces

Differential pair technology has evolved to require more stringent checking in the area of phase control. This is evident on the higher data rates associated with parallel buses such as PCI Gen 2, DDR, LVDS, or Ethernet. In the simplest of terms, Diff Pair technology sends opposite and equal signals down a pair of traces. Keeping these opposite signals in phase is essential to assuring that they function as intended.

Figure 17 and *Figure 18* show two examples of static routing where a match is achieved without needing to tune one element of the differential pair.

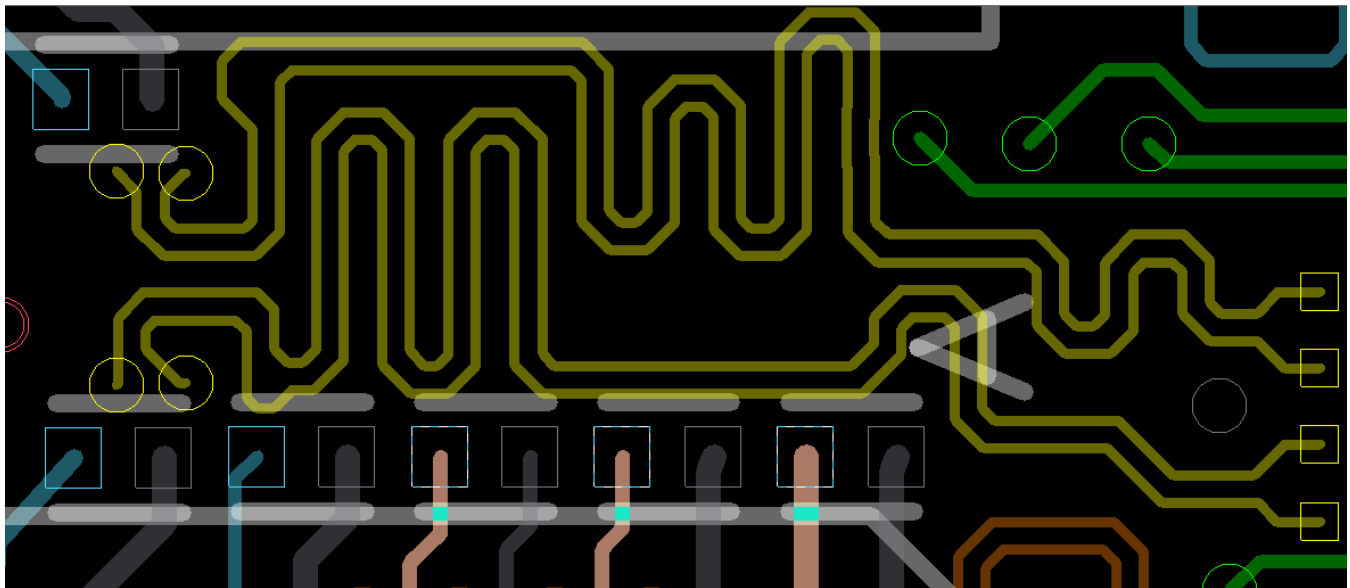


Figure 17. Yellow traces diff pairs 1

The following figure shows the addition of a delay trace to one element of the differential pair to avoid length mismatch (which reduces skew and phase problems). The green box marks the detail.

