Hardware Development Guide for i.MX 6QuadPlus, 6Quad, 6DualPlus, 6Dual, 6DualLite, 6Solo Families of Applications Processors

Document Number: IMX6DQ6SDLHDG
Rev. 4
02/2019

How to Reach Us:

Home Page:
nxp.com

Web Support:
nxp.com/support

Document Number: IMX6DQ6SDLHDG
Rev. 4, 02/2019

Contents

Contents

Contents

Contents

Chapter 1
About This Book

1.1 Overview

This document's purpose is to help hardware engineers design and test their i.MX 6 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 properly configuring JTAG tools, 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 www.nxp.com.

1.2 Devices supported

This Hardware Developer's Guide currently supports the i.MX 6QuadPlus, 6Quad, 6DualPlus, 6Dual, 6DualLite, and 6Solo families of application processors.

1.3 Essential reference

This guide is intended as a companion to the i.MX 6 series chip reference manuals and data sheets. For reflow profile and thermal limits during soldering, see application note AN3300, "General Soldering Temperature Process Guidelines." These documents are available on www.nxp.com.

1.4 Suggested reading

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

1.4.1 General Information

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

For information about the ARM Cortex-A9 processor see:
http://www.arm.com/products/processors/cortex-a/cortex-a9.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

1.5 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 www.nxp.com.

1.6 Conventions

This document uses the following notational conventions:

`Courier`	Used to indicate commands, command parameters, code examples, and file and directory names.
Italics	*Italics indicates* 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
*nnnn nnnn*h	Denotes hexadecimal number
0b	Denotes binary number
rA, **r**B	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.
x	An italicized *x* indicates an alphanumeric variable
n, m	An italicized *n* indicates a numeric variable

NOTE

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

1.7 Signal conventions

$\overline{\text{PWR\_ON\_RESET}}$	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

1.8 Acronyms and abbreviations

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

Table 1-1. Definitions and acronyms

Term	Definition
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

Table 1-1. Definitions and acronyms (continued)

CSI	MIPI camera serial interface
DDC	VESA Data Display Channel
DDR	Dual data rate DRAM
DDR3	DDR3 DRAM
DDR3L	Low voltage DDR3 DRAM
DRAM	Dynamic random access memory
DSI	MIPI display serial interface
ECSPI	Enhanced Configurable SPI peripheral
EDID	Extended Display Identification Data
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)
FSL	Freescale Semiconductor
GND	Ground
GPC	General Power Controller
GPIO	General-purpose input/output
HDCP	High-bandwidth Digital Content Protection
HDMI	High-definition multimedia interface
I2C	Inter-integrated circuit interface
IBIS	Input output buffer information specification
IOMUX	i.MX6 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
LPDDR2	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

Table 1-1. Definitions and acronyms (continued)

PCIe	PCI Express
PCISig	Peripheral Component Interconnect Special Interest Group
PMIC	Power management integrated circuit
PoP	Package-on-package
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
SATA	Serial ATA
SDMA	Smart Direct Memory Access Controller
UART	Universal asynchronous receiver/transmitter
USB	Universal Serial Bus
USB OTG	USB On-the-go
USB2.0	USB version 2.0 peripheral
VPU	Video processing units

Chapter 2
Design Checklist

2.1 Design checklist overview

This chapter provides a design checklist for the following i.MX 6 series families of processors:

- i.MX 6QuadPlus
- i.MX 6Quad
- i.MX 6DualPlus
- i.MX 6Dual
- i.MX 6DualLite
- i.MX 6Solo

The design checklist tables (Table 2-1–Table 2-14) 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.

See also the application note *Common Hardware Design for i.MX 6Dual/6Quad and i.MX 6Solo/6DualLite* (document AN4397) and *i.MX 6Dual/6Quad to i.MX 6DualPlus/6QuadPlus Migration Guide* (document EB810).

2.2 Design checklist tables

Table 2-1. DDR recommendations

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.	• The user may tie DDR_VREF to a precision external resistor divider. Shunt the resistor between DRAM_VREF and ground with a closely-mounted 0.1 μF capacitor to ground. NXP designs are migrating to using only a single capacitor. See Table 2-15 for resistor values. Using resistors with recommended tolerances ensures the ±2% DDR_VREF tolerance per the DDR3 specification. • The user can use a PMIC's tracking regulator as used on NXP reference designs. A tracking regulator is recommended as a reference for memory configurations of more than four devices.

Table 2-1. DDR recommendations (continued)

Checkbox	Recommendation	Explanation/supplemental recommendation
	3. Connect DRAM_RESET to a 10 kΩ 5% pulldown resistor to GND.	• 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.	• 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.[1] • For DDR3: SDCKE[1:0] pull-down is not required to meet JEDEC.
	5. Make sure that the correct LPDDR2 function is connected to the correct I/O. Note that this does not necessarily correspond to the I/O name.	MMDC IO names are for the DDR3 default. When LPDDR2 is selected, the I/O name (DDR3 MMDC PAD) does not match with the LPDDR2 functionality. See the "LPDDR2 and DDR3 pin mux mapping" table in the "Multi Mode DDR Controller (MMDC)" chapter in the appropriate reference manual.
	6. In a 64-bit LPDDR2 memory design, include EIM_A20 and EIM_A21 pins in the boot mode switch the matrix (CFG3[4] and CFG3[5]) to allow the selection of either the dual channel mode or the interleaved mode during development.	Include EIM_A20 and EIM_A21 in the boot mode switch matrix (see Figure 2-1) if a 64-bit LPDDR2 is used.

[1] Note that the SDCKE signals for use with the LPDDR2 are muxed out on these DRAM IO PADS: DRAM_ADDR07, DRAM_ADDR09, DRAM_SDWE, and DRAM_CS0. See the relevant Reference Manual MMDC chapter for more details.

Table 2-2. EIM recommendations for developer's boot modes

Checkbox	Recommendation	Explanation/supplemental recommendation
	1. When EIM boot signals are used as the system's EIM 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, EIM bus loads can cause current drain, leading to higher (false) supply current measurements. Each EIM boot signal should connect to a series resistor to isolate the bus from the resistors and/or switchers; see Figure 2-1. Each configured EIM boot signal sees either a 14.7 kΩ pulldown or a 4.7 kΩ pullup. For each switch-enabled pulled-up signal, the supply is presented with a 10 kΩ current load.
	2. Include EIM_A20 and EIM_A21 in the boot mode switch matrix (see Figure 2-1) if 64-bit LPDDR2 is being used.	In a 64-bit LPDDR2 memory design, EIM_A20 and EIM_A21 pins must be included in the boot mode switch matrix (CFG3[4] and CFG3[5]) in order to allow the selection of either dual channel mode or interleaved mode during development.

Table 2-2. EIM recommendations for developer's boot modes (continued)

Checkbox	Recommendation	Explanation/supplemental recommendation
	3. To reduce incorrect boot-up mode selections, do one of the following: • Use EIM boot interface lines only as processor outputs. Ensure EIM 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 EIM boot signal must be configured as an input, isolate the EIM 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 EIM 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 EIM signal to have an external or on-chip resistor to minimize signal floating. If the usage of the EIM 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.
	4. If using GPIO override instead of fuses, the BOOT_CFG signals are required for proper functionality and operation and should not remain unconnected.	See the "System Boot" chapter in the appropriate reference manual for the correct boot configuration. Note that an incorrect setting may result from an improper booting sequence.

Table 2-3. 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: • Tie to GND through any size external resistor • Tie directly to GND • Leave unconnected For logic 1, use one of the following: • 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 kΩ to 10 kΩ 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 kΩ, a projected minimum of 60 kΩ, and a projected maximum of 140 kΩ. 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 2-4. I²C recommendations

Checkbox	Recommendation	Explanation/supplemental recommendation
	1. Verify the target I²C 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 I²C port.
	2. Verify that the target I²C address range is supported and does no conflict with other peripherals. If there is an unavoidable address conflict, move the offending device to another I²C port. See Table 2-16.	These chips support up to: • Three I²C ports for the i.MX 6QuadPlus, 6Quad, 6DualPlus, and 6Dual families. • Four I²C ports for the i.MX 6DualLite and 6Solo families. If it is undesirable to move a conflicting device to another I²C 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 I²C lines.	This can result in excessive loading. Good design practice is to place one pair of pullups only.

Table 2-5. 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 2-19 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 2-19 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 $\leq$ 4.7 kΩ.	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 2-6. Power and decouple 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: • Excessive current during power-up phase • Prevention of the device from booting • Irreversible damage to the processor (worst-case scenario)
	2. Comply with the data sheet guidelines for the i.MX LDO setpoints on VDDARM_CAP/VDDARM23_CAP, VDDSOC_CAP, and VDDPU_CAP for the desired application requirements.	Incorrect LDO configuration such as an LDO setpoint below specification can reduce voltage margin which can lead to application crashes or incorrect functionality. An LDO setpoint above specification can cause damage to the device (worst case scenario).
	3. 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: • POR_B – on-chip pullup; see Table 2-8 #1 • ONOFF – on-chip pullup; see Table 2-8 #2 • BOOT_MODE0 – on-chip pulldown; see Table 2-3 #1 • BOOT_MODE1 – on-chip pulldown; see Table 2-3 #1 • TAMPER – on-chip pulldown • PMIC_STBY_REQ – open-drain output • PMIC_ON_REQ – open-drain output, on-chip 100 kΩ pull-up • TEST_MODE – on-chip pulldown; see Table 2-14 #1 If unused, the following signals should be left unconnected: ONOFF, TAMPER, PMIC_STBY_REQ and PMIC_ON_REQ	NXP PMIC PMPF0100 VSNVS regulator is rated to supply 400 μA 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.MX6: • 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. • 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.

Table 2-6. Power and decouple recommendations (continued)

Checkbox	Recommendation	Explanation/supplemental recommendation
	4. Only one 22 μF bulk capacitor should be connected to each of these on-chip LDO regulator outputs: • VDD_ARM_CAP • VDDARM23_CAP • VDD_SOC_CAP • VDD_PU_CAP A 22 μF bulk capacitor must be placed as near as possible with pins/vias. The distance should be less than 50mil between bulk cap and VDD_xx_CAP pins. Decoupling capacitors such as 0.1 μF or 0.22 μF should also be used. **Note:** For 6QuadPlus and 6DualPlus designs either two 22 μF or a single 47 μF bulk capacitor should be connected to VDD_PU_CAP to support the additional current supply required for the increased graphics performance.	If the nominal capacitance value is larger than recommended, power-up ramp time is excessive and operation cannot be guaranteed. Note that the ramp up time is constant. Larger capacitors mean more inrush current. Select small capacitors with low ESR (equivalent series resistance). Do not connect any loads to these LDO outputs: VDDARM_CAP, VDDARM23_CAP, or VDDPU_CAP. VDDSOC_CAP is restricted to MX6 loads.
	5. Only one 10 μF bulk capacitor should be connected to each of these on-chip LDO regulator outputs: • VDD_HIGH_CAP • NVCC_PLL_OUT • VDD_USB_CAP Decoupling capacitors such as 0.1 μF or 0.22 μF should also be used.	If the nominal capacitance value is larger than recommended, power-up ramp time is excessive and operation cannot be guaranteed. Select small capacitors with low ESR. These LDOs should only be used to power the loads as described in the reference manual or data sheet. Do not connect any loads to these LDO outputs: NVCC_PLL_OUT or VDDUSB_CAP. VDDHIGH_CAP is restricted to MX6 loads.
	6. One 0.22 μF decoupling capacitor should be connected to VDD_SNVS_CAP, an on-chip LDO regulator output. A bulk capacitor is not necessary.	If the nominal value is larger than recommended, power-up/down ramp time is excessive and suspend/resume operation cannot be guaranteed. Select a small capacitor with low ESR. **Note:** Do not connect any loads to VDD_SNVS_CAP.
	7. VDD_SOC_IN requires a 66 μF bulk capacitance.	This capacitance is required because it supplies power for both VDD_SOC_CAP and VDD_PU_CAP.
	8. 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.
	9. NVCC_LVDS2P5 must be powered-on even when not using the LVDS interface.	The DDR pre-drivers (DRAM and RGMII interfaces) share the NVCC_LVDS2P5 power rail with the LVDS interface. VDDHIGH_CAP can be utilized as the power source; tie NVCC_LVDS2P5 to VDDHIGH_CAP.

Table 2-6. Power and decouple recommendations (continued)

Checkbox	Recommendation	Explanation/supplemental recommendation
	10. Account for the different power design on NVCC_EIM between i.MX 6Quad and 6Dual chips and i.MX 6DualLite and 6Solo chips. **Note:** i.MX6QuadPlus and i.MX6DualPlus are configured the same as i.MX6Quad and i.MX6Dual.	• i.MX 6Quad and 6Dual chips can support three different EIM power rail voltage levels: NVCC_EIM0(K19), NVCC_EIM1(L19), and NVCC_EIM2(M19). • i.MX 6DualLite and 6Solo chips support one EIM power rail: NVCC_EIM (K19, L19, M19). The three power contacts must be connected to same power supply.
	11.i.MX 6Dual/6Quad only: NXP simulations indicate a minimum of 85 mohm (milli-ohm) ESR (Equivalent Series Resistance) bulk capacitor is required to satisfy stability requirements over process and temperature conditions for the VDD_HIGH 2.5V regulator (LDO_2P5). If the ESR of the Bulk Capacitor does not meet this requirement, a series resistor should be added to the Bulk Capacitor." i.MX 6DualPOP/6QuadPOP only: NXP simulations indicate a minimum of 45 mohm (milli-ohm) ESR bulk capacitor is required to satisfy stability requirements over process and temperature conditions for the VDD_HIGH 2.5V regulator. If the ESR of the Bulk Capacitor does not meet this requirement, a series resistor should be added to the Bulk Capacitor..	Existing designs that do not demonstrate stability problems on this supply have enough resistance in the ESR of the bulk capacitance or the board traces that this resistor is not required. New designs or design exhibiting stability problems on this supply should add the resistor. See *Minimum/Maximum ESR Requirement for i.MX 6Dual/6Quad Analog Regulators* (document EB814) for additional information
	12. 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 is not needed during system power-down.	If no power is supplied to VDD_SNVS_IN (due to a fault or a discharged coin cell), the diode ensures that VDD_SNVS_IN is powered whenever VDD_HIGH is powered to meet the datasheet specification.
	13. If boundary scan test (BSDL) will be used, the following supplies must be powered: • PCIE_VP, PCIE_VPH, PCIE_VPTX • SATA_VP, SATA_VPH The SATA supplies do not apply to the i.MX 6DualLite/6Solo.	The boundary scan test scan chain runs through the PCIe and SATA PHYs. If the PCIe and SATA PHY supplies are not powered, the scan chain will not function.

Table 2-6. Power and decouple recommendations (continued)

Checkbox	Recommendation	Explanation/supplemental recommendation
	14. When using a PF0100 PMIC to charge a coin cell on VDD_SNVS_IN, a current-limiting series resistor is required between the PMIC output and VDD_SNVS_IN.	If the PMIC power is removed or failing and the battery is discharged, the PMIC will switch over to the battery when the PMIC voltage falls below approximately 2.8V. This will cause an inrush current from the i.MX6 VDD_SNVS_IN pin to the battery (which is at a lower voltage). This current path is a combination of VDD_HIGH and the SNVS_CAP power sources. The i.MX6 must have a VDD_HIGH voltage greater than approximately 2.4V to enable the VDD_HIGH to SNVS charging path. This instantaneous current flow from VDD_SNVS_IN can cause damage to the i.MX6. The current limiting series resistor should be sized to match the charging current limits of the battery.
	15. Minimum 85mohm external ESR is required for VDD_HIGH_CAP(H10,J10).	See *Minimum/Maximum ESR Requirement for i.MX 6Dual/6Quad Analog Regulators* (document EB814) for details.

Table 2-7. 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 kΩ ESR (equivalent series resistance) and follow the manufacturer's recommendation for loading capacitance. Do not use an external biasing resistor because the bias circuit is on-chip. Recommended crystal parameters are included below: • Typical Cload—10 pF • Maximum drive level—1 μW • Typical C0—1 pF	The capacitors implemented on either side of the crystal are about twice the crystal load capacitance. To hit the target oscillation frequency, board capacitors need to be reduced to compensate for board and chip parasitic capacitance; typically 10–16 pF is employed. 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 MΩ) 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, left unconnected, 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 6 series datasheet available at www.nxp.com. Note that if this external clock is stopped, the internal ring oscillator starts automatically.

Table 2-7. Oscillator and clock recommendations (continued)

Checkbox	Recommendation	Explanation/supplemental recommendation
	3. Loose-tolerance 40 kHz oscillator An on-chip loose-tolerance ring oscillator is available of approximately 40 kHz. If RTC_XTALI is tied to GND and RTC_XTALO is left unconnected, the on-chip oscillator is automatically engaged.	When a high-accuracy real-time clock is not required, the system may use the on-chip 40 kHz oscillator. The tolerance is ± 50%. The ring oscillator starts faster than an external crystal and is used until the external crystal reaches stable oscillation. The ring oscillator also starts automatically if no clock is detected at RTC_XTALI at any time.
	4. Precision 24 MHz oscillator Connect a fundamental-mode crystal between XTALI and XTALO. An 80 Ω typical ESR crystal rated for a maximum drive level of 250 μW is acceptable. Alternately, a 50 Ω typical ESR crystal rated for a maximum drive level of 200 μW may be used.	NXP BSP software requires 24 MHz on this clock. This clock is used as a reference for USB, PCIe, and SATA, so there are strict frequency tolerance and jitter requirements. See Table 2-20 for guidelines. See the crystal oscillator (XTALOSC) reference manual chapter and relevant interface specification chapters for details. To access a calculator for the 24 MHz crystal drive level, see EB830 on the i.MX Community.
	5. External megahertz source If feeding an external clock into the device, XTALI can be driven DC-coupled with XTALO left unconnected.	For XTALI VIL and VIH voltage levels, see the latest i.MX 6 series datasheet. This clock is used as a reference for USB, PCIe, and SATA, so there are strict frequency tolerance and jitter requirements. See Table 2-20 for guidelines. See the crystal oscillator (XTALOSC) reference manual chapter and relevant interface specification chapters for details.
	6. CLK1_P/CLK1_N and CLK2_P/CLK2_N are LVDS input/output differential pairs compatible with TIA/EIA-644 standard. The frequency range is 0 to 600 MHz. Alternatively, a single-ended signal can be used to drive a CLKx_P input. In this case, the corresponding CLKx_N input should be tied to a constant voltage level equal to 50% of VDD_HIGH_CAP. Termination should be provided with high-frequency signals. See the LVDS pad electrical specification in the data sheet for further details. After initialization, the CLKx inputs/outputs can be disabled (if not used) by the PMU_MISC1 register. If unused, any or both of the CLKx_N/P pairs may be left unconnected.	The clock inputs/outputs are general-purpose differential high-speed clock Input/outputs. Any or both of them can be configured: • As inputs to feed external reference clocks to the on-chip PLLs and/or modules, for example as alternate reference clock for PCIe or/and SATA or video/audio interfaces. • As outputs to be used as either a reference clock or as a functional clock for peripherals. See the chip reference manual for details on the respective clock trees. When using the differential clock outputs in differential mode, a 100 Ohm 1% resistor should be placed across the CLK1_P/CLK1_N, or the CLK2_P/CLK1_N pins. The load resistor is required for the LVDS driver to properly output the correct signal. If decoupling capacitors are used in the differential traces, they should be placed after the load resistor, and not between the CLK pins and the load resistor. The clock signal is generated by passing current through the load resistor.
	7. XTALI must be biased with a 2.2 MΩ resistor to GND. Mount the resistor close to the XTALI ball.	The XTALI bias must be adjusted externally to ensure reasonable start-up time. Without the resistor, start-up time may be 200 ms or may not start at all.