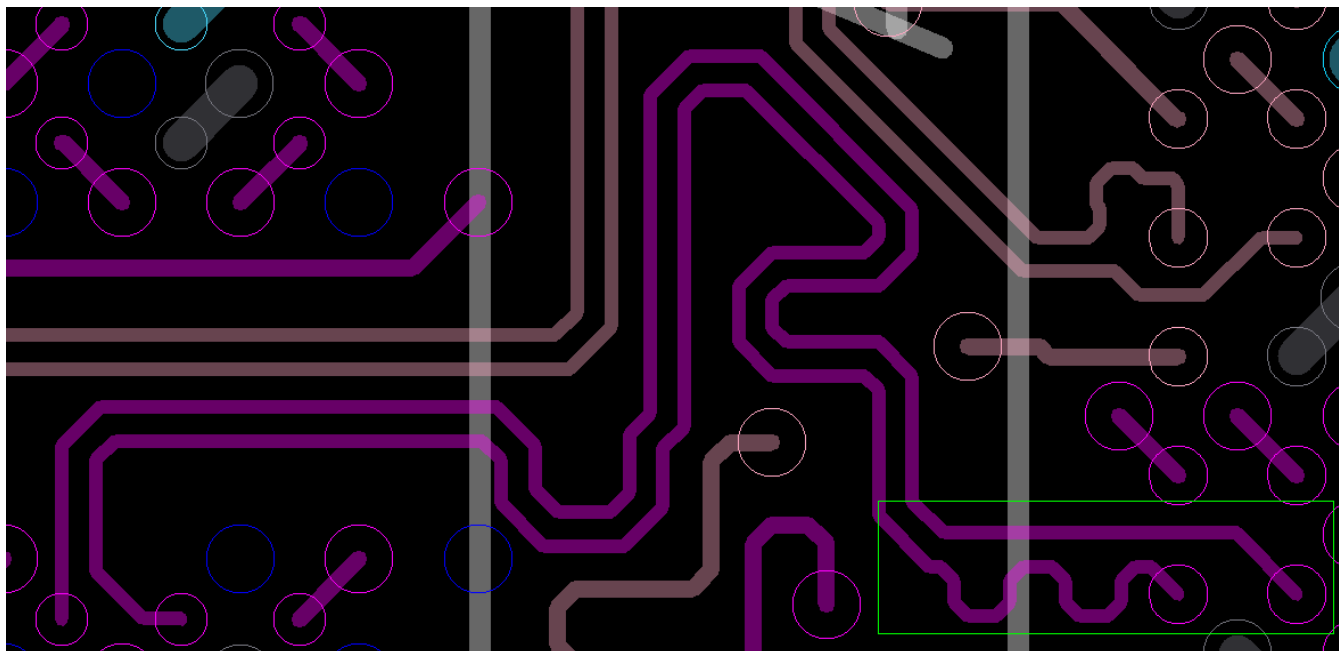


Figure 18. Small bumps added to the shorter differential pair

4. Avoiding Board Bring-up Problems

4.1. Introduction

This chapter provides recommendations for avoiding typical mistakes when bringing up a board for the first time. These recommendations consist of basic techniques that have proven useful in the past for detecting board issues and addressing the three most typical bring-up pitfalls: power, clocks, and reset. A sample bring-up checklist is provided at the end of the chapter.

4.2. Using a current monitor to avoid power pitfalls

Using incorrect voltage rails is a common power pitfall. To help avoid this mistake, create a basic table called a voltage report prior to bringing up your board. This table helps validate that all the supplies are reaching the expected levels.

To create a voltage report, list the following:

- Your board voltage sources
- Default power-up values for the board voltage sources
- Best location on the board to measure the voltage level of each supply

Carefully determine the best measurement location for each power supply to avoid a large voltage drop (IR drop) on the board, which causes inaccurate current values to be measured. The following guidelines help produce the best current measurements:

- Measure closest to the load (in this case the i.MX6 processor)
- Make two measurements: the first after initial board power-up and the second while running a heavy use-case that stresses the i.MX7 processor

Ensure that the supplies that are powering the i.MX7 meet the DC electrical specifications as listed in your chip-specific data sheet.

The shaded cells in the PMIC LDO2 row call your attention to the difference in the expected value and measured value, which indicates a potential problem with that power rail.

Table 23. Sample voltage report

Source	Net Name	Expected (V)	Measured (V)	Measure Point	Comment
Main	PSU_5V0	—	—	—	—
3.3 V /1A	MEM_3V3	3.3 V	—	—	—
PMIC Switcher 1A	VDD_ARM	1.1 V	—	—	—
PMIC Switcher 1B	VDD_SOC	1.0 V	—	—	—
PMIC Switcher 3	NVCC_DRAM	1.35 V	—	—	—
PMIC LDO1	VLDO1_1P8	1.8 V	—	—	—
PMIC LDO2	VLDO2_1V5	1.5 V	—	—	—
PMIC LDO3	VLDO3_3V3	3.3 V	—	—	—
PMIC LDO4	VLDO4_2V8	2.8 V	—	—	—
PMIC	NVCC_SD1	—	—	—	—

Table 23. Sample voltage report

Source	Net Name	Expected (V)	Measured (V)	Measure Point	Comment
VCC_SD					
PMIC V33	NVCC_3V3	3.3 V	—	—	—
PMIC SWBSTLX	PMIC_5V	5.0 V	—	—	—
iMX7	VDDD_1P0_CAP	1.0 V	—	—	—
iMX7	VDDA_PHY_1P8	1.8 V	—	—	—
i.MX7	VDD_1P2_CAP	1.2 V	—	—	—
i.MX7 SNVS	VDD_SNVS_1P8_CAP	1.8 V	—	—	—
i.MX7	VDD_LPSR_1P0_CAP	1.0 V	—	—	—
i.MX7	PVCC_GPIO_CAP	3.3 V	—	—	—

4.3. Checking for clock pitfalls

Problems with the external clocks are another common source of board bring-up issues. Ensure that all of your clock sources are running as expected.