Table 2-8. Reset and ONOFF 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, VDD_SOC_CAP, and VDD_PU_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 2-9. Gigabit Ethernet Recommendations

Checkbox	Recommendation	Explanation/supplemental recommendation
	1. External ENET clock source option: A 125 MHz reference clock is required to feed the ENET_REF_CLK input. This reference clock can be sourced from an external 125 MHz oscillator or an external PHY. Designers should be aware of the 125 MHz reference output level of the PHY because ENET_REF_CLK is on the NVCC_ENET supply rail, not the NVCC_RGMII rail.	For IEEE-1588 timestamp operation, the GPIO_16 ball must be configured as one of the following: • A no connect to allow the internal time stamp clock to route through its IOMUX cell to the RGMII interface; program GPR1[21] = 1. • Driven by an external clock source for the time stamp; GPR1[21] = 0.
	2. On-chip ENET clock source option: A 125 MHz reference clock (derived from PLL6) must feed the ENET_REF_CLK input. This reference clock can be sourced from an on-chip 125 MHz reference. In this case, output GPIO_16 must be externally connected to input ENET_REF_CLK via a short printed circuit board trace. **Note:** i.MX6QuadPlus and i.MX6DualPlus designs do not require this external PCB connection. The internal clock routing meets all the ENET clocking requirements.	Configuration for on-chip reference clock with IEEE-1588 timestamp operation: • GPIO_16 = ALT2 which is ENET_REF_CLK output • Register IOMUXC_ENET_REF_CLK_SELECT_INPUT = GPIO16_ALT2 • GPR1[21] = 1 to select internal reference clock • GPIO_16 SION = 1 to route clock for IEEE-1588 timestamp • ENET_REF_CLK = ALT1 which is ENET_TX_CLK reference input NXP strongly suggests that users validate their designs over temperature while running their system software. Noisy system conditions may dictate use of an external reference oscillator. ENET_REF_CLK is used as a clock source for MII and RGMII modes only. RMII mode uses either GPIO_16 or RGMII_TX_CTL as a clock source. For more infromation on these clocks, see the Reference Manuals for i.MX 6Quad, 6QuadPlus, 6Dual, 6DualPLus, 6DualLite, 6Solo families of applications processors.

Table 2-10. PCIe recommendations

Checkbox	Recommendation	Explanation/supplemental recommendation
	1. An external PCIe reference clock generator is recommended. i.MX differential clock is not compliant with PCIe standard.	i.MX differential clock does not meet PCIe compliance standards.
	2. The differential transmitter must be ac coupled. Use a 0.1 uF-series capacitor on PCIE_TXP and a second 0.1 uF on PCIE_TXM.	To ensure PCIe specification compliance, ac coupling is required at each transmitter. The receiver must be dc coupled.

Table 2-11. HDMI recommendations

Checkbox	Recommendation	Explanation/supplemental recommendation
	1. The designer must ensure that a suitable level shifter and driver be used to interface the chip's I^2C with the HDMI monitor. In addition, ESD (electrostatic discharge) protection must be used on all HDMI single-ended and differential signals mounted near the board's HDMI connector.	The i.MX 6 processors' I^2C cannot operate at the 5 V required by HMDI EDID. The i.MX 6 processors' supply limit is 3.6 V maximum. The designer could consider the ON Semiconductor CM2020 for ESD protection and I^2C level conversion. **Note:** NXP cannot recommend one supplier over another and does not suggest that this is the only HDMI interface chip supplier.
	2. DDC (EDID) must be on a dedicated I^2C (DDC_SCL/DDC_SDA) port when HDCP (High-Bandwidth Digital Content Protection) is enabled.	When HDCP is enabled, a dedicated I^2C is controlled by the HDMI PHY to exchange the HDCP encryption key and must sync several times per second. DDC does not behave like a common I^2C and cannot be controlled by the ARM® CPU with HDCP enabled.

Table 2-12. 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 *off* when the boards power up.	The processor should turn VBUS *on* as required.
	2. USB Host USB_H1_VBUS should be directly connected to a 5 V supply.	Tie USB_H1_VBUS to an unswitched 5 V supply for the typical use case. This power connection feeds the on-chip USB LDO regulator.
	3. USB OTG and Host External ESD (electrostatic discharge) protection circuits should be employed on USB_OTG_DN, USB_OTG_DP, USB_OTG_ID, OTG_VBUS, USB_H1_DP, USB_H1_DN, and USB_H1_VBUS. Clamp VBUS at approximately 5.25 V. Preferred clamp for the other signals is approximately 3.3-3.5 V. Refer to the Absolute Maximum Ratings table in the data sheet.	—
	4. VDD_USB_CAP Adopters should consider adding a zener diode or clamp to VDDUSB_CAP, in parallel with the required capacitors of Table 2-6 #4. 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 2-13. Reference resistor recommendations

Checkbox	Recommendation	Explanation/supplemental recommendation
	1. HDMI_REF – Connect an external 1.6 kΩ 1% resistor to GND.	If HDMI is unused, the reference resistor may be populated if desired for manufacturability purposes, or left no-connect for cost savings.
	2. SATA_REXT – Connect an external 191 Ω 1% resistor to GND.	The impedance calibration process requires connection of this reference resistor. If SATA is unused, the reference resistor may be populated if desired for manufacturability purposes, or left no-connect for cost savings.
	3. PCIE_REXT – Connect an external 200 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.
	4. CSI_REXT – Connect an external 6.04 kΩ 1% resistor to GND.	If CSI is unused, the reference resistor may be populated if desired for manufacturability purposes, or left no-connect for cost savings.
	5. DSI_REXT – Connect an external 6.04 kΩ 1% resistor to GND.	If DSI is unused, the reference resistor may be populated if desired for manufacturability purposes, or left no-connect for cost savings.

Table 2-14. Miscellaneous recommendations

Checkbox	Recommendation	Explanation/supplemental recommendation
	1. The TEST_MODE input is internally connected to an on-chip pulldown device. The user can either tie this signal to Vss or leave it unconnected.	This input is reserved for NXP manufacturing use.
	2. For termination of unused analog interfaces, see Table 2-21.	—
	3. VDD_FA and FA_ANA should be tied to GND.	These inputs are reserved for NXP manufacturing use. Best practice is to tie them to ground to avoid floating inputs.
	4. GPANAIO must be a no connect.	This output is reserved for NXP manufacturing use.
	5. NC contacts are no connect and must be left unconnected.	Depending on the feature set, some versions of the IC may have NC contacts connected inside the BGA.
	6. Only the i.MX 6Solo and 6DualLite support Manufacture Mode.	See the System Boot chapter, section 'SD/MMC Manufacture Mode' in the *i.MX 6Solo/6DualLite Applications Processor Reference Manual* (document IMX6SDLRM).

2.3 Bus isolation circuit

The following figure provides supporting information for Table 2-2, recommendation #1

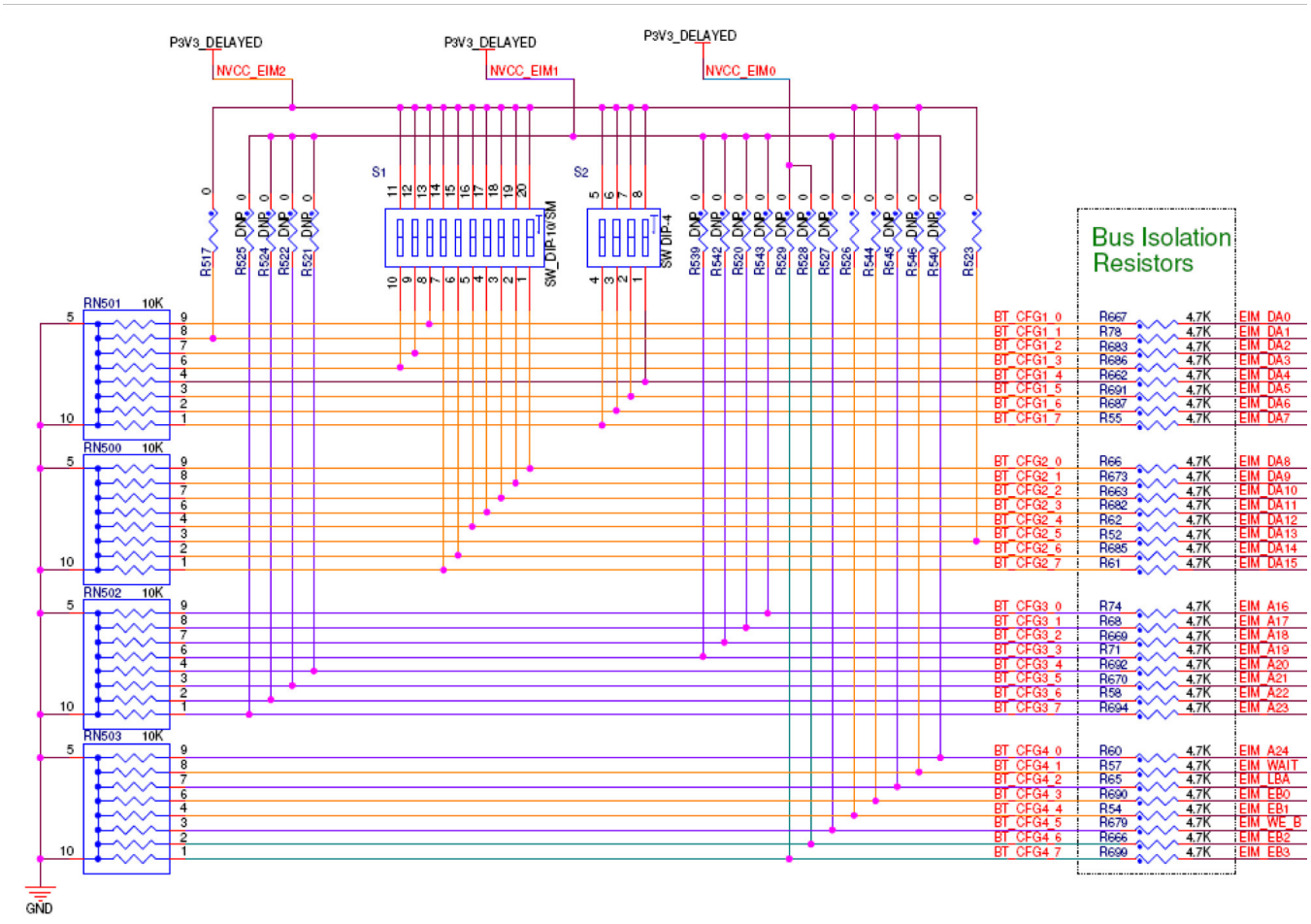


Figure 2-1. Boot configuration for development mode

2.4 DDR reference circuit

The following table is a resistor chart (see Table 2-1 recommendation #2). The recommendations are appropriate for designs with DDR memory chips with a maximum Vref input current of 2µA each.

Table 2-15. DDR Vref resistor sizing guideline

Number of DRAM Vref loads (2 µA each)	Resistor divider value (2 resistors)
2	≤1.21 kΩ 1%
2	≤1.54 kΩ 0.5%

Table 2-15. DDR Vref resistor sizing guideline (continued)

Number of DRAM Vref loads (2 μA each)	Resistor divider value (2 resistors)
2	≤2.32 kΩ 0.1%
4	≤768 Ω 1%
4	≤1 kΩ 0.5%
4	≤1.5 kΩ 0.1%

2.5 I²C address

The following table shows a spreadsheet approach to avoid I²C conflicts as referenced in Table 2-4 recommendation #2.

NOTE

The example in this section only applies to the NXP reference design board.

Notice that although there are no slave address conflicts, the shaded cell in the table highlights a potential bus speed issue as discussed in Table 2-4. The AM-FM tuner limits the maximum bus rate to 250 kbps, but the bus data rate cannot exceed the slowest peripheral on the bus.

If the system cannot tolerate the 250 kbps rate for proper operation, the AM-FM tuner must be moved to another I²C port. If the I²C bus rate exceeded the AM-FM tuner module's maximum bus rate, the I2C bus operation might fail or become unpredictable. The slow peripheral may unpredictably take over the bus or might malfunction in some other way.

Table 2-16. I²C bus example spreadsheet

Peripheral	Bus activity level	Speed (kbps)	Slave addresses supported on the peripheral (hex)	Selected system address (hex)
PMIC	Low	400	68	68
Port Expander	Low	400	30, 32, 34	30
AM-FM Tuner	Med	250	C0, C2, C4, C6	C0
A/D Converter	Med	400	40, 42	40
Audio CODEC	Low	400	90, 92, 94, 96	90

Assuming the system can function properly with a reduced bus rate of 250 kbps, the following table provides a possible I^2C port usage table.

Table 2-17. I^2C port usage scenario

i.MX6 I^2C ports	Ball name	Function	Speed (kbps)
Port 1			
Port 1			
Port 2	KEY_ROW3	I2C2_SDA	250
Port 2	EIM_EB2	I2C2_SCL	250
Port 3			
Port 3			

2.5.1 I^2C clock speed and division factors (IFDR)

The I^2C clock is sourced from PERCLK_CLK_ROOT which is routed from IPG_CLK_ROOT. The I^2C clock frequency can be easily obtained using the following formula:

I^2C clock Frequency = (PERCLK_ROOT frequency)/(division factor corresponding to IFDR)

By default, the IPG_CLK_ROOT and PERCLK_CLK_ROOT frequencies are set to 49.5MHz, where the root clock is sourced from PLL2's PFD2. Obtaining the frequencies can be accomplished using the following:

PLL2 = 528MHz

PLL2_PFD2 = 528MHz * 18 / 24 = 396MHz

IPG_CLK_ROOT = (PLL2_PFD2 / ahb_podf)/ ipg_podf = (396MHz/4)/2 = 49.5MHz

PER_CLK_ROOT = IPG_CLK_ROOT/perclk_podf = 49.5MHz/1 = 49.5MHz

NOTE

The above calculation assumes that the default CCM register settings, routing, and division factors are used. If different routing, PFD values, and/or division factors are used, the user must adjust the parameters accordingly to calculate the correct clock frequency.

IFDR, division factor and resulting I^2C CLK frequencies are indicated in the table below. Resulting frequencies will vary according to the PERCLK_CLK_ROOT frequencies selected.

Table 2-18 assumes PERCLK_CLK_ROOT = 49.5MHz.

Table 2-18. IFDR[1]

IFDR	Division factor	Frequency (kHz)
0	30	1650
1	32	1546.875
2	36	1375
3	42	1178.571
4	48	1031.25
5	52	951.9231
6	60	825
7	72	687.5
8	80	618.75
9	88	562.5
A	104	475.9615
B	128	386.7188
C	144	343.75
D	160	309.375
E	192	257.8125
F	240	206.25
10	288	171.875
11	320	154.6875
12	384	128.9063
13	480	103.125
14	576	85.9375
15	640	77.34375
16	768	64.45313
17	960	51.5625
18	1152	42.96875
19	1280	38.67188
1A	1536	32.22656
1B	1920	25.78125
1C	2304	21.48438
1D	2560	19.33594
1E	3072	16.11328
1F	3840	12.89063

Table 2-18. IFDR[1] (continued)

20	22	2250
21	24	2062.5
22	26	1903.846
23	28	1767.857
24	32	1546.875
25	36	1375
26	40	1237.5
27	44	1125
28	48	1031.25
29	56	883.9286
2A	64	773.4375
2B	72	687.5
2C	80	618.75
2D	96	515.625
2E	112	441.9643
2F	128	386.7188
30	160	309.375
31	192	257.8125
32	224	220.9821
33	256	193.3594
34	320	154.6875
35	384	128.9063
36	448	110.4911
37	512	96.67969
38	640	77.34375
39	768	64.45313
3A	896	55.24554
3B	1024	48.33984
3C	1280	38.67188
3D	1536	32.22656
3E	1792	27.62277
3F	2048	24.16992

[1] Shaded cells indicate frequency is outside of the range that guarantees operation.

2.6 JTAG signal termination

The following table is a JTAG termination chart (see recommendations in Table 2-5).

Table 2-19. JTAG interface summary

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

2.7 Oscillator tolerance

The following table provides 24 MHz oscillator tolerance guidelines (see Table 2-7, recommendations #4 and #5). Because these are guidelines, the designer must verify all tolerances per the official specifications.

Table 2-20. 24 MHz crystal tolerance guidelines

Interface	Tolerance (± ppm)
Ethernet	50
HDMI	100
SATA	350
USB2.0	150

2.8 Unused analog interfaces

Table 2-21 shows the recommended connections for unused analog interfaces (see Table 2-14, recommendation #2).

Table 2-21. Recommended connections for unused analog interfaces

Module	Contact name	Recommendations if unused
CCM	CLK1_N, CLK1_P, CLK2_N, CLK2_P	Leave unconnected
CSI[1]	CSI_CLK0M, CSI_CLK0P, CSI_D0M, CSI_D0P, CSI_D1M, CSI_D1P, CSI_D2M, CSI_D2P, CSI_D3M, CSI_D3P, CSI_REXT	Leave unconnected
DSI[1]	DSI_CLK0M, DSI_CLK0P, DSI_D0M, DSI_D0P, DSI_D1M, DSI_D1P, DSI_REXT	Leave unconnected
HDMI	HDMI_CLKM, HDMI_CLKP, HDMI_D0M, HDMI_D0P, HDMI_D1M, HDMI_D1P, HDMI_D2M, HDMI_D2P, HDMI_DDCEC, HDMI_HPD, HDMI_REF	Leave unconnected
	HDMI_VP, HDMI_VPH	Ground
LDB	LVDS0_CLK_N, LVDS0_CLK_P, LVDS0_TX0_N, LVDS0_TX0_P, LVDS0_TX1_N, LVDS0_TX1_P, LVDS0_TX2_N, LVDS0_TX2_P, LVDS0_TX3_N, LVDS0_TX3_P, LVDS1_CLK_N, LVDS1_CLK_P, LVDS1_TX0_N, LVDS1_TX0_P, LVDS1_TX1_N, LVDS1_TX1_P, LVDS1_TX2_N, LVDS1_TX2_P, LVDS1_TX3_N, LVDS1_TX3_P	Leave unconnected
MLB	MLB_CN, MLB_CP, MLB_DN, MLB_DP, MLB_SN, MLB_SP	Leave unconnected
PCIe	PCIE_REXT, PCIE_RXM, PCIE_RXP, PCIE_TXM, PCIE_TXP	Leave unconnected
	PCIE_VP, PCIE_VPH, PCIE_VPTX	Ground[2]
RGMII	RGMII_RD0, RGMII_RD1, RGMII_RD2, RGMII_RD3, RGMII_RX_CTL, RGMII_RXC, RGMII_TD0, RGMII_TD1, RGMII_TD2, RGMII_TD3, RGMII_TX_CTL, RGMII_TXC	Leave unconnected
SATA[3]	SATA_REXT, SATA_RXM, SATA_RXP, SATA_TXM, SATA_TXP	Leave unconnected
	SATA_VP, SATA_VPH	Ground[2]
USB	USB_H1_DN, USB_H1_DP, USB_H1_VBUS, USB_OTG_CHD_B, USB_OTG_DN, USB_OTG_DP, USB_OTG_VBUS	Leave unconnected

1 NVCC_MIPI must remain powered even if the CSI and DSI interfaces are not used.

2 These supplies must remain powered if boundary scan test (BSDL) needs to be done.

3 SATA is not present on the i.MX 6Solo/6DualLite. The NC signals on the i.MX 6Solo/6DualLite that correspond to SATA supplies and signals on the i.MX 6Dual/6Quad require no special consideration. They are not connected inside the i.MX 6Solo/6DualLite package.

Chapter 3
i.MX 6 Series Layout Recommendations

This chapter provides recommendations to assist design engineers with the correct layout of their i.MX 6 series-based system. The majority of the chapter discusses the implementation of the DDR interface, but it also provides recommendation for power, the HDMI, SATA, LVDS, PCIe, USB, reference resistors, ESD and related emissions.

This chapter uses the i.MX6DQ SABRE SD board as its reference for illustrating the key concepts. See the i.MX6DQ SABRE SD board layout files as a companion to this chapter.

3.1 Introduction

This chapter provides recommendations to assist design engineers with the correct layout of their i.MX 6 series-based system. The majority of the chapter discusses the implementation of the DDR interface, but it also provides recommendation for power, the HDMI, SATA, LVDS, PCIe, USB, reference resistors, ESD and related emissions.

This chapter uses the i.MX6DQ SABRE SD board as its reference for illustrating the key concepts. See the i.MX6DQ SABRE SD board layout files as a comparison to this chapter.

3.2 Basic design recommendations

The i.MX 6Dual/6Quad processor comes in a 21 × 21 mm package with 0.8 mm ball pitch. The ball-grid array contains 25 rows and 25 columns, making it a 624 ball BGA package. For detailed information about the package, see the i.MX 6 series Consumer and Automotive datasheets.

The following figure shows the ball-grid array. Figure 3-2 shows additional package information.

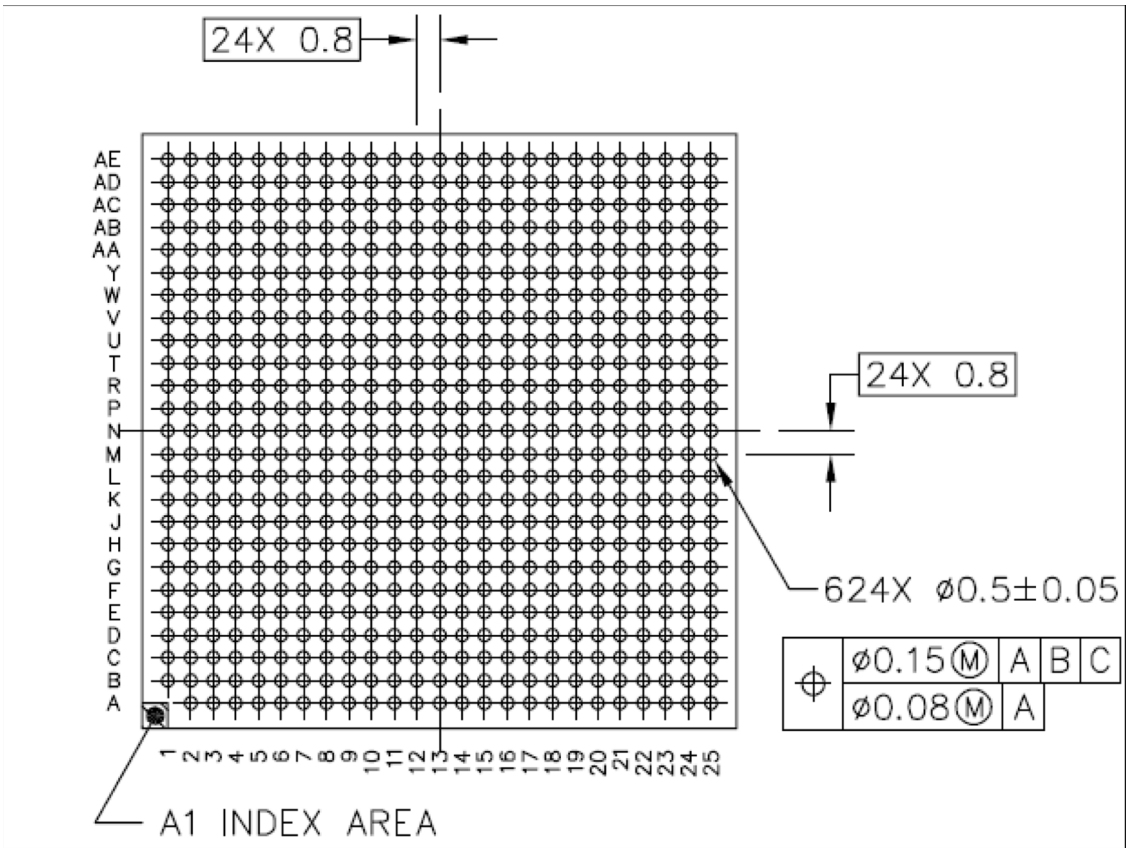


Figure 3-1. i.MX 6DPQP/6DQ/SDL ball-grid array

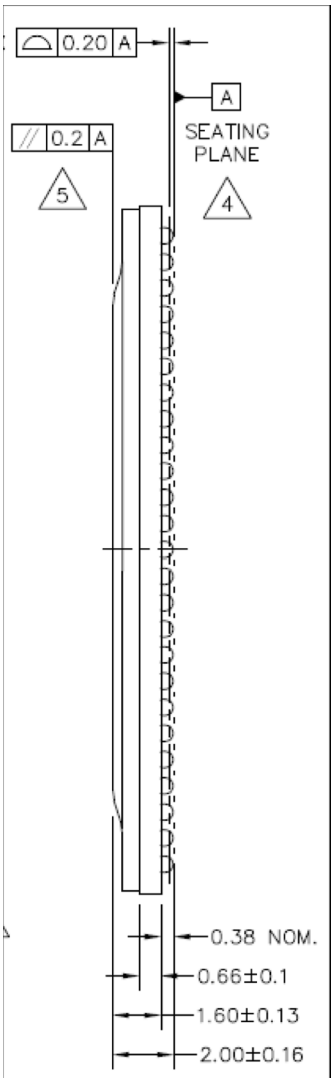


Figure 3-2. i.MX 6DPQP/6DQ/6SDL package information

It is critical to maintain the recommended footprint of a 16 mils pad with a 20 mil open solder mask for ease of fanout. In this case, the solder paste is the same as the pad with 16 mil, which allows an air gap of 15.496-mil between pads.

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 as per the package mechanical dimensions outlined in the product data sheet.

3.2.1 Fanout illustrations

The following figures show the top and bottom layer fanouts for the i.MX 6Dual/6Quad chip.

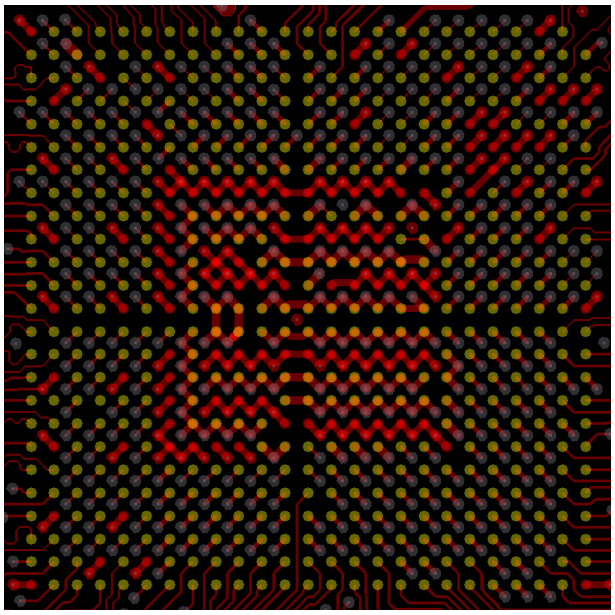


Figure 3-3. i.MX6DQ fanout example, top layer view

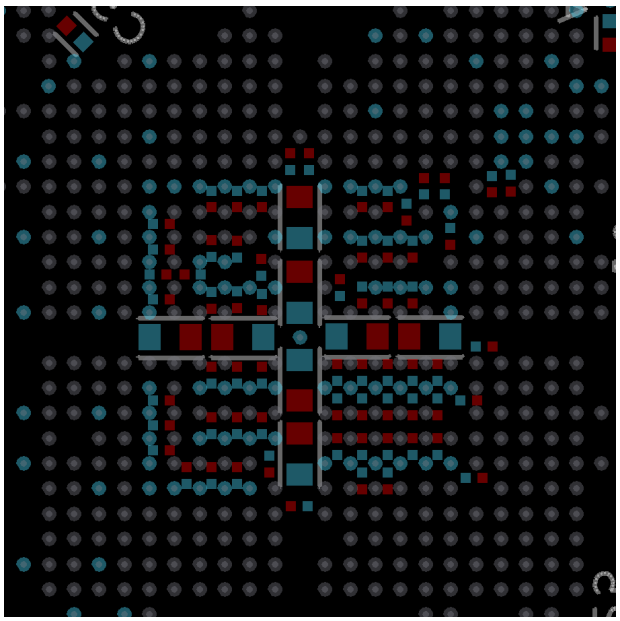


Figure 3-4. i.MX6DQ fanout example, bottom view

The colors signify the following:

- Top layer
  - Red = etch
  - Yellow = pad

  - Gray = vias
- Bottom layer
  - Cyan = GND net
  - Brown = power rails

3.2.2 Placing decoupling capacitors

The fanout scheme creates a four quadrant structure that facilitates the placement of decoupling and 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 for the i.MX6 family boards.

- 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 μF 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.

The i.MX6 SABRE SD (Smart Devices) CPU uses the preferred BGA power decoupling design. Note that the layout is available through www.nxp.com. Customers should use the reference design strategy for power and decoupling.

3.3 Stackup recommendations

High-speed design requires a good stackup in order to 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 stackup 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 stackup also determines the constraints for routing and spacing.

Consider the following when designing the stackup 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 stackup is 8-layers, with the layer stack as shown in the following figure. The lefthand image shows the detail provided by NXP inside the fabrication detail as a part of the Gerber files. The righthand side shows the solution suggested by the PCB fabrication company for our requirements.

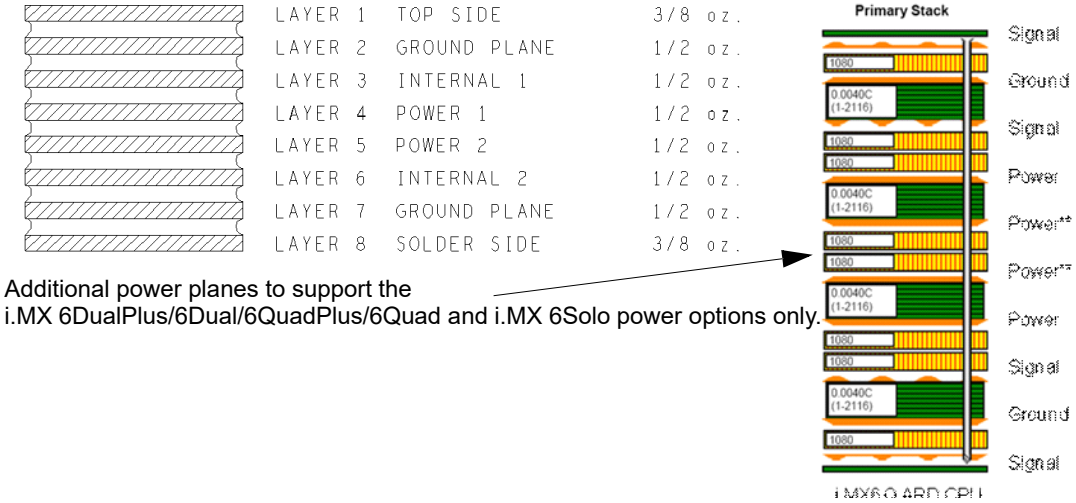


Figure 3-5. Layer stack SABRE SD board

The following table shows a working stack-up implementation:

Table 3-1. Stackup implementation

Layers	Single ended		Differential					
	Trace width (Mils)	Impedance (Ωs)	Trace width (Mils)	Trace spacing 'Airgap' (Mils)	Impedance (Ωs)	Trace width (Mils)	Trace spacing 'Airgap' (Mils)	Impedance (Ωs)
TOP	4.7	50	4.3	5.7	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	5.3	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 3-6. Example top layer impedance solution from PCB fabricator

3.4 DDR connection information

The following figures show the block diagram from the reference design boards for the DDR3 interface with the i.MX 6DualPlus/6Dual/6QuadPlus/6Quad/6DualLite/6Solo.

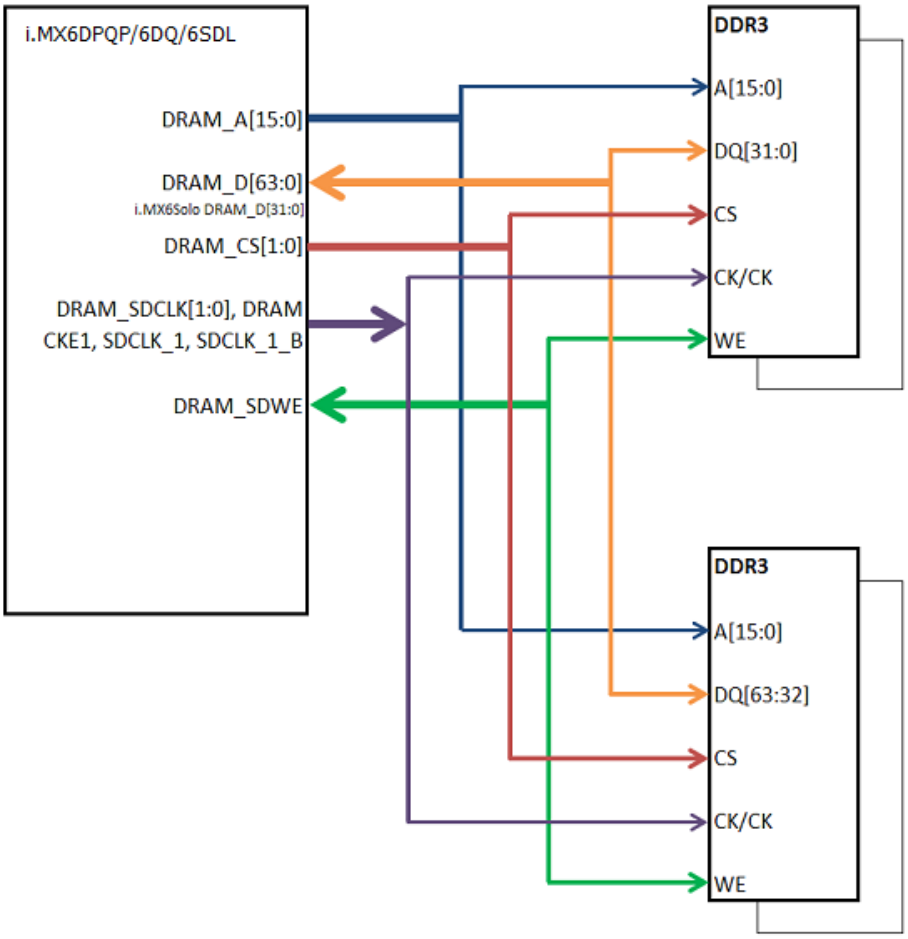


Figure 3-7. Connection between i.MX6DPQP/6DQ/6SDL and DDR3

The DDR3 interface is one of the most critical interfaces for chip routing. It must have controlled impedance for the single ended traces equal to 50 Ω and for the differential pairs 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.

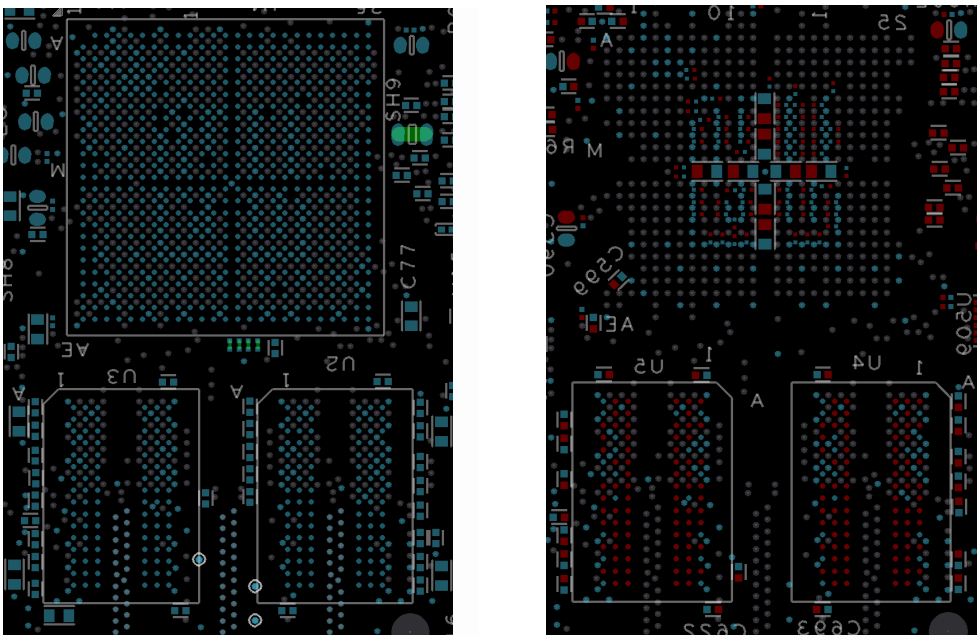


Figure 3-8. Final placement of memories and decoupling capacitors

3.5 DDR routing rules

DDR3 routing can be accomplished in two different ways: routing all signals at the same length or routing by byte group.