The XTALI/XTALO and the RTC_XTALI/RTC_XTALO clocks are the main clock sources for 24 MHz and 32 kHz reference clocks respectively on the i.MX6. Although not required, the use of low jitter external oscillators to feed CLK1_P/N or CLK2 on the i.MX7 can be an advantage if low jitter or special frequency clock sources are required by modules driven by CLK1_P/N or CLK2. See the CCM chapter in your i.MX7 chip reference manual for details. If a 32.768 kHz crystal is not connected to the i.MX7, an on-chip ring oscillator is automatically used for the low-frequency clock source.

When checking crystal frequencies, use an active probe to avoid excessive loading. A parasitic probe typically inhibits the 32.768 kHz and 24 MHz oscillators from starting up. Use the following guidelines:

- RTC_XTALI clock is running at 32.768 kHz (can be generated internally or applied externally).
- XTALI/XTALO is running at 24 MHz (used for the PLL reference).
- CLK1_P/N/CLK2 can be used as oscillator inputs for low jitter special frequency sources.
- CLK1_P/N and CLK2 are optional.

In addition to probing the external input clocks, you can check internal clocks by outputting them at the debug signals CLKO1 and CLKO2 (iomuxed signals). See the CCM chapter in the chip reference manual for more details about which clock sources can be output to those debug signals. JTAG tools can be used to configure the necessary registers to do this.

4.4. Avoiding reset pitfalls

Follow these guidelines to ensure that you are booting using the correct boot mode.

- During initial power on while asserting the POR_B reset signal, ensure that 24 MHz clock is active before releasing POR_B.
- Follow the recommended power-up sequence specified in the i.MX7 data sheet.
- Ensure the POR_B signal remains asserted (low) until all voltage rails associated with bootup are on.

The GPIOs and internal fuses control how the i.MX7 boots. For a more detailed description about the different boot modes, see the system boot chapter of the chip reference manual.

4.5. Sample board bring-up checklist

Note that the checklist incorporates the recommendations described in the previous sections. Blank cells should be filled in during bring-up as appropriate.

Table 24. Board bring-up checklist

Checklist Item	Details	Owner	Finding & status
Note: The following items must be completed serially.			
Perform a visual inspection	Check major components to make sure nothing has been misplaced or rotated before applying power.		
Verify all i.MX voltage rails.	Confirm that the voltages match the data sheet's requirements. Be sure to check voltages not only at the voltage source, but also as close to the i.MX as possible (like on a bypass capacitor). This reveals any IR drops on the board that will cause issues later. Ideally all of the i.MX voltage rails should be checked, but VDD_ARM_IN and VDD_SOC_IN are particularly important voltages. These are the core logic voltages and must fall within the parameters provided in the i.MX6 data sheet. VDD_SNVS_IN, NVCC_JTAG, and NVCC_DRAM are also critical to the i.MX boot up.		
Verify power-up sequence	Verify that power on reset (POR_B) is de-asserted (high) after all power rails have come up and are stable. See the i.MX data sheet for details about power-up sequencing.		
Measure/probe input clocks (32 kHz, 24MHz, others).	Without a properly running clock, the i.MX will not function properly.		
Check JTAG connectivity	This is one of the most fundamental and basic access points to the i.MX to allow the debug and execution of low level code.		
Note: The following items may be worked on in parallel with other bring up tasks.			
Access internal RAM.	Verify basic operation of the i.MX in system. Perform a basic test by performing a write-read-verify to the internal RAM. No software initialization is necessary to access internal RAM.		
Verify CLKO outputs (measure and verify default clock frequencies for desired clock output options) if the board design supports probing of the CLKO pin.	This ensures that the corresponding clock is working and that the PLLs are working. Note that this step requires chip initialization, for example via the JTAG debugger, to properly set up the IOMUX to output CLKO and to set up the clock control module to output the desired clock. See the reference manual for more details.		
Measure boot mode frequencies. Set the boot mode switch for each boot mode and measure the following (depending on system availability): • NAND (probe CE to verify boot, measure RE frequency) • SPI-NOR (probe slave select and measure clock frequency) • MMC/SD (measure clock frequency)	This verifies the specified signals' connectivity between the i.MX7 and boot device and that the boot mode signals are properly set. See the "System Boot" chapter in the reference manual for details about configuring the various boot modes.		

Table 24. Board bring-up checklist

Checklist Item	Details	Owner	Finding & status
Run basic DDR initialization and test memory.	<ol style="list-style-type: none"> 1. Assuming the use of a JTAG debugger, run the DDR initialization and open a debugger memory window pointing to the DDR memory map starting address. 2. Try writing a few words and verify if they can be read correctly. 3. If not, recheck the DDR initialization sequence and whether the DDR has been correctly soldered onto the board. <p>It is also recommended that users recheck the schematic to ensure that the DDR memory has been connected to the i.MX7 correctly.</p>		

5. Ethernet Connections

The i.MX7DS Ethernet MAC supports RGMII, RMII, and MII physical layer interfaces. [Table 25](#) and [Table 26](#) help decode the i.MX7 signal names to the various physical layer interfaces.