Routing all signals at the same length can be more difficult at first because of the tight space between the DDR and the processor and the large number of required interconnects. However, it is the better way because it makes signal timing analysis straightforward. Ideally, we could route all the signals at the same length, However, 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 3-2. DDR3 routing by the same length

Signals	Total length	Recommendations
Address and Bank	Clock length	Match the signals ±25 mils of the value specified in the length column
Data and Buffer	Clock length	
Control signals	Clock length	

Table 3-2. DDR3 routing by the same length (continued)

Signals	Total length	Recommendations
Clock DRAM_SDCLK[1:0]	Longest trace ≤ 3 inches	Match the signals of clocks signals ±5 mils. Each differential clock pair
DRAM_SDQS[7:0] and DRAM_SDQS[7:0]_B	Clock length	Match the signals of DQS signals ±10 mils of the value specified in the length column.

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 3-3. 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	2.25 inches	Match the signals ± 5 mils. 2.25 inches is recommended.
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 ± 25 mils.

Table 3-3. DDR3 routing by byte group (continued)

Chip signals	Group	Length		Recommendations
		Min	Max	
DRAM_D[7:0] DRAM_DQM0 DRAM_SDQS0 DRAM_SDQS0_B	Byte Group 1	—	Clock (min)	Match the signals of each byte group ± 25 mils. Match the differential signals of DQS ± 10 mils.
DRAM_D[15:8] DRAM_DQM1 DRAM_SDQS1 DRAM_SDQS1_B	Byte Group 2	—	Clock (min)	
DRAM_D[23:16] DRAM_DQM2 DRAM_SDQS2 DRAM_SDQS2_B	Byte Group 3	—	Clock (min)	
DRAM_D[31:24] DRAM_DQM3 DRAM_SDQS3 DRAM_SDQS3_B	Byte Group 4	—	Clock (min)	
DRAM_D[39:32] DRAM_DQM4 DRAM_SDQS4 DRAM_SDQS4_B	Byte Group 5	—	Clock (min)	
DRAM_D[47:40] DRAM_DQM5 DRAM_SDQS5 DRAM_SDQS5_B	Byte Group 6	—	Clock (min)	
DRAM_D[55:48] DRAM_DQM6 DRAM_SDQS6 DRAM_SDQS6_B	Byte Group 7	—	Clock (min)	
DRAM_D[63:56] DRAM_DQM7 DRAM_SDQS7 DRAM_SDQS7_B	Byte Group 8	—	Clock (min)	
DRAM_CS[1:0] DRAM_SDCKE[1:0] DRAM_SDODT[1:0]	Control signals	Clock (min) – 200	Clock (min)	Match the signals ± 50 mils.

1. Clock (min)—The shortest length of the clock group signals because this group has a ± 5 mil matching tolerance.

Finally, the impedance for the signals should be 50 Ω for single ended and 100 Ω for differential pairs.

3.6 Routing considerations

The chip can handle up to 4 Gbytes of DRAM memory. i.MX6 DDR routing needs to be separated into three groups: data, address, and control. Each group has its own method of routing from an i.MX 6 series processor to DDR memory. The DDR layout has 2 Gbyte and 4 Gbyte options.

3.6.1 Swapping data lines

The DDR3 pin swapping technique for the data bus lines within bytes makes it easier to:

- Route direct lines
- Avoid changes between layers

The rules are as follows:

- Hardware write leveling – lowest order bit within byte lane must remain on lowest order bit of lane by JEDEC compliance (see the “Write Leveling” section in JESD79-3E)
  - D0, D8, D16, D24, D32, D40, D48, and D56 are fixed
  - Other data lines free to swap within byte lane
- JEDEC DDR3 memory restrictions are:
  - No restrictions for complete byte lane swapping
  - DQS and DQM must follow lanes

NOTE

If byte lane swapping was done, target DDR IC register read value must be transposed according to the data line swapping.

3.6.2 DDR3 (64 bits) T topology considerations

Be sure to take into account the following when designing a T-topology system.

- Follow the routing rules described in Table 3-3.
- Termination resistors not required.
- Short routing lengths and on-chip drive strength control.
- Your design is limited to 4 DDR chips.
- DDR3, 2 GBytes using latest memories (4 GBytes available now).

3.6.3 DDR3 (64 bits) Fly-by topology considerations

Pay attention to the following recommendations when the Fly-by topology and routing technique is used.

- DDR controller provides address mirroring when using two chip selects, which aids address line routing for memories on both sides of board.
- Bus termination resistors are required.

3.6.4 2-Gigabyte recommendations

The 2 Gbyte option has four memories. You should follow these recommendations for best practice:

- Have a balanced routing for the T connection.
- Avoid having many layer transitions.
- Do not cross split reference planes during the routing.

The topology for the ADDR/CMD/CTRL signals has a tree topology. Note the balanced T routing.

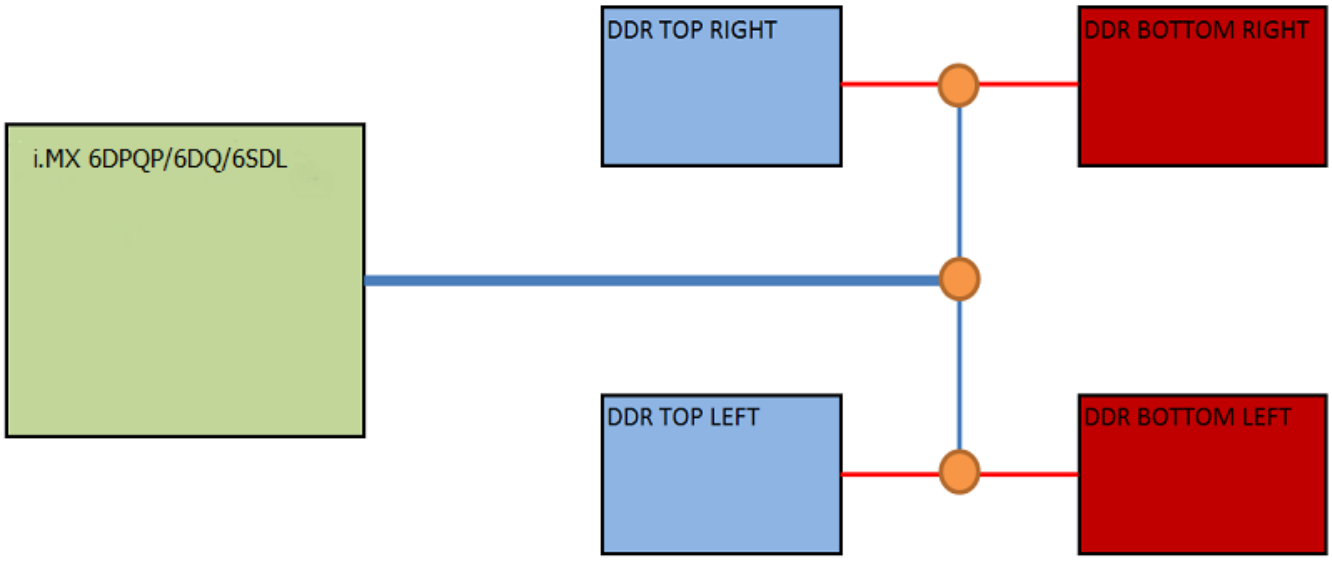


Figure 3-9. ADDR/CMD/CTRL signal topology

The routing for the data groups depends on the bus size. The following figure shows the point-to-point data bus connection, with routing by byte group.

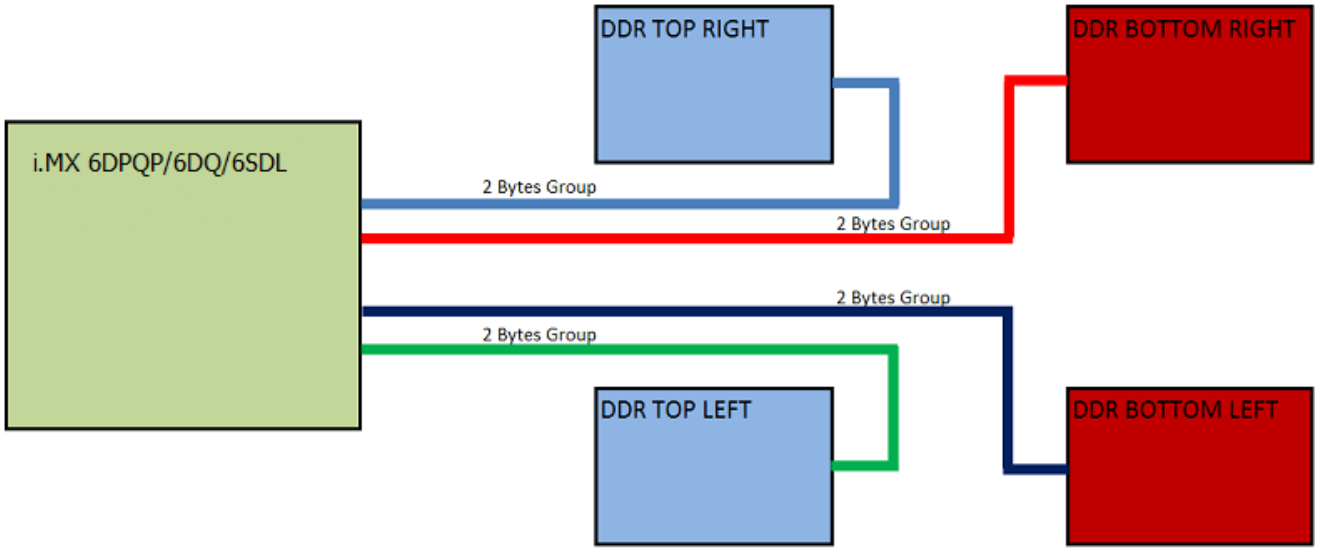


Figure 3-10. Point-to-point data bus connection (routing by byte group)

NOTE

i.MX 6Solo only uses the first two pairs of the 2 Bytes groups. All others are disabled.

3.6.5 4-Gigabyte recommendations

The following diagrams show the 4 Gbyte recommendations using both chip selects (CS[1:0]) and loading 2 GBytes to each one. This option has eight memories and requires the addition of a termination resistor.

Route the ADDR/CMD signals.

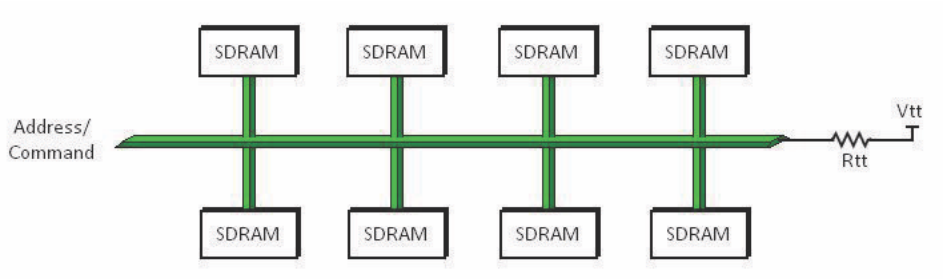


Figure 3-11. ADDR/CMD signal topology

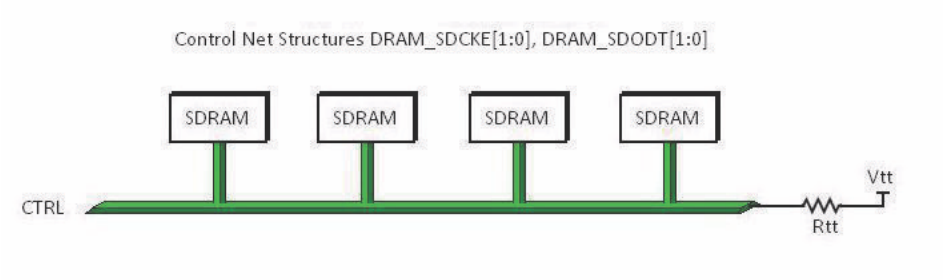


Figure 3-12. CTRL signal topology

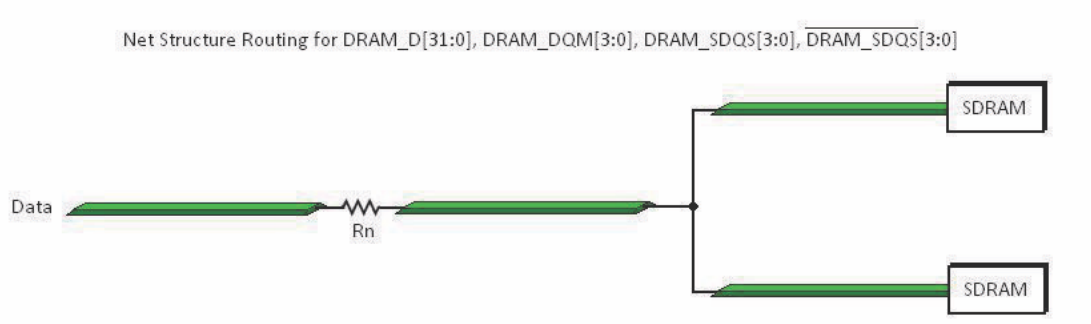


Figure 3-13. Data bus routing topology

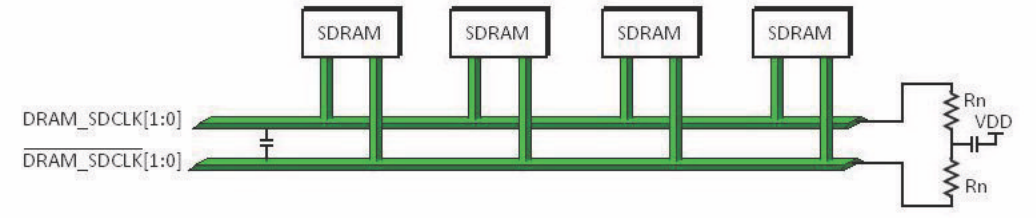


Figure 3-14. Clock routing topology

3.6.6 Four chips T topology routing examples

The figures in this section show examples for the routing of the 2 GByte DDR3 memories. Figure 3-15 through Figure 3-17 are a guideline of the T configuration routing with eight layers PCB. Table 3-4 shows the color coding used in the figures.

Table 3-4. Color code

Color	Meaning
Soft Green	ADD & CMD Signals
Yellow	Clocks
Soft Pink	Data Byte Group 0
Purple	Data Byte Group 1
Blue	Data Byte Group 2
Brown	Data Byte Group 3
Orange	Data Byte Group 4
Green	Data Byte Group 5
Olive Green	Data Byte Group 6
Soft Brown	Data Byte Group 7
Gray	DDR_1V5 & DDR_VREF
Soft Red	Control Signals

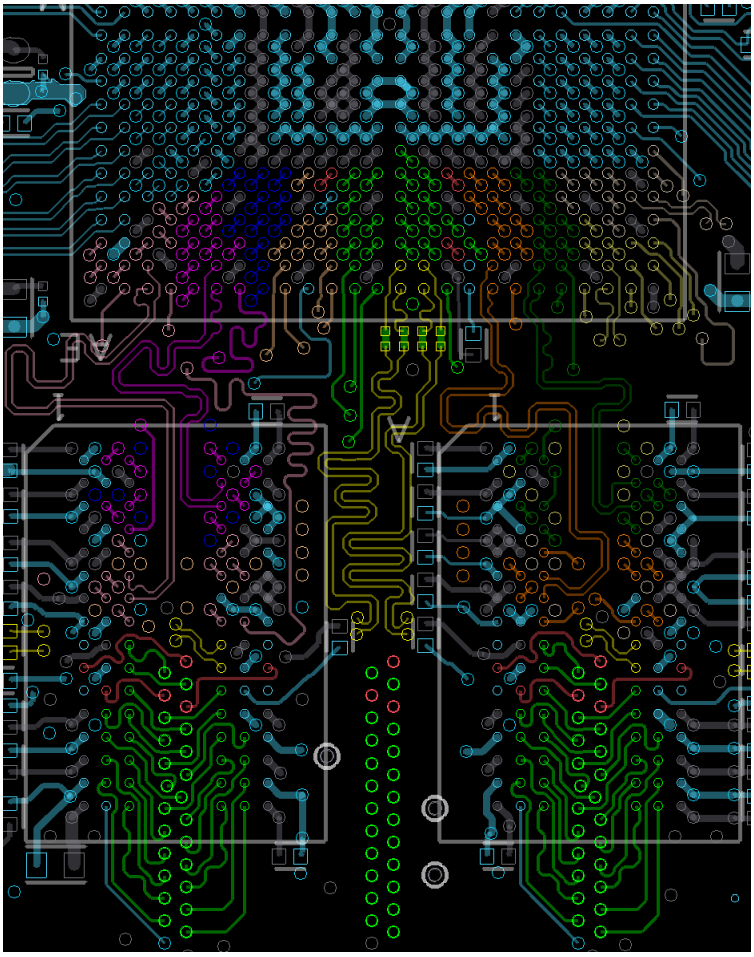


Figure 3-15. Top layer DDR3 routing

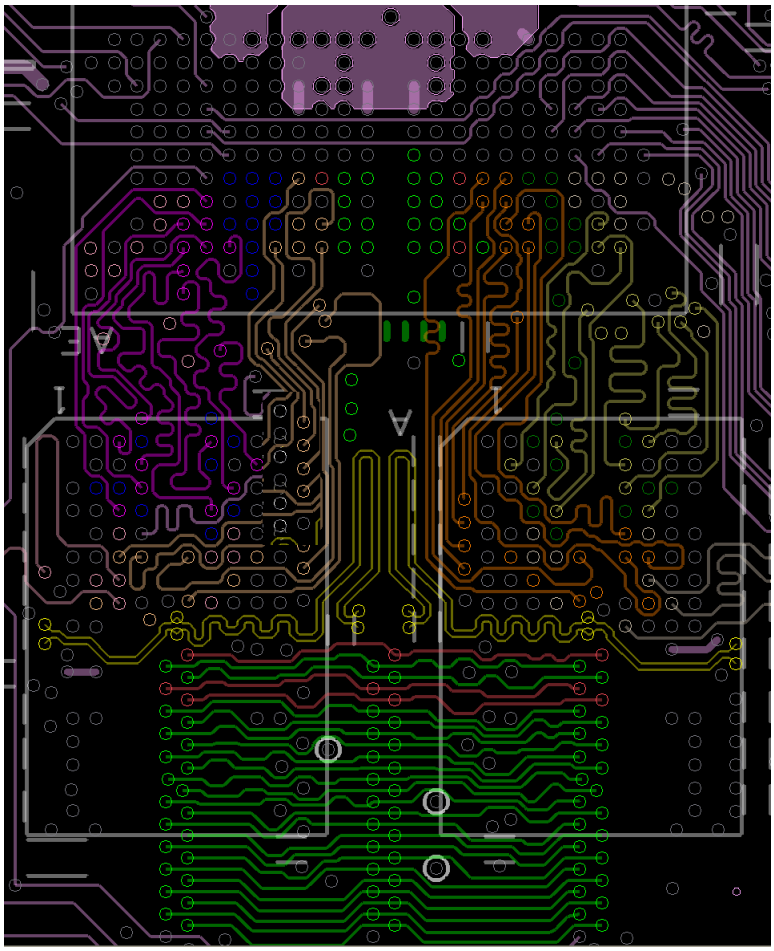


Figure 3-16. Internal L6 DDR3 routing

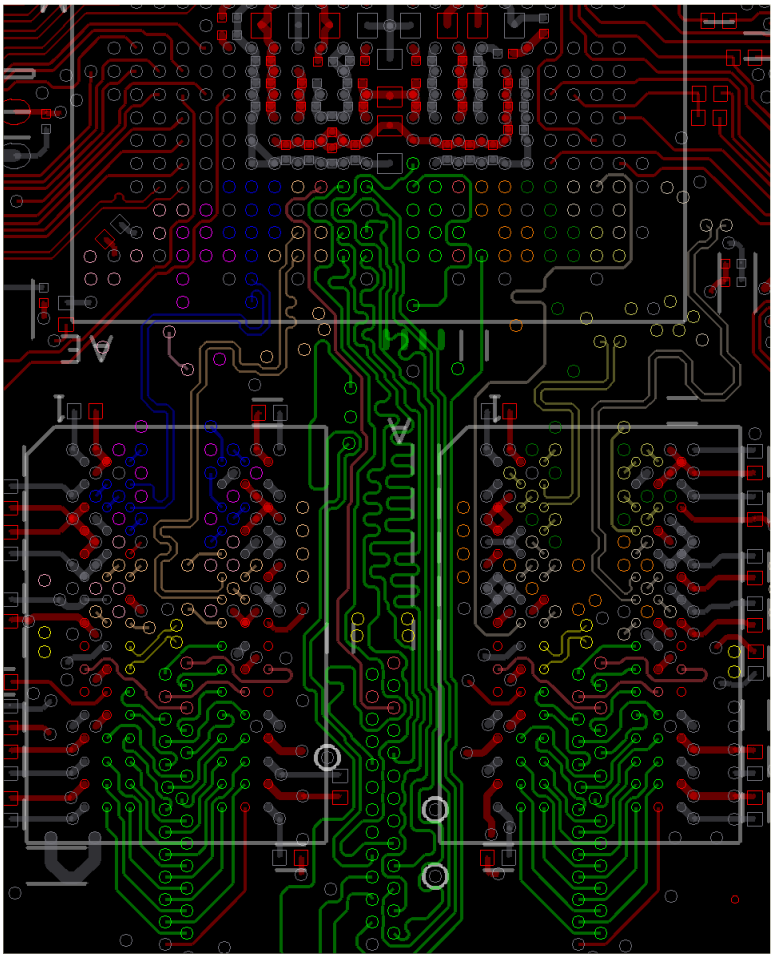


Figure 3-17. Bottom layer DDR3 routing

The following table shows the total etch of the signals for the byte 0 and byte 1 groups. The layout is an example, using 2000 mils for the clock.

Table 3-5. Total signal etch (DDR3)

Signals	Length (Mils)
DRAM_D0	1025.349
DRAM_D1	1028.996
DRAM_D2	1028.752
DRAM_D3	1021.158
DRAM_D4	1021.930
DRAM_D5	1025.398
DRAM_D6	1025.564
DRAM_D7	1029.326

Table 3-5. Total signal etch (DDR3) (continued)

Signals	Length (Mils)
DRAM_DQM0	1028.555
DRAM_SDQS0	1023.419
DRAM_SDQS0_B	1023.373
DRAM_D8	648.862
DRAM_D9	654.371
DRAM_D10	652.653
DRAM_D11	653.712
DRAM_D12	650.961
DRAM_D13	648.433
DRAM_D14	649.588
DRAM_D15	651.781
DRAM_DQM1	653.106
DRAM_SDQS1	669.240
DRAM_SDQS1_B	669.736
DRAM_SDCLK0	2120.044
DRAM_SDCLK0_B	2118.283
DRAM_SDCLK1	2112.518
DRAM_SDCLK1_B	2112.829

3.6.7 Eight chips fly-by topology routing examples

The figures in this section show examples for the routing of 4-Gbyte DDR memories. These figures are a guideline of the routing by layer using the fly by configuration topology. They use the same color code shown in Table 3-4.

NOTE

The SABRE SD board referenced in the beginning of this chapter does not use eight DDR chips. The following screen shots are from the validation board layout.