5.1. ENET Connections for RGMII

The i.MX signal names are, by default RGMII.

5.2. ENET Connections for RMII

The following tables show the interconnects and decoder ring for the Ethernet interface on the i.mx7D/S. The RMII Signals column are the RMII phy reference names, the i.MX7 Signals are the names of the signals as found in the imx7D.

Table 25. Ethernet rmii chart for enet2

RMII Signals	i.MX7 Signals	i.MX7 Pad	i.mx7 Alt Mode
TXD0	Enet2.RGMII_TD0	EPDC_SDCE2	Alt2
TXD1	Enet2.RGMII_TD1	EPDC_SDCE3	Alt2
TX_EN	Enet2.RGMII_TX_CTL	EPDC_GDRL	Alt2
RXD0	Enet2.RGMII_RD0	EPDC_SDCLK	Alt2
RXD1	Enet2.RGMII_RD1	EPDC_SDLE	Alt2
CRS_DV	ENET2_RX_CTL	EPDC_SDCE0	Alt2
RX_ER	Enet2.RX_ER	EPDC_SDCE1	Alt3
REF_CLK (50MHz)	Ccm.ENET2_REF_CLK_ROT	EPDC_BDR0 GPIO1_IO03 GPIO_IO13 I2C2_SCL	Alt3 Alt2 Alt2 Alt4

Table 26. Ethernet rmii chart for enet1

RMII Signals	i.MX7 Signals	i.MX7 Pad	i.mx7 Alt Mode
TXD0	enet1.RGMII_TD0	ENET1_TD0	Alt0
TXD1	enet1.RGMII_TD1	ENET1_TD1	Alt0
TX_EN	enet1.RGMII_TX_CTL	ENET1_TX_CTL	Alt0
RXD0	enet1.RGMII_RD0	ENET1_RD0	Alt0
RXD1	enet1.RGMII_RD1	ENET1_RD1	Alt0
CRS_DV	ENET1_CRD	ENET1_RX_CTL	Alt0
RX_ER	enet1.RX_ER	ENET1_RXC	Alt1
REF_CLK (50MHz)	cmm.ENET1_REF_CLK_ROT	GPIO1_IO02 GPIO1_IO12 I2C1_SDA ENET1_TX_CLK	Alt2 Alt2 Alt4 Alt1

6. IBIS Model

NXP provides IBIS models for the two different packages (19x19 & 12x12), two different GPIO voltage, and various ddr3 memory types.

6.1. Naming convention for model names and usage for i.MX7

The model names are defined per each [Model selector]. The models may differ from each other by having different parameters—such as voltage, drive strength, mode of operation, and slew rate. The mode of operation, drive strength, and slew rate parameters are programmable by software.

See [Table 25](#) for the filenames and supported models.

Table 27. IBIS Model naming convention

Filename	Package	IO Voltage	Memory type
mx7d_12x12_1p8v_ddr3_odt.ibs	12x12	1.8 V	DDR3 with ODT
mx7d_12x12_1p8v_ddr3l.ibs	12x12	1.8 V	DDR3L
mx7d_12x12_1p8v_lpddr3.ibs	12x12	1.8 V	LPDDR3
mx7d_12x12_3p3v_ddr3_odt.ibs	12x12	3.3 V	DDR3 with ODT
mx7d_12x12_3p3v_ddr3l_odt.ibs	12x12	3.3 V	DDR3L with ODT
mx7d_12x12_3p3v_lpddr3.ibs	12x12	3.3 V	LPDDR3
mx7d_19x19_1p8v_ddr3_odt.ibs	19x19	1.8 V	DDR3 with ODT
mx7d_19x19_1p8v_ddr3l_odt.ibs	19x19	1.8 V	DDR3L with ODT
mx7d_19x19_1p8v_lpddr3.ibs	19x19	1.8 V	LPDDR3
mx7d_19x19_3.3v_ddr3_odt.ibs	19x19	3.3 V	DDR3 with ODT
mx7d_19x19_3p3v_ddr3l_odt.ibs	19x19	3.3 V	DDR3L with ODT
mx7d_19x19_3p3v_lpddr3.ibs	19x19	3.3 V	LPDDR3

6.2. Understanding the IBIS Model

This chapter explains how to use the IBIS (input output buffer information specification) model, which is an Electronic Industries Alliance standard for the electronic behavioral specifications of integrated circuit input/output analog characteristics. The model is generated in ASCII text format and consists of multiple tables that capture current vs. voltage (IV) and voltage vs. time (VT) characteristics of each buffer. IBIS models are generally used to perform PCB-board-level signal integrity (SI) simulations and timing analyses.

The IBIS model's features are as follows:

- Supports fast chip-package-board simulation, with SPICE-level accuracy and faster than any transistor-level model
- Provides the following for portable model data
- I/O buffers, series elements, terminators
 - Package RLC parasitics
 - Electrical board description

6.3. IBIS structure and content

An IBIS file contains the data required to model a component's input, output, and I/O buffers behaviorally in ASCII format. The basic IBIS file contains the following data:

- Header information regarding the model file
- Information about the component, the package's electrical characteristics, and the pin-to-buffer model mapping (in other words, which pins are connected to which buffer models)
- The data required to model each unique input, output, and I/O buffer design on the component

IBIS models are component-centric, meaning they allow users to model an entire component rather than only a particular buffer. Therefore, in addition to the electrical characteristics of a component's buffers, an IBIS file includes the component's pin-to-buffer mapping and the electrical parameters of the component's package.

6.4. Header Information

The first section of an IBIS file provides the basic information about the file and its data. The following table explains the header information notation.

Table 28. IBIS Header Information

Keyword	Required	Description
[IBIS Ver]	Yes	Version of IBIS Specification this file uses.
[Comment char]	No	Change the comment character. Defaults to the pip () character.
[File Name]	Yes	Name of this file. All file names must be lower case. The file name extension for IBIS file is ".ibs"
[File Rev]	Yes	The revision level of this file. The specification contains guideline for assigning revision levels.
[Date]	No	Date the file was created.
[Source]	No	The source of the data in this file. Data is taken from a simulation and validated on a board.
[Notes]	No	Component or file-specific notes.
[Disclaimer]	No	May be legally required
[Copyright]	No	The file's copyright notice

Example 1. IBIS Header Information

```
[IBIS Ver] 4.2
[File name] ult1_12x12_3p3v_ddr3_odt.ibs
[File Rev] 1.0
[Date] Oct 6, 2015
[Source]
[Disclaimer]
```

6.5. Component and pin information

The second section of an IBIS file is where the data book information regarding the component's pinout, pin-to-buffer mapping, and the package and pin electrical parameters is placed.

Table 29. Component and Pin Information

Keyword	Required	Description
[Component]	Yes	The name of the component being modeled. Standard practice has been to use the industry standard part designation. Note that IBIS files may contain multiple [Component] descriptions.
[Manufacturer]	Yes	The name of the component manufacturer
[Package]	Yes	This keyword contains the range (minimum, typical and maximum values) over which the packages' lead resistance, inductance, and capacitance vary (the R_pkg, L_pkg, and C_pkg parameters).
[Pin]	Yes	This keyword contains the pin-to-buffer mapping information. In addition, the model creator can use this keyword to list the package information: R, L, and C data for each individual pin (R_pin, L_pin, and C_pin parameters).
[Package Model]	No	If the component model includes an external package model (or uses the [Define Package Model] keyword within the IBIS file itself), this keyword indicates the name of that package model.
[Pin Mapping]	No	This keyword is used if the model creator wishes to include information on buffer power and ground connections. This information may be used for simulations involving multiple outputs switching.
[Diff Pin]	No	This keyword is used to associate buffers that should be driven in a complementary fashion as a differential pair.
[Model Selector]		This keyword provides a simple means by which several buffers can be made optionally available for simulation at the same physical pin of the component.

Example 2. Component and pin information

```
[Component] mx7_ult1_12x12
[Manufacturer] Freescale

|||||

[Package]
|variable      typ              min              max
R_pkg  0.2177              0.0666              0.37705
L_pkg  2.3724nH            0.7912nH            4.095nH
C_pkg  0.6482pF            0.3055pF            1.6214pF

[Pin]      signal_name      model_name      R_pin      L_pin      C_pin
A05  SD1_CD_B      MPBI 0.354724  3.18226nH  0.60709pF
B04  SD1_DATA0      MPBI 0.188298  2.01806nH  0.82959pF
A04  SD1_DATA1      MPBI 0.349718  3.33815nH  0.60202pF
```

6.6. Model information

The [Model] keyword starts the description of the data for a particular buffer.

Table 30. Model Information

Keyword	Comment
[Model Spec]	General set of parameters for the model simulation.
[Receiver Thresholds]	Threshold information for the different simulation cases.
[Temperature Range]	The temperature range over which the min, typ and max IV and switching data have been gathered.
[Voltage Range]	The range over which Vcc is varied to obtain the min, type and max pullup and power clamp data.
[Pulldown] [Pullup] [GND_clamp] [POWER_clamp]	IV information.
[Ramp] [Rising Waveform] [Falling Waveform]	VT information
[Test Data] [Rising Waveform Near] [Rising Waveform Far] [Falling Waveform Near] [Falling Waveform Far] [Test Load]	VT golden model information

6.6.1. IV information

IV information composed of four Current-over-Voltage tables: [Pullup], [Pulldown], [[GND_clamp], and [Power_clamp]. Each look-up table describes a different part of the IO cell model.