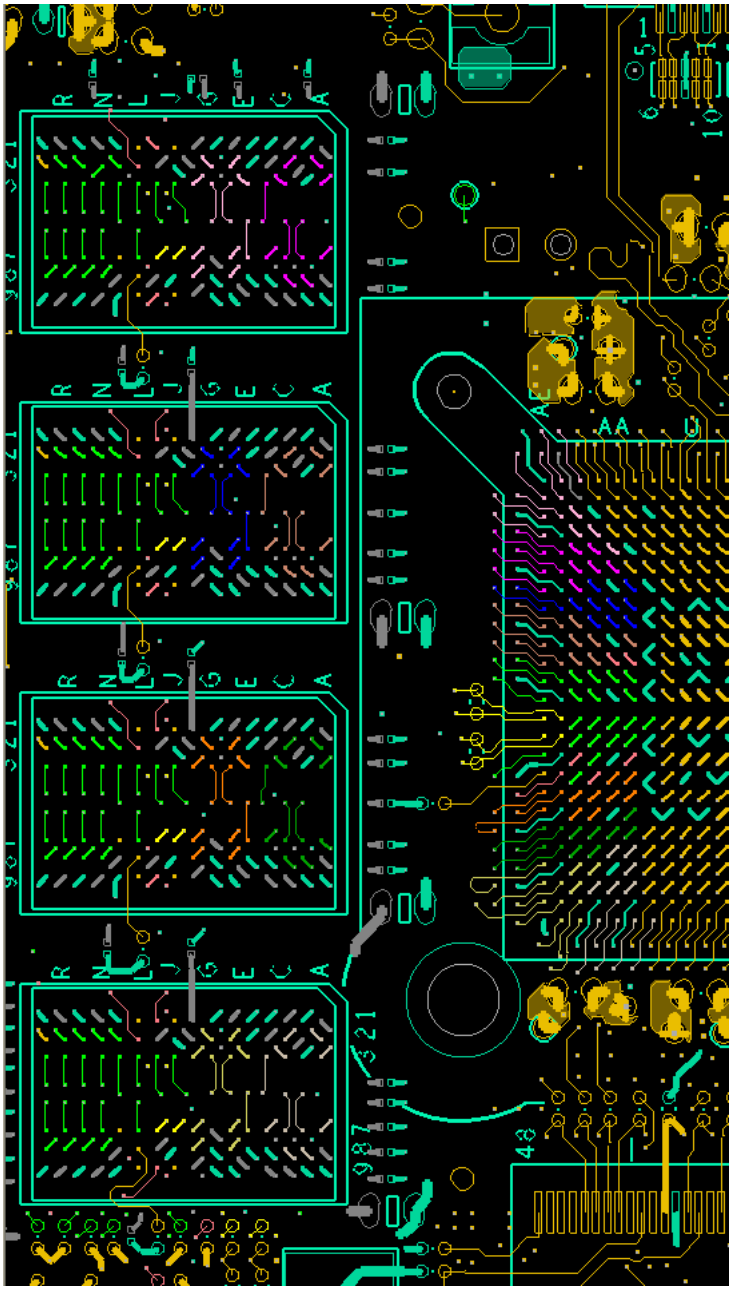


Figure 3-18. Top DDR3 routing

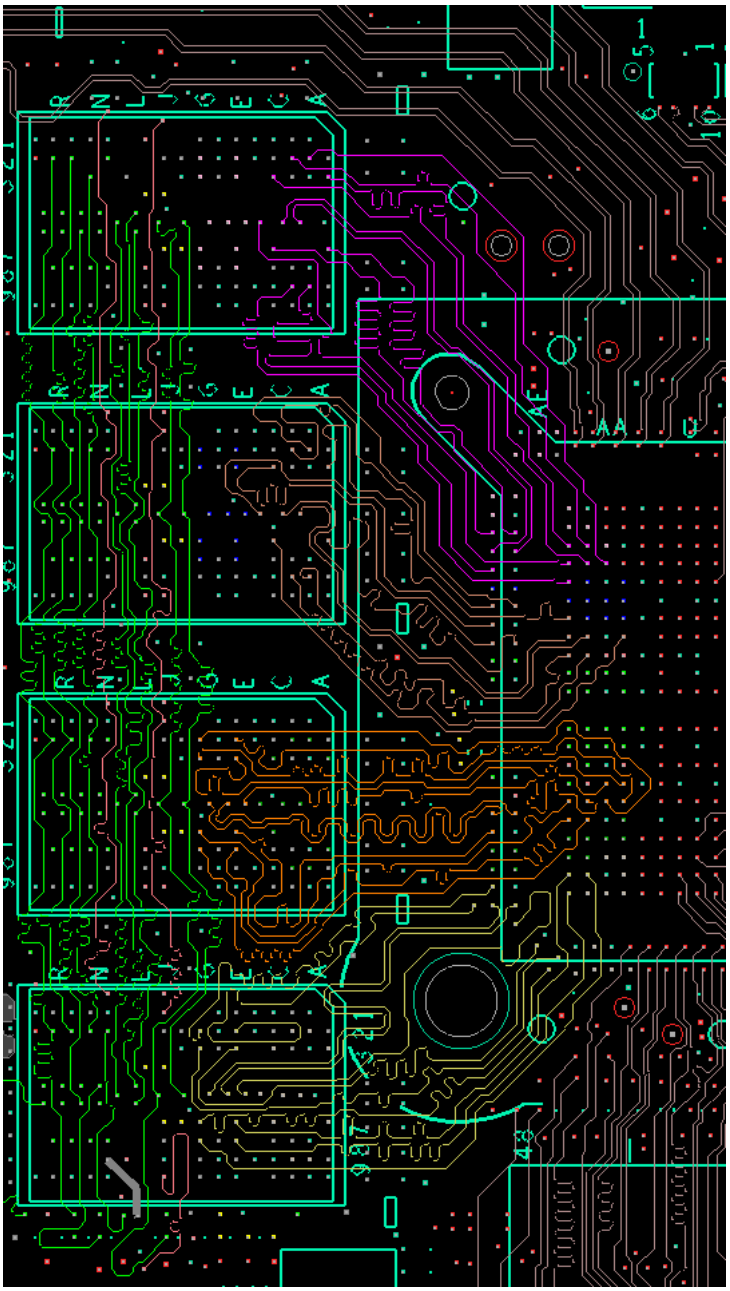


Figure 3-19. Internal L3 DDR3 routing

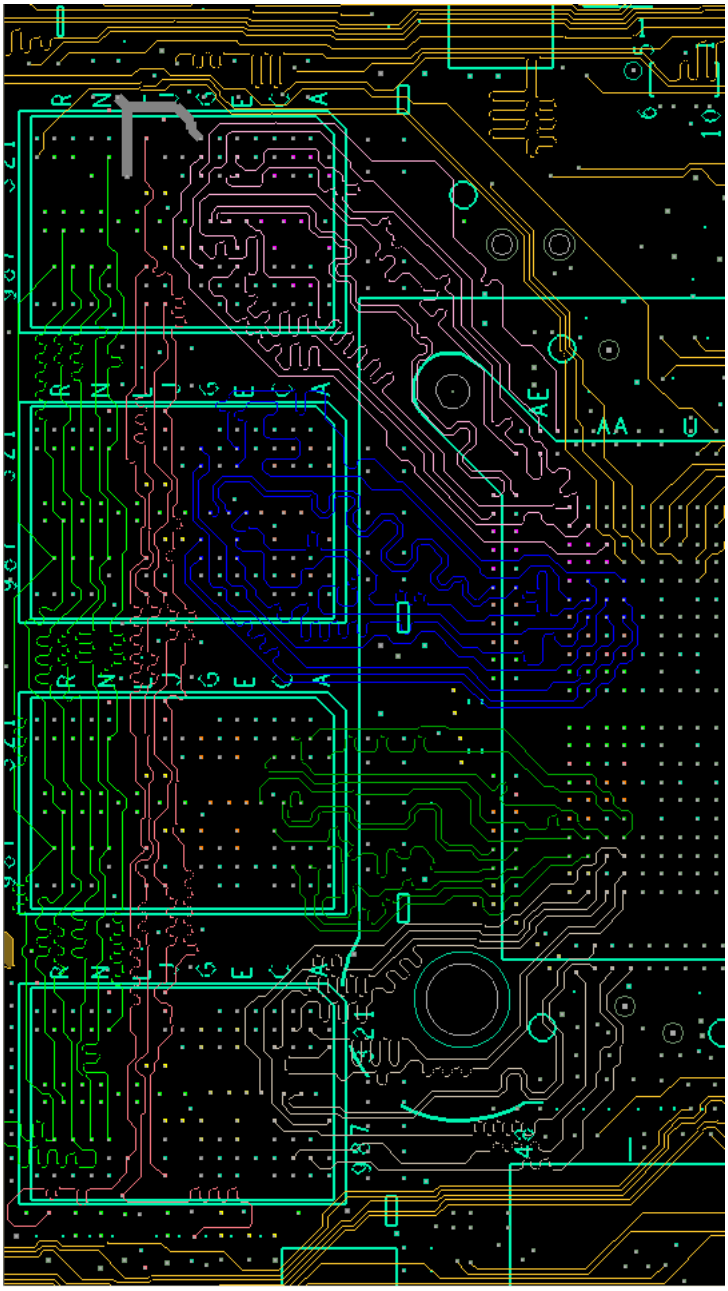


Figure 3-20. Internal L4 DDR3 routing

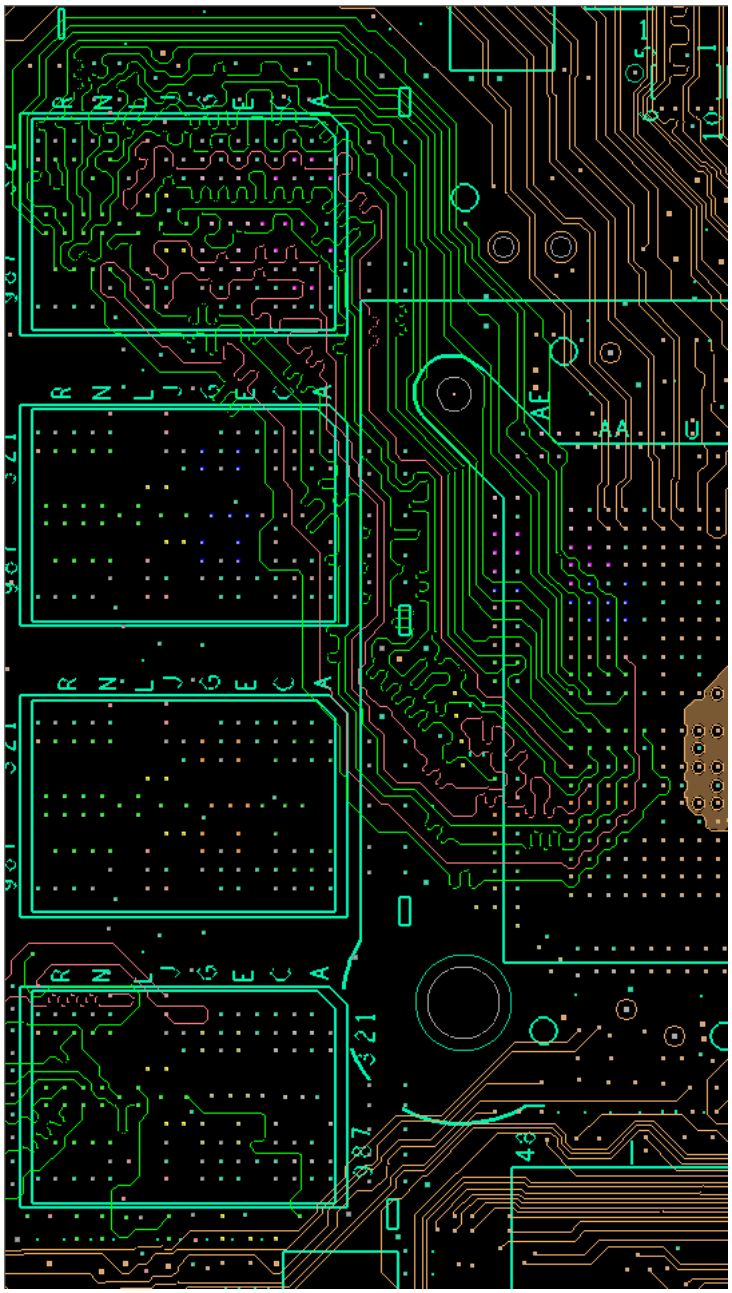


Figure 3-21. Internal L11 DDR3 routing

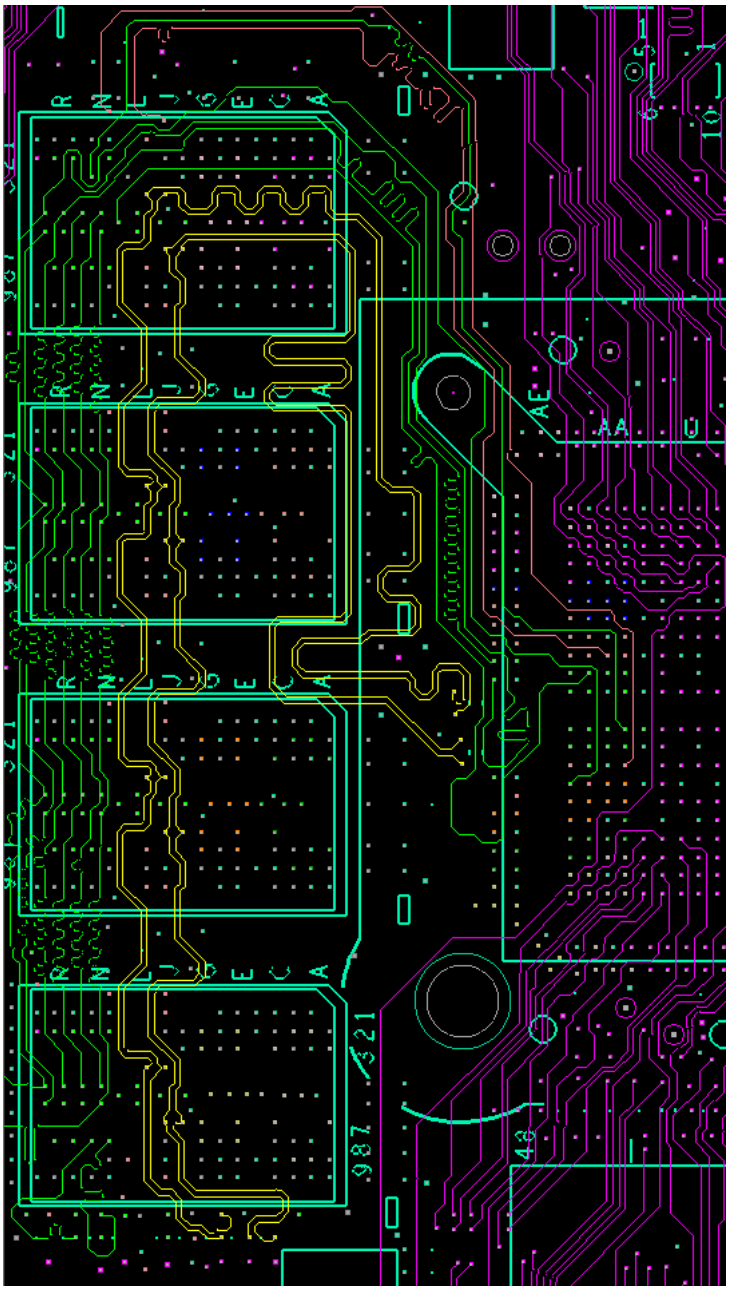


Figure 3-22. Internal L12 DDR3 routing

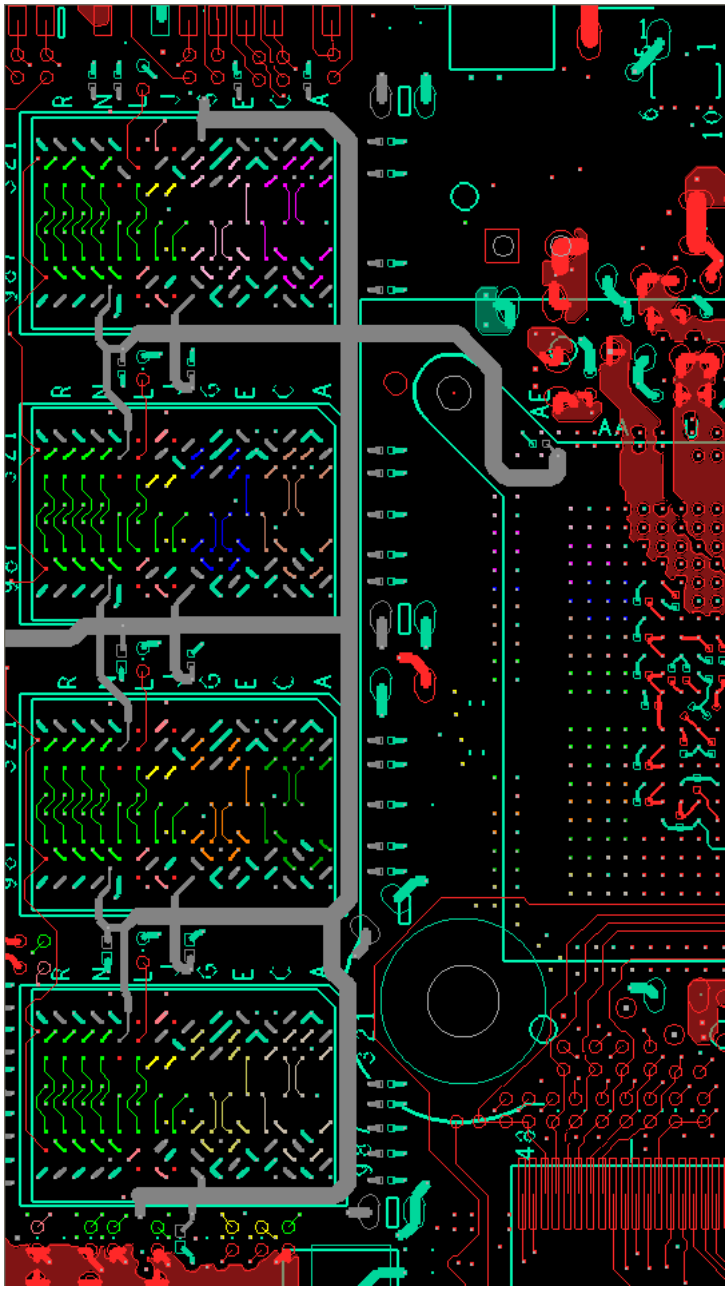


Figure 3-23. Bottom DDR3 routing

The following table shows the total etch of the signals for the byte 0 and byte 1 groups.

Table 3-6. Total signal etch (DDR3)

Signals	Length (Mils)
DRAM_D0	1244.97
DRAM_D1	1252.82
DRAM_D2	1237.48
DRAM_D3	1242.95
DRAM_D4	1240.12
DRAM_D5	1254.37
DRAM_D6	1254.58
DRAM_D7	1238.18
DRAM_DQM0	1297.45
DRAM_SDQS0	1295.34
DRAM_SDQS0_B	1295.68
DRAM_D8	1103.69
DRAM_D9	1116.14
DRAM_D10	1105.01
DRAM_D11	1105.17
DRAM_D12	1120.4
DRAM_D13	1123.06
DRAM_D14	1105.72
DRAM_D15	1111.24
DRAM_DQM1	1152.16
DRAM_SDQS1	1158.48
DRAM_SDQS1_B	1162.29
DRAM_SDCLK0	4723.96
DRAM_SDCLK0_B	4681.95
DRAM_SDCLK1	4750.69
DRAM_SDCLK1_B	4699.00

3.6.8 High speed signal 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).
- 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× spacing from an adjacent trace (2.5× height from reference plane) to reduce cross-talk.
- 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.

3.6.9 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. The copper plane is represented by the color gray in Figure 3-24 and by the horizontal green lines in Figure 3-25.

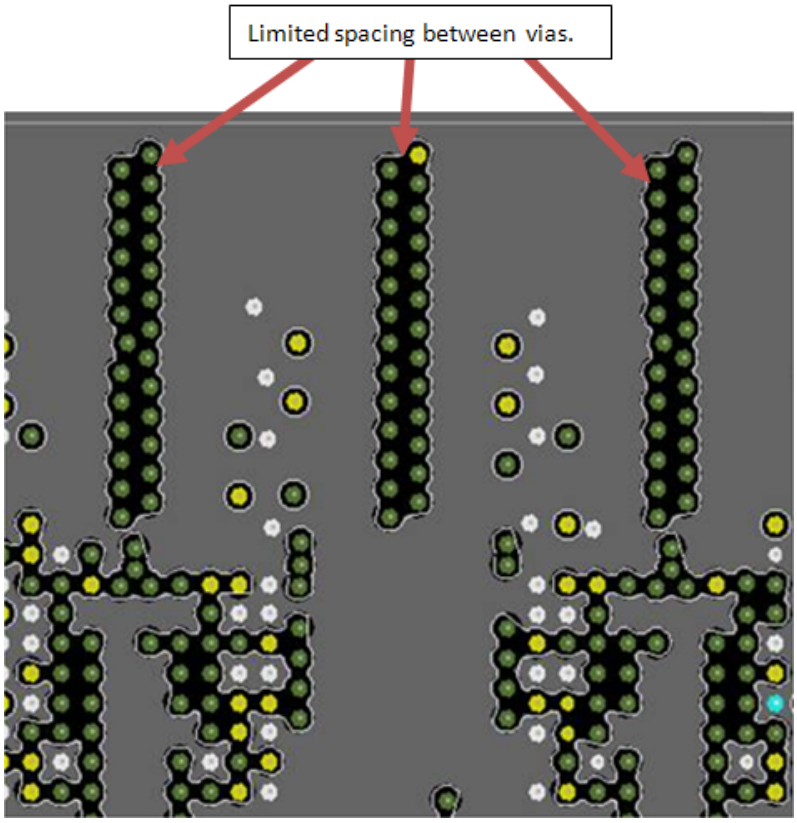


Figure 3-24. Poor GND plane 1

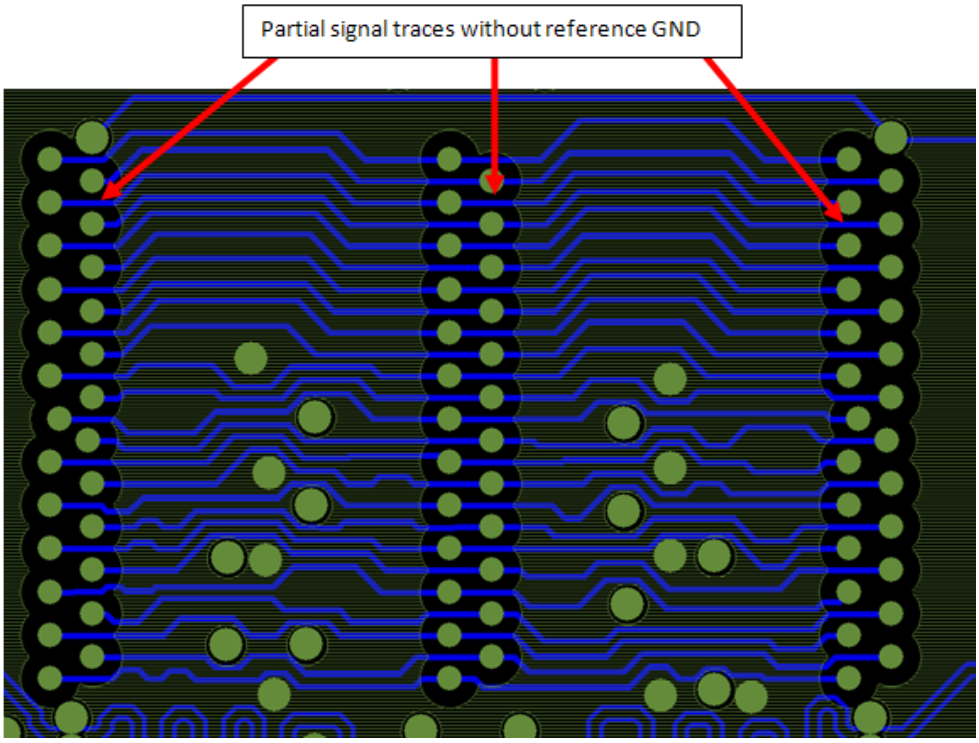


Figure 3-25. Poor GND plane 2

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

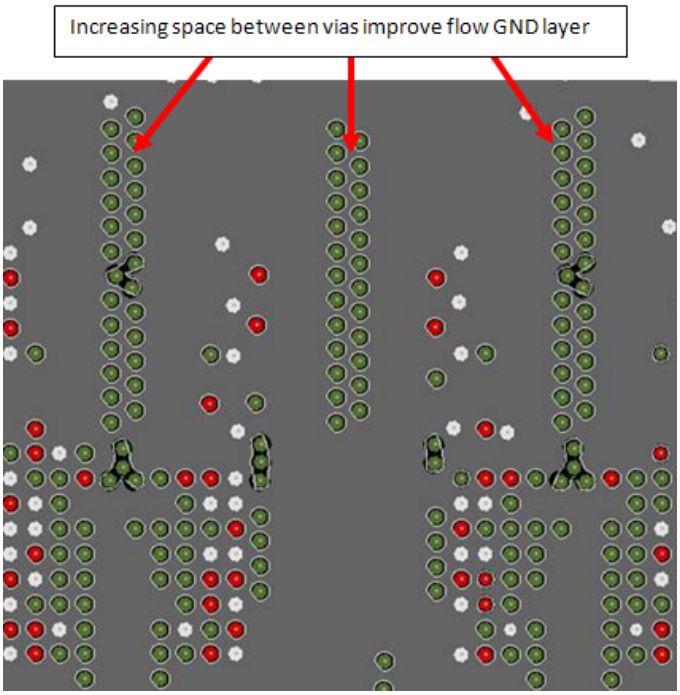


Figure 3-26. Good layout GND plane detail

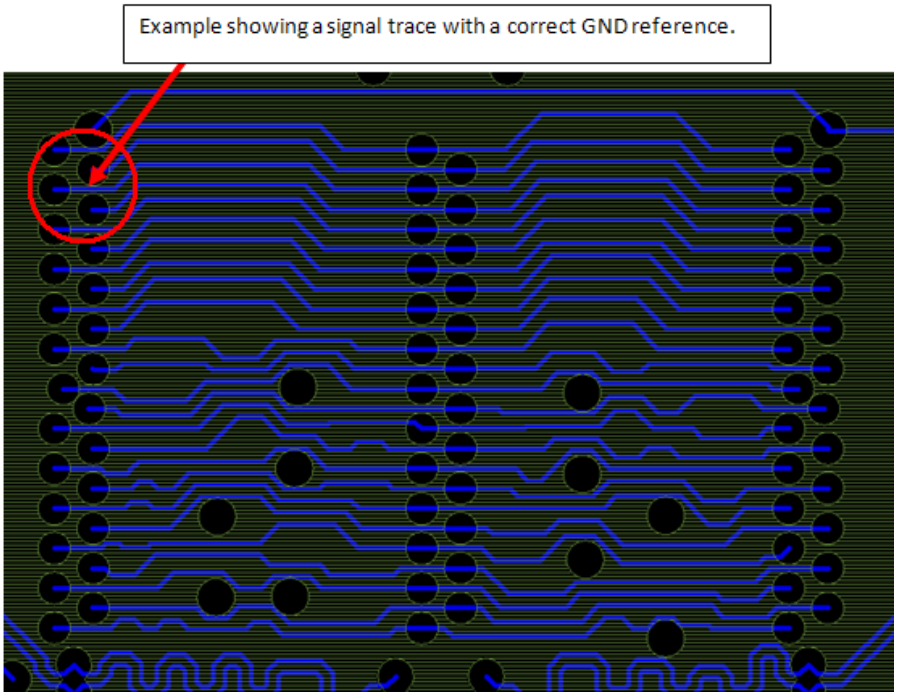


Figure 3-27. Good layout GND plane detail

3.7 DDR power recommendations

The following recommendations apply to the VREF (P0V75_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 µF capacitors by the regulator, controller, and devices.
- Place one 1.0 µF 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 µF 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.

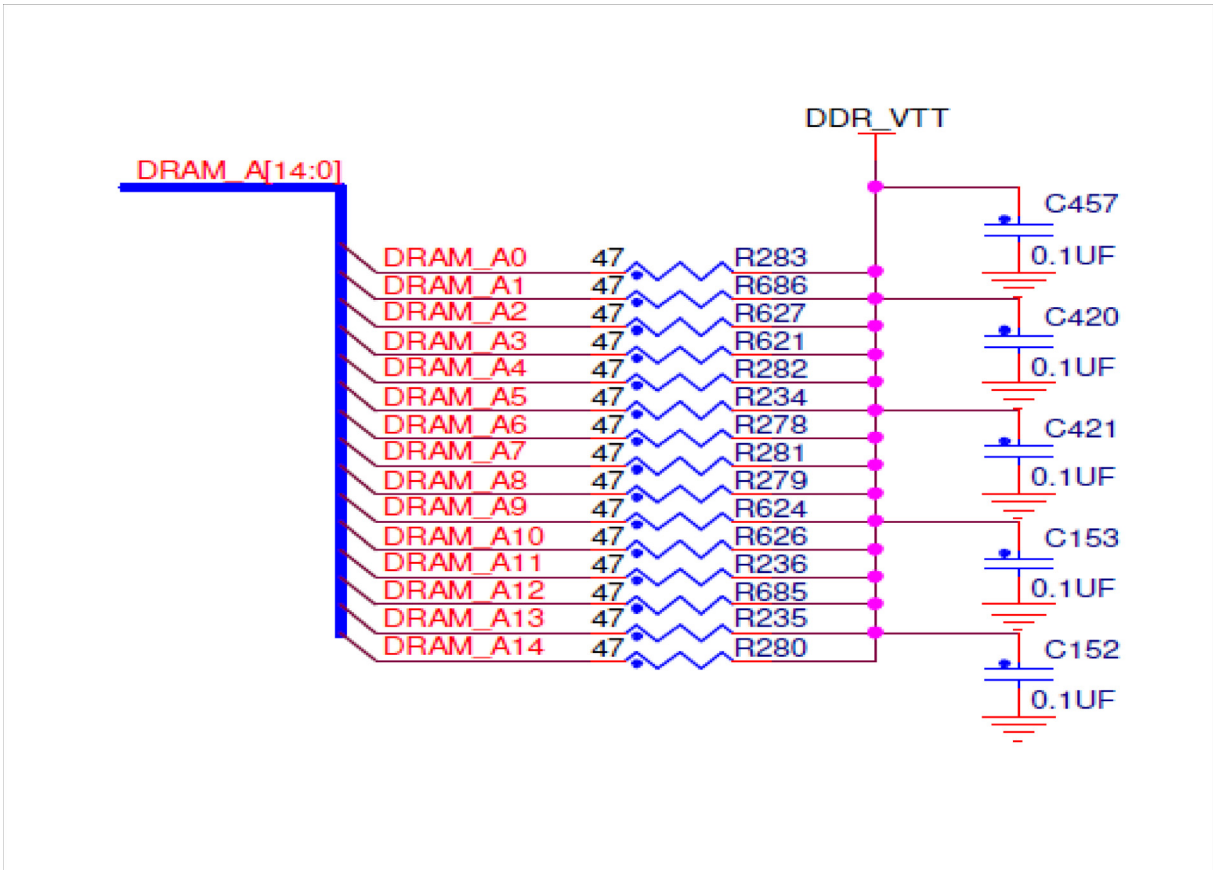


Figure 3-28. DDR_VTT evaluation board example

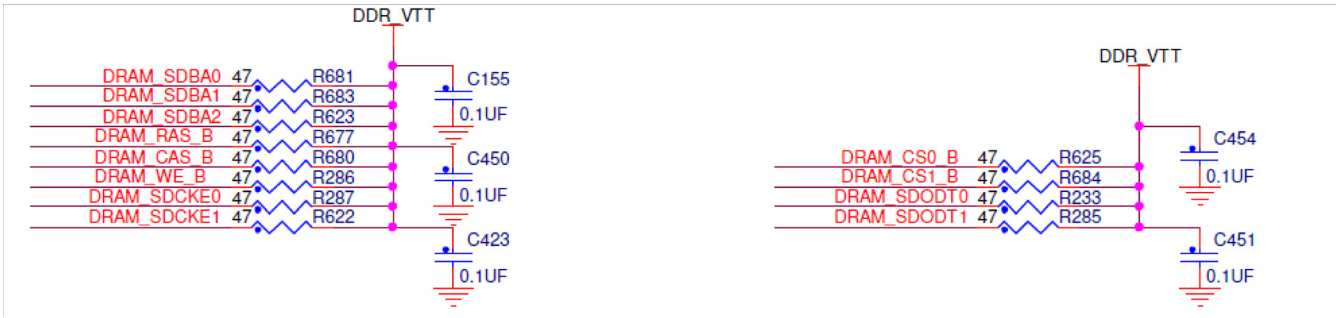


Figure 3-29. DDR_VTT evaluation board examples

3.8 PCI Express interface recommendations

This chip provides a ×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 ×1 is composed of two differential signals pairs: one TXD signal pair and one RXD signal pair.