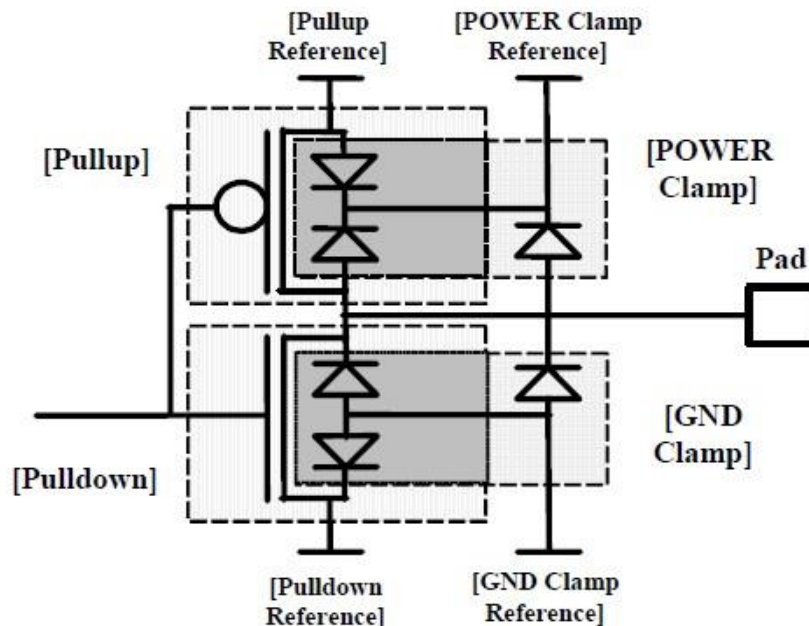


Figure 19. Model IV parameters' structure

6.6.2. VT information

Table 31. VT information

Keyword	Required	Comment
[Ramp]	YES	Basic ramp rate information, given as dV/dt_r for rising edges and dV/dt_f for falling edges, see the following equation. $dV/dt = [20\% \text{ to } 80\% \text{ voltage swing}] / \text{time takes to swing above voltage}$ Note: the dV value is 20 % to 80 % voltage swing of the buffer when driving into a specified load, R_{load} (for [Ramp], this load defaults to 50). For CMOS drivers or I.O buffers, this load assumed to be connected to the voltages defined by the [Voltage Range] keyword for falling edges and to ground for rising edges.
[Rising Waveform]	No	The actual rising (low to high transition) waveform, provided as a VT Table
[Falling Waveform]	No	The actual falling (high to low transition) waveform, provided as a VT table.

Example 3. Ramp and waveform keywords example

```
[Ramp]
dV/dt_r      0.325115/1.08702n  0.195492/1.69623n  0.514829/0.760527n
dV/dt_f      0.266424/0.668118n  0.154283/0.890592n
0.400068/0.550282n
R_load = 50
|
|
[Falling Waveform]
R_fixture = 50
V_fixture = 0
V_fixture_min = 0
V_fixture_max = 0
|
| Time V(typ) V(min) V(max)
|
0 0.565241 0.328108 0.878902
0.7837n 0.565242 0.328109 0.878902
0.7938n 0.565242 0.328109 0.878889
0.7991n 0.565242 0.328109 0.878875
0.8172n 0.565242 0.328109 0.878788
0.8208n 0.565242 0.328109 0.878779
0.8231n 0.565242 0.328109 0.87878
0.8272n 0.565242 0.328109 0.878796
0.8307n 0.565242 0.328109 0.878823
0.8414n 0.565242 0.328109 0.878936
...
[Rising Waveform]
R_fixture = 50
V_fixture = 0
V_fixture_min = 0
V_fixture_max = 0
|
| Time V(typ) V(min) V(max)
```

```

|
0 0.285809u 3.76004u 0.793549u
0.6839n -0.153867u 3.31119u 1.18951u
0.6878n -0.156374u 3.30863u 6.01681u
0.6905n -0.15811u 3.30686u 12.3329u
0.6922n -0.159203u 3.30574u 17.8409u
0.6988n -0.163446u 3.30141u 47.6826u
0.7005n -0.164539u 3.30029u 53.6969u

```

The [Ramp] keyword is always required, even if the [Rising Waveform] and [Falling Waveform] keywords are used. However, the VT tables under [Rising Waveform] and [Falling Waveform] are generally preferred to [Ramp] for the following reasons:

- VT data may be provided under a variety of loads and termination voltages
- VT tables may be used to describe transition data for devices as then turn on and turn off.
- [Ramp] effectively averages the transitions of the device, without providing any details on the shapes of the transitions themselves. All detail of the transition ledges would be lost.

The VT data should be included under two [Rising Waveform] and two [Falling Waveform] sections, each containing data tables for Vcc-connected load and a Ground-connected load (although other loading combinations are permitted).

The most appropriate load is a resistive value corresponding to the impedance of the system transmission lines the buffer will drive (own impedance). For example, a buffer intended for use in a 60 Ω system is best modeled using a 60 Ω load (R_fixture).

- I_down [GND clamp] + [Power clamp] + [Pulldown]
- I_up [GND clamp] + [Power clamp] + [Pullup]
- I_recvr [GND clamp] + [Power clamp]

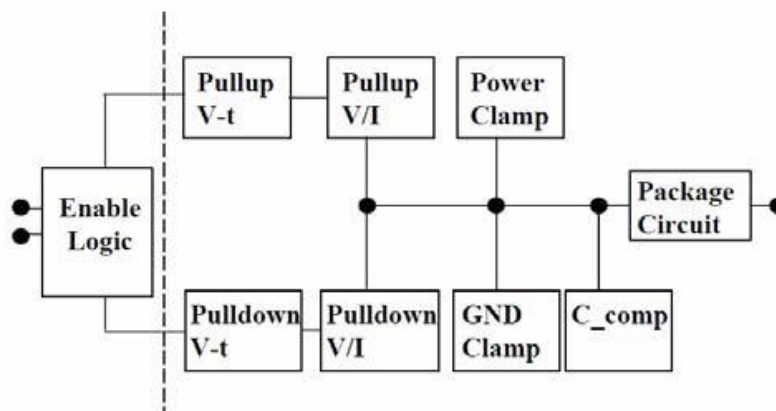


Figure 20. Model data interpretation

6.6.3. Golden Model VT information

Golden waveforms are a set of waveforms simulated using known ideal test loads. They are useful for verifying the accuracy of the behavioral simulation results against the transistor level circuit model from which the IBIS model parameters originate.

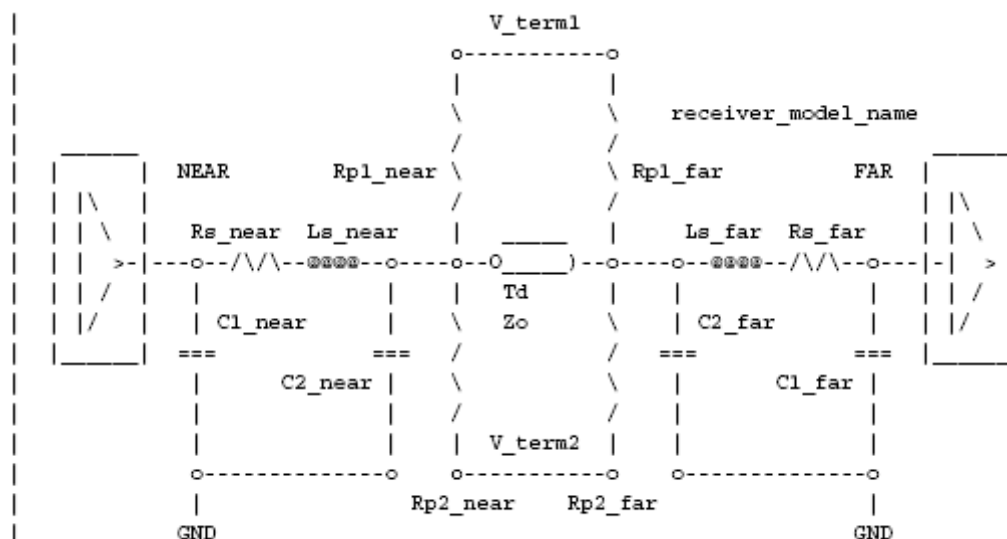


Figure 21. Generic test load network

Table 32. Golden waveform keywords

Keyword	Required	Comment
[Test Data]	No	Provides a set of golden waveforms and references the conditions under which they were derived. Useful for verifying the accuracy of the behavioral simulation results against the transistor level circuit from which the IBIS model parameters originate.
[Rising Waveform Near] [Rising Waveform Far] [Falling Waveform Near] [Falling Waveform Far]	Yes	Current-Over-Voltage tables, for far and near portions of the golden model as described in Figure 21 .
[TestLoad]	Yes	Defines a test load network and its associated electrical parameters for reference by the golden waveforms under the [Test Data] keyword. If Test_load_type is Differential, the test load is a pair of circuits as shown above. If the R_diff_near or R_diff_far subparameter is used. A resistor is connected between the near and far nodes of the two circuits. If Test_load_type is Single_ended, R_diff_near and R_diff_far are ignored.