Table 3-7. PCI Express signal descriptions

Signal name	Signal group	Description
PCIE_TXP, PCIE_TXM	Data	PCI Express transmit differential pair
PCIE_RXP, PCIE_RXM	Data	PCI Express receive differential pair

3.8.1 PCI Express general routing guidelines

Use the following recommendations for PCI Express general routing:

- The trace width and spacing of the lanes ×1 signals should be such that the differential impedance is 85 Ω ± 10%.
- The PCIE_REXT contacts should be connected to a 200 Ω 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). Avoid split planes, plane slots, or anti-etch.
- Maintain the parallelism (skew matched) between differential signals; these traces should be the same overall length.
- Keep signals with traces as short as possible.
- Route signals with a minimum amount of corners. Use 45-degree turns instead of 90-degree turns.
- Do not create stubs or branches.
- Maintain symmetry of differential pair routing.

3.8.2 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 Ω resistors
- AC-coupled Tx signals with 0.1µF capacitors

3.8.3 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/files/training_presentation/TP_HARDWARE_DESIGN_PCI_SMGIII.pdf)
- 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)

3.9 HDMI recommendations

Use the following recommendations for the HDMI.

- HDMI differential pairs should have a impedance of 100 Ω in all paths to the connector.
- It is highly recommended to use an HDMI transmitter port protection for ESD, level shifting, isolation, overcurrent and backdrive protecion.

3.10 SATA recommendations

Use the following recommendations for the SATA.

- SATA differential pairs should have a differential impedance of 100 Ω.
- Each differential pair should be length matched to ± 5 mils.
- Follow standard high-speed differential routing rules for signal integrity.

3.11 LVDS recommendations

Use the following recommendations for the LVDS.

- Follow standard high-speed differential routing rules for signal integrity.
- Each differential pair should be length matched to ± 5 mils.
- LVDS differential pairs should have a differential impedance of 100 Ω.
- Note that there are no pad control registers for the LVDS signals. So drive strength is not controllable through software.

3.12 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 be such that the differential impedance is 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; these traces should be the same overall length.
- 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.

3.13 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 3-8. Impedance signal recommendations

Signal Group	Impedance	Layout Tolerance (±)
All signals, unless specified	50 Ω SE	10%
PCIe TX/RX Diff signals	85 Ω Diff	10%
USB Diff signals	90 Ω Diff	10%
Diff signals: LVDS, SATA, HDMI, DDR, MIPI (CSI & DSI), MLB, Phy IC to Ethernet Connector	100 Ω Diff	10%

The following figure shows the dimensions of a stripline and microstrip pair. Figure 3-31 shows the differential pair routing.

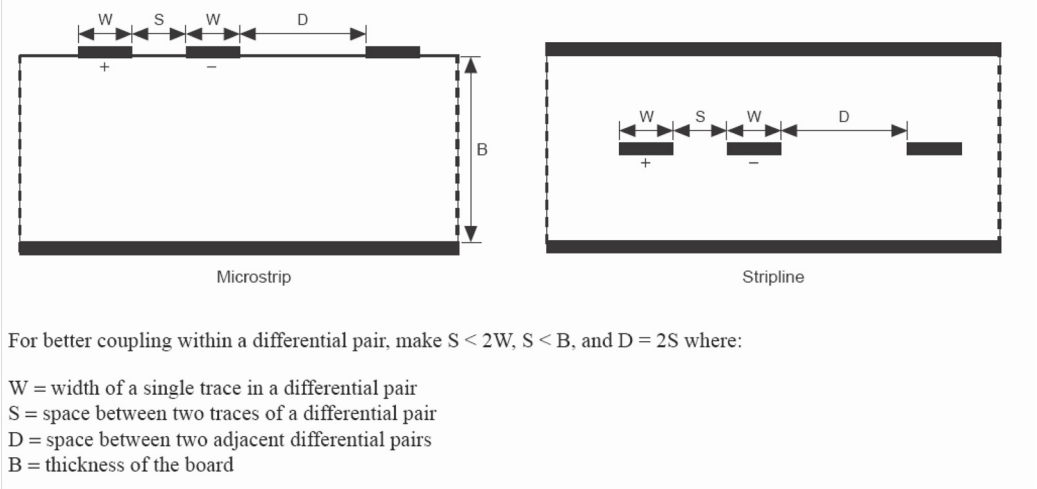


Figure 3-30. Microstrip and stripline differential pair dimensions

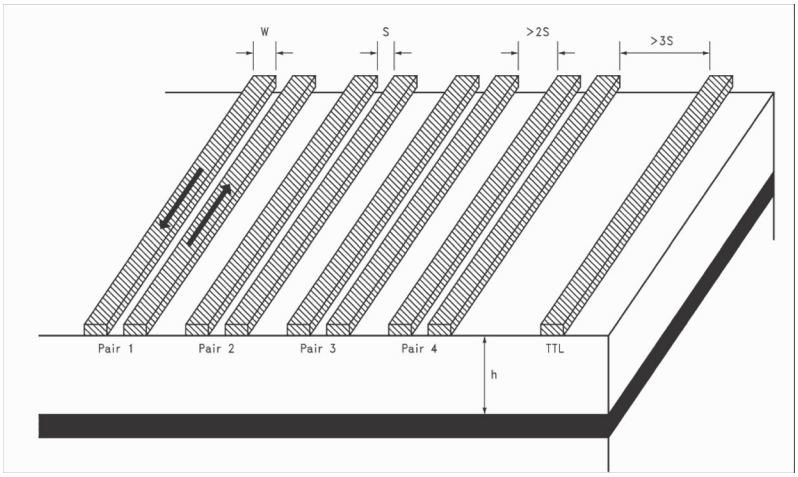


Figure 3-31. Differential pair routing

- 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).

3.14 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.

3.15 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 forth) 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 500 Ω 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 www.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.

3.16 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

3.17 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 3-32 and Figure 3-33 show two examples of static routing where a match is achieved without needing to tune one element of the differential pair.

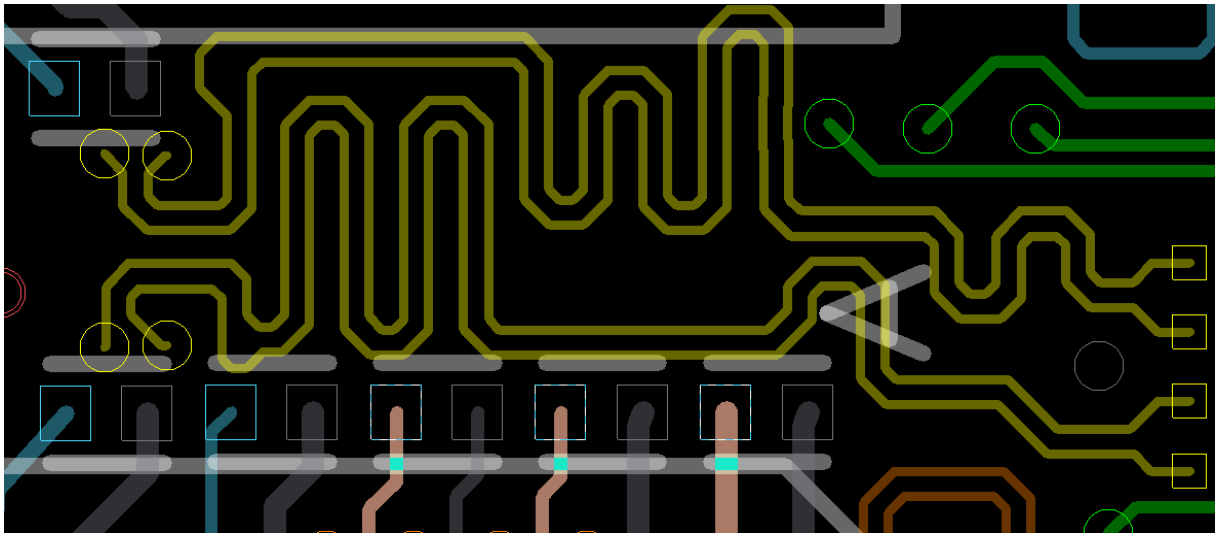


Figure 3-32. Yellow traces diff pairs 1

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.

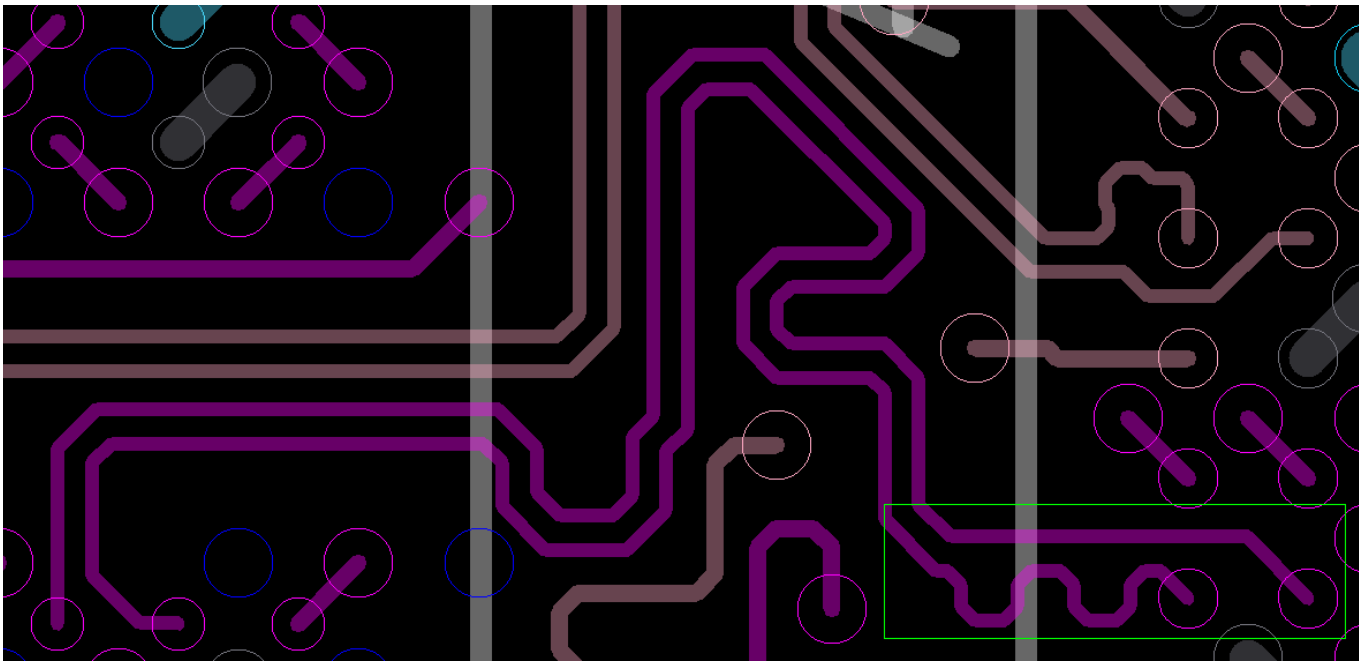


Figure 3-33. Small bumps added to the shorter differential pair

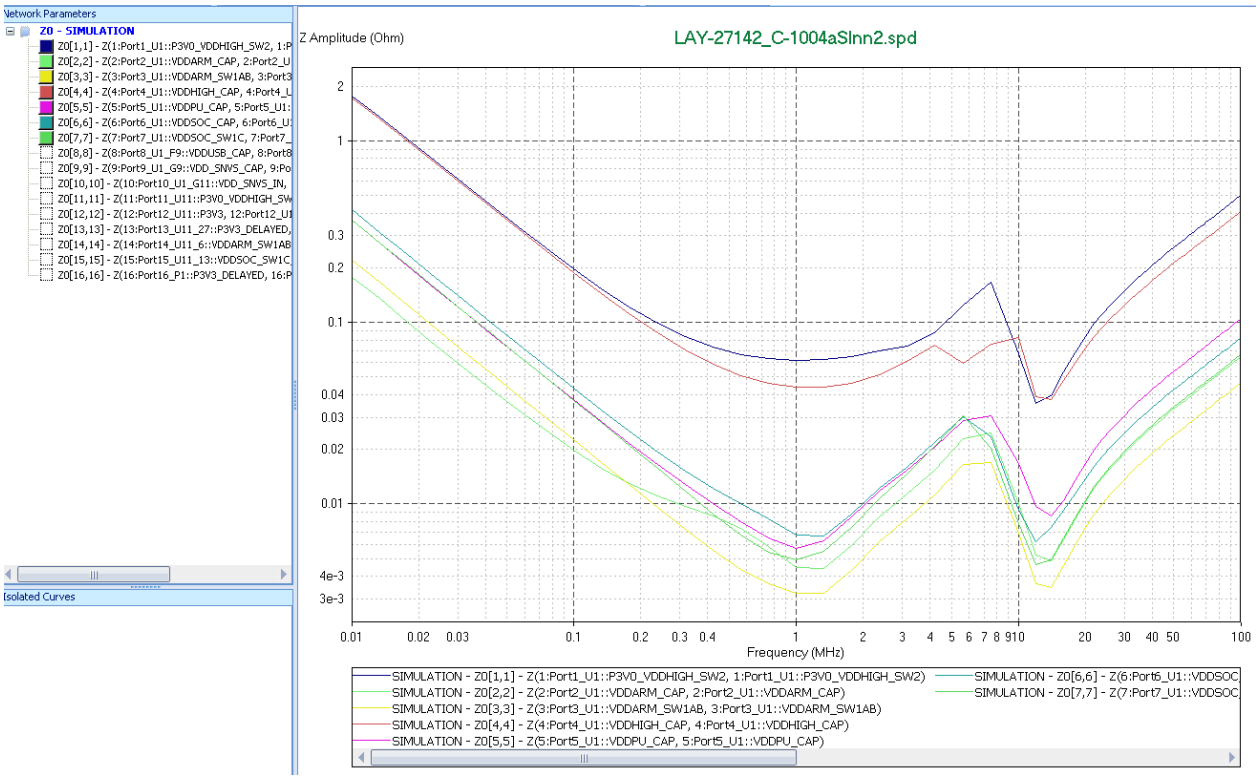
Having this delay reduces skew and phase problems.

3.18 Guideline for power net electrical performance

The following figure shows the simulated open-circuit impedance of the SABRE AI platform power nets. The board number is 700-27142, layout version is LAY-27142_C-1004.

The graph is provided as impedance guidance for the various power nets relative to ground.
NXP recommends that a user's board be at or below the impedances curves shown. For example, consider the 100 MHz point on the VDD_ARM_CAP curve at 0.065 ohms. User layouts at 100 MHz should have impedance from VDD_ARM_CAP to ground of 0.065 ohms or less.

Figure 3-34. Power Net Open-Circuit Impedance – 10 kHz to 100 MHz



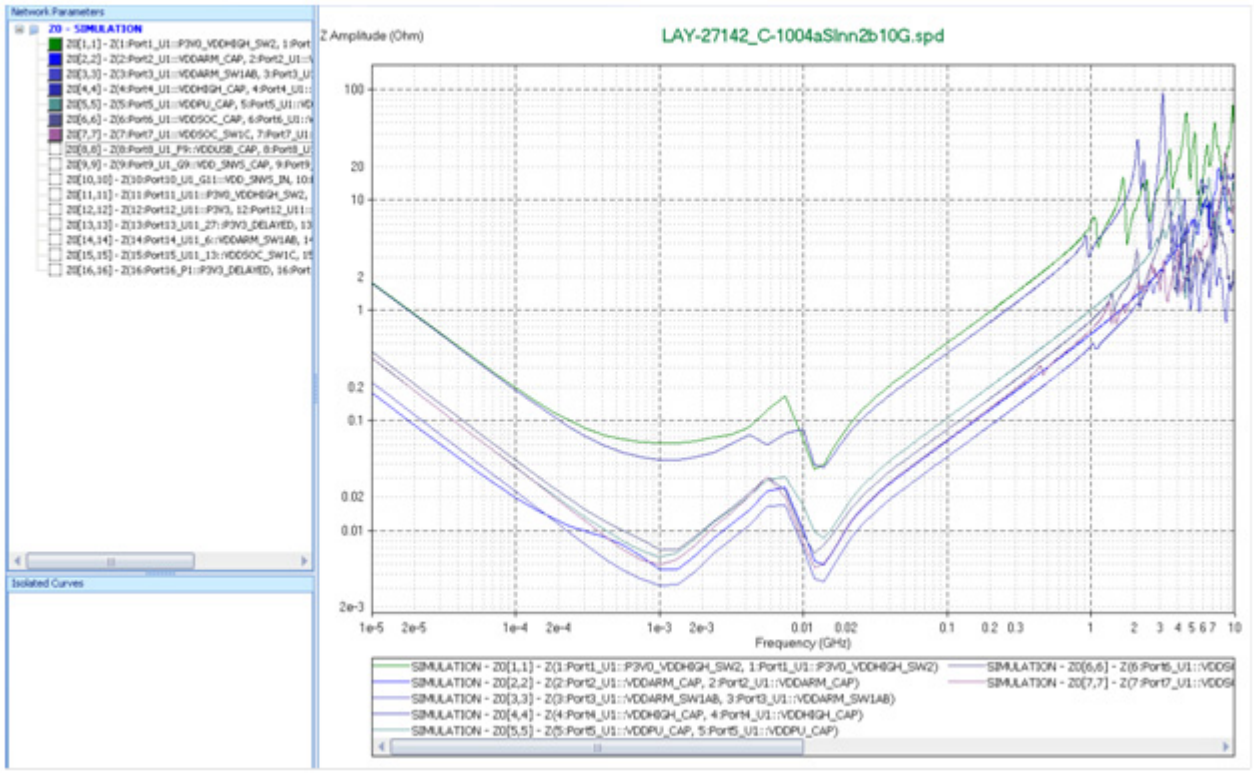


Figure 3-35. Power net open-circuit impedance-10 kHz to 10 GHz

Chapter 4
Requirements for Power Management

4.1 Power management requirements overview

This chapter provides the power requirements for the following i.MX 6 series families of processors:

- i.MX 6QuadPlus
- i.MX 6Quad
- i.MX 6DualPlus
- i.MX 6Dual
- i.MX 6DualLite
- i.MX 6Solo

4.1.1 Voltage domains overview

These chips have several voltage domains that may need to be supplied with different voltages depending on system needs. The chip internal regulators and its complementary PMIC PF0100 provide a complete and simple way to supply each voltage domain with different voltages when needed. Section 4.4, "Connection diagrams," shows the internal regulators and the connections to PF0100.