6.7. IBIS references

Consult the following references for more information about the IBIS model.

- IBIS Open Forum (<https://ibis.org/>)
The IBIS Open Forum consists of EDA vendors, computer manufacturers, semiconductor vendors, universities, and end-users. It proposes updates and reviews, revises standards, and organizes summits. It promotes IBIS models and provides useful documentation and tools.
- IBIS specification (<https://ibis.org/specs/>)

7. Using BSDL for Board-level Testing

7.1. BSDL overview

Boundary scan description language (BSDL) is used for board-level testing after components have been assembled. The interface for this test uses the JTAG pins. The definition is contained within IEEE Std 1149.1.

7.2. How BSDL functions

The BSDL file defines the internal scan chain, which is the serial linkage of the IO cells, within a particular device. The scan chain looks like a large shift register, which provides means to read the logic level applied to a pin or to output a logic state on that pin. Using JTAG commands, a test tool uses the BSDL file to control the scan chain so the device-board connectivity can be tested.

For example, when using an external ROM test interface, the test tool would do the following:

1. Output a specific set of addresses and controls to pins connected to the ROM
2. Perform a read command and scan out the values of the ROM data pins.
3. Compare the values read with the known golden values.

Based on this procedure, the tool can determine whether the interface between the two parts is connected properly and does not contain shorts or opens.

7.3. Downloading the BSDL file

The BSDL file for each i.MX process is stored on the NXP website upon product release. Contact your local sales office or field applications engineer to check the availability of information prior to product releases.

7.4. Pin Coverage of BSDL

Each pin is defined as a port within the BSDL file. You can open the file with a text editor to review how each pin will function. The BSDL file defines these functions as shown:

```
-- PORT DESCRIPTION TERMS
-- in = input only
-- out = three-state output (0, Z, 1)
-- buffer = two-state output (0, 1)
-- inout = bidirectional
-- linkage = OTHER (vdd, vss, analog)
```

The appearance of “linkage” in a pin’s file implies that the pin cannot be used with boundary scan. These are usually power pins or analog pins that cannot be defined with a digital logic state.

7.5. Boundary scan operation

The boundary scan operation is controlled by:

- TEST_MODE, POR_B, and JTAG_MOD pins
- On-chip fuse bits

The JTAG_MOD pin state controls the selection of JTAG to the core logic or boundary scan operation.

See the following references for further information:

- The “System JTAG controller (SJC)” chapter in the chip reference manual for the definitions of the HTAG interface operations.
- The “JTAG Security Modes” section in the same chapter for an explanation of the operation of the e-Fuse bit definitions in the following table.
- The “Fusemap” chapter in the chip reference manual in the fusemap tables.

Table 33. System considerations for BSDL

Pin name	Logic state	Description
JTAG_MOD	1	IEEE 1149.1 JTAG compliant mode
BOOT_MODE[1:0]	[0:0] [0:1] [1:0]	Boot from Fuses Serial Downloader Internal Boot
POR_B	1	Power On Reset for the device
e-Fuse bits		
JTAG_SMODE[1:0]	[0:0] [0:1]	JTAG enable mode Secure JTAG mode
SJC_DISABLE	0	Secure JTAG Controller is enabled.

8. Revision History

Table 34. Revision history

Revision number	Date	Substantive changes
0	07/2016	Initial release

How to Reach Us:

Home Page:

nxp.com

Web Support:

nxp.com/support

Information in this document is provided solely to enable system and software implementers to use NXP products. There are no express or implied copyright licenses granted hereunder to design or fabricate any integrated circuits based on the information in this document. NXP reserves the right to make changes without further notice to any products herein.

NXP makes no warranty, representation, or guarantee regarding the suitability of its products for any particular purpose, nor does NXP assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation consequential or incidental damages. "Typical" parameters that may be provided in NXP data sheets and/or specifications can and do vary in different applications, and actual performance may vary over time. All operating parameters, including "typicals," must be validated for each customer application by customer's technical experts. NXP does not convey any license under its patent rights nor the rights of others. NXP sells products pursuant to standard terms and conditions of sale, which can be found at the following address:

nxp.com/SalesTermsandConditions.

NXP, the NXP logo, NXP SECURE CONNECTIONS FOR A SMARTER WORLD, I2C BUS, Freescale, the Freescale logo, the Energy Efficient Solutions logo, and Kinetis are trademarks of NXP B.V. All other product or service names are the property of their respective owners.

ARM, the ARM powered logo, and Cortex are registered trademarks of ARM Limited (or its subsidiaries) in the EU and/or elsewhere. All rights reserved.

© 2016 NXP B.V.

Document Number: IMX7DSHDG

Rev. 0

07/2016