4.1.2 PF0100 overview

PF0100 consists of the following components used to supply the i.MX6 voltage domains as well as other blocks on the system:

- 4 buck regulators
- 1 boost regulator
- 8 LDOs

The default PF0100 power-up sequence is programmed to fit the requirements of the i.MX 6 series families of processors. However, the PF0100 can be adjusted to meet the specific requirements for system applications by using the one time programmable (OTP) feature.

To meet the increased graphical power requirements on i.MX 6QuadPlus/6DualPlus the F9 PF0100 configuration was introduced, refer to the MCIMX6QP-SDB reference design for further details.

4.2 Requirements for a generic interface between chip and PF0100

Table 4-1 shows an example interface between the i.MX6DQ and PF0100, using a suitable power-up sequence.

For more info about PF0100 functionality and the i.MX 6 series families of processors' power requirements, see the product data sheets.

Table 4-1. Interface between 6Quad/6Dual and PF0100

Voltage rail	Supply reg	Voltage (V)	Supply reg current capability (A)	Generated by	Power up sequence	Notes
VDDARM_IN VDDARM23_IN	SW1A/B	1.35	2.5	PF0100	1	Short these together with a shunt for quad core operation. Cut shunt for dual core operation and connect VDDARM23_IN to GND
VDDARM_CAP VDDARM23_CAP	Note 2	(1.2)	—	i.MX	—	Short these together with a shunt for quad core operation. Cut shunt for dual core operation and connect VDDARM23_IN to GND
VDDSOC_IN	SW1C	1.325	2.0	PF0100	1	VDDARM_IN and VDDSOC_IN supplies can be shorted together and run off of one fewer switcher for cost-sensitive platforms.
VDDSOC_CAP VDD_CACHE_CAP	Note 2	(1.1)	—	i.MX		Short these together. VDDSOC_CAP is the output of an i.MX6 internal LDO that can supply more voltage domains as indicated below in this table.
VDDPU_CAP	Note 2	(1.2)	—	i.MX	—	—
VDDHIGH_IN	SW2	3.0	2.0	PF0100	2	—
VDDHIGH_CAP	Note 2	(2.5)	—	i.MX		VDDHIGH_CAP is the output of an i.MX6 internal LDO that can supply more voltage domains as indicated below in this table.
VDD_SNVS_IN	VSNVS	3.0	400 μA	PF0100	0	According to table 13 of the chip data sheet, VDD_SNVS_IN can draw up to 1 mA depending on the application. For those cases, an external regulator is needed because the PF0100 VSNVS regulator supplies 400 μA.
NVCC_RGMII	SW3A/B	1.5	1.25	PF0100	3	—
NVCC_DRAM						SW3 can be configured from 0.4 to 3.3 V so that the right voltage can be chosen for the respective DDR technology.

Table 4-1. Interface between 6Quad/6Dual and PF0100 (continued)

Voltage rail	Supply reg	Voltage (V)	Supply reg current capability (A)	Generated by	Power up sequence	Notes
DRAM_VREF	VREFDDR or SW4	0.5×SW3	0.01 or 1.0	PF0100	3	—
NVCC_CSI	SW4, VGEN4 or external regulator	1.8–3.3	1.0 or 0.35	PF0100 or external regulator	5	Depending on system needs, these voltage domains can be supplied together or independently with equal or different voltages and regulators. Be sure to account for the current needs of the domains and the current capability of the regulator when making this decision.
NVCC_EIM 0, 1, 2[1]						
NVCC_ENET						
NVCC_GPIO						
NVCC_LCD						
NVCC_NANDF						
NVCC_SD1, 2						
NVCC_SD3						
NVCC_JTAG						
NVCC_PLL_OUT	Note 2	1.1	—	i.MX	—	—
NVCC_MIPI	VDDHIGH_CAP	2.5	—	i.MX	—	—
NVCC_LVDS2P5	VDDHIGH_CAP	2.5	—	i.MX	—	This supply also powers the pre-drivers of the DDR IO pins (DRAM and RGMII). Therefore, it must always be provided, even when LVDS is not used.
USB_OTG_VBUS	SWBST	5.0	0.6	PF0100	—	• In Host configuration, USB_OTG_VBUS can be fed from the SWBST output of the PF0100. • In device configuration, USB_OTG_VBUS is the external host that provides this voltage.
USB_H1_VBUS	SWBST	5.0	0.6	PF0100	—	Connect to VBUS pin of USB connector
VDDUSB_CAP	Note 2	(3.0)	—	i.MX	—	—
SATA_VP	VDDSOC_CAP	1.1	—	i.MX	—	—
SATA_VPH	VDDHIGH_CAP	2.5	—	i.MX	—	—
HDMI_VP	VDDSOC_CAP	1.1	—	i.MX	—	—
HDMI_VPH	VDDHIGH_CAP	2.5	—	i.MX	—	—
PCIE_VP	VDDSOC_CAP	1.1	—	i.MX	—	—
PCIE_VPTX			—		—	—

Table 4-1. Interface between 6Quad/6Dual and PF0100 (continued)

Voltage rail	Supply reg	Voltage (V)	Supply reg current capability (A)	Generated by	Power up sequence	Notes
PCIE_VPH	VDDHIGH_CAP	2.5	—	i.MX	—	—

1 For Solo and DualLite chips, these 3 voltage rails should be connected together to the same voltage value. Quad and Dual chips can support three different EIM power rails.

2 These voltage domains are supplied by i.MX6 internal regulators.

The following table shows the PF0100 regulators that are available to supply the rest of the system circuitry.

Table 4-2. PF0100 regulators for other system circuitry

Supply	Output voltage(V)	Step size (mV)	Maximum Load current (mA)
SW4[1]	0.5 × SW3A_OUT, 0.4 – 3.3	25/50	1000
VGEN1	0.75 – 1.5	50	100
VGEN2	0.75 – 1.5	50	250
VGEN3	1.8 – 3.3	100	100
VGEN4[1]	1.8 – 3.3	100	350
VGEN5	1.8 – 3.3	100	100
VGEN6	1.8 – 3.3	100	200

1 In Table 4-1, it was recommended to supply the NVCC_x voltage domains with SW4 or VGEN4. Depending on the decision, one of them may not be available to supply the rest of the system circuitry.

4.3 i.MX6 internal regulators

These chips have been equipped with 7 internal regulators that simplify the power supply scheme of the system. The following table shows the regulators' power requirements. See Section 4.4, "Connection diagrams," for the distribution and connections of these LDOs.

Table 4-3. Internal regulator power requirements

LDO	Output voltage (V)	Output current (mA)
LDO_ARM	1.1	—
LDO_SOC	1.2	—
LDO_PU	1.1	—
LDO_2P5	2.5	350

Table 4-3. Internal regulator power requirements (continued)

LDO	Output voltage (V)	Output current (mA)
LDO_1P1	1.1	—
LDO_SNVS	1.1	—
LDO_USB	3.0	50

4.4 Connection diagrams

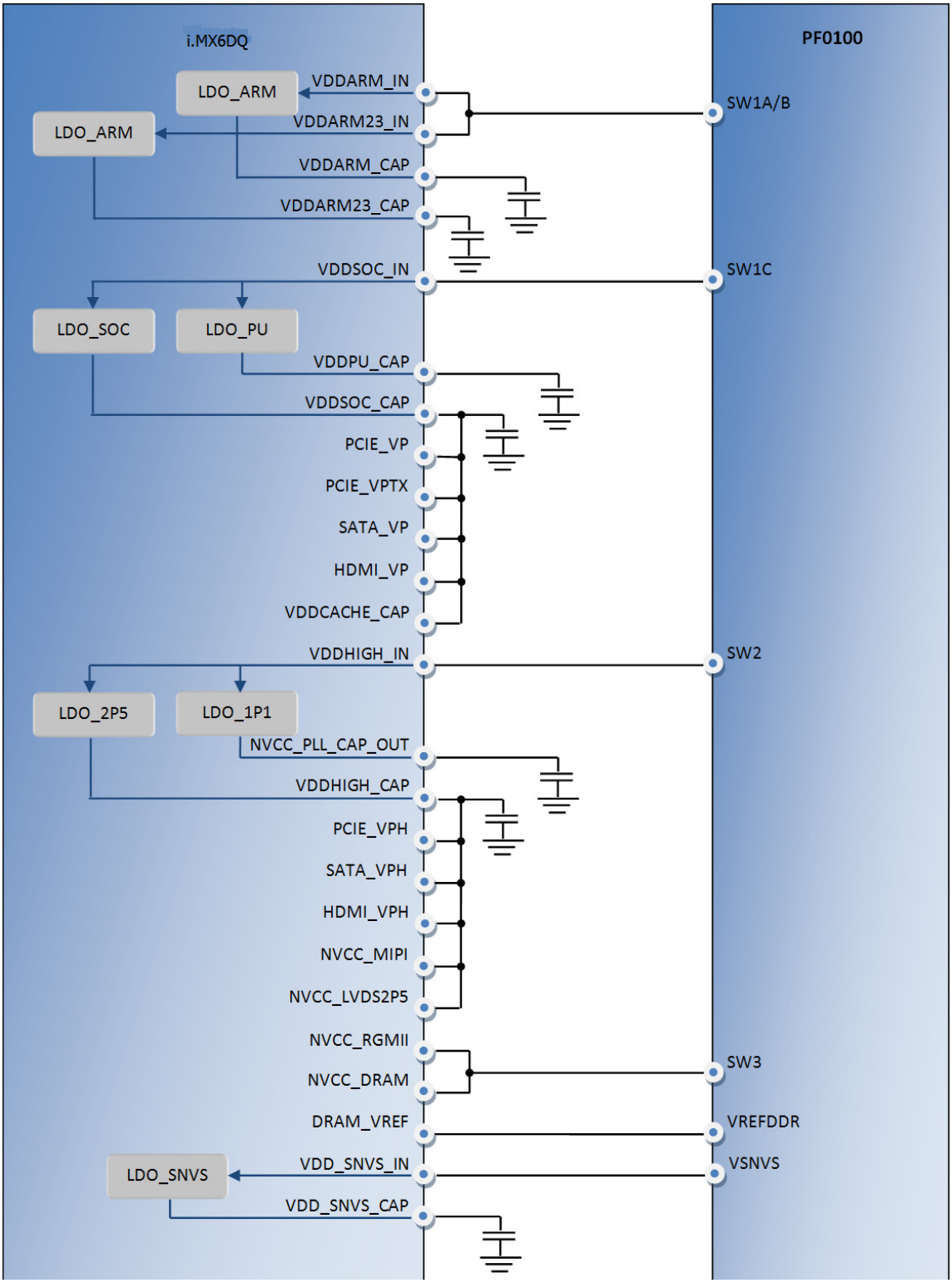
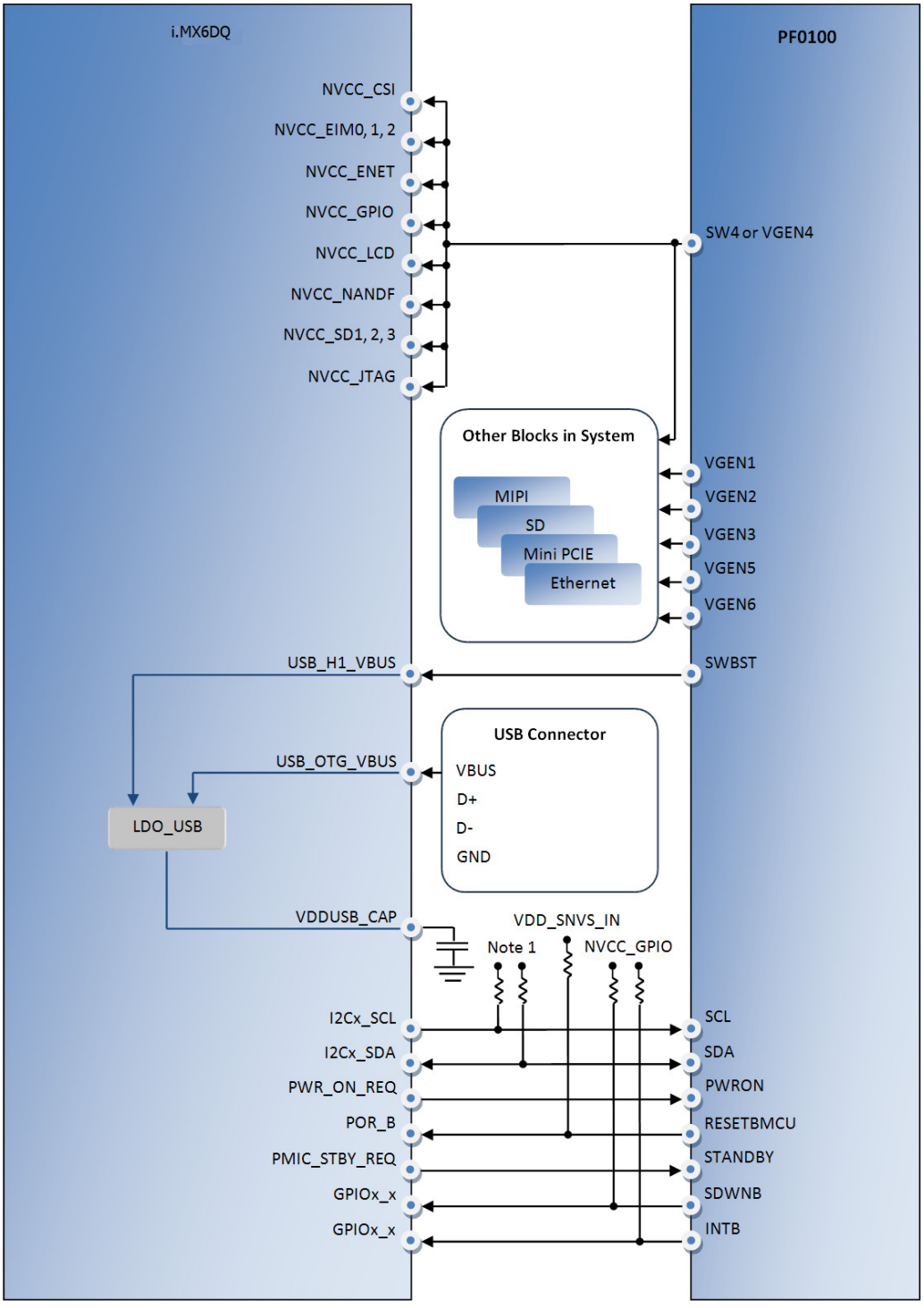


Figure 4-1. i.MX6x/PF0100 connection diagram, 1 of 2



[1] Choose the pullup voltage for the I^2C lines based on the I^2C channel chosen. For example, for the I2C3 channel, the corresponding voltage domain is NVCC_GPIO.

Figure 4-2. i.MX6x/PF0100 connection diagram, 2 of 2

4.5 Video power recommendations

VDD_PU_CAP is the supply for the internal video processing units (VPU). For video intensive operations, the VPU requires a large amount of power and may undergo large swings of instantaneous current requirements. Therefore, the power supply to the VPU must be designed to handle relatively large surges of current at high frequencies from the original source to the processor input for power (VDD_SOC_IN) and at the output of the internal regulator for VPU operations (VDD_PU_CAP). The following list provides recommendations for each specific point along the current supply path. It may be necessary to implement all of these recommendations to ensure that one particular point along the supply path does not become a current choke point.

- The voltage with which VDD_SOC_IN will be fed must have a maximum tolerance of ± 25 mV. PF0100's SW1C is already designed with this tolerance. Care must be taken if the design uses a different regulator.
- VDD_SOC_CAP and VDD_PU_CAP bulk capacitance must be equal to 22 µF so that start up current through the on board LDOs is reduced. For i.MX 6QuadPlus/6DualPlus designs, either two 22 µF or a single 47 µF bulk capacitor must be connected to VDD_PU_CAP.
- These bulk capacitors must be very close to the VDD_SOC_CAP and VDD_PU_CAP pins respectively and the connecting traces must be as thick as the design allows so the ability of being a bulk capacitor for high speed operations is not limited.
- VDD_SOC_IN requires 66 µF of bulk capacitance because it supplies power for both VDD_SOC_CAP and VDD_PU_CAP.

Chapter 5
Using the Clock Connectivity Table

This chapter provides a reference table of the root clock default speed and a list of the i.MX modules available to exit stop mode.

5.1 Introduction

This chapter provides a reference table of the root clock default speed and a list of the i.MX modules available to exit stop mode.

5.2 Root clocks

Clock connectivity is described in the "System Clocks Connectivity" section in the CCM chapter of the chip reference manual. This section contains a series of tables that describe the clock inputs of each module and which clock is connected to it.

NOTE

In some cases, a clock is associated with an external interface and is sourced from a pad (mainly through IOMUX) and not from the CCM. Such clocks do not appear in the clock connectivity table. They are found in the "External Signals and Pin Multiplexing" chapter.

Clock gating is done with the low power clock gating (LPCG) module based on a combination of the clock enable signals. For information about how the clock gating signals are logically combined, see the LPCG section in the CCM chapter of the chip reference manual.

Table 5-1 lists the available clock sources and the default frequencies that are configured by design. In some cases, users need to divide the clock inside the module when the maximum frequency is used in order to meet the protocol requirements. The CCM (clock controller module) generates and drives the clock sources.

For information about how the root clocks are generated, see the clock generation diagrams in the CCM chapter of the appropriate reference manual.

Table 5-1. Clock roots

Clock Root Name (from CCM)	Description	Default frequency [MHz]
ARM_CLK_ROOT	ARM core clock	792
MMDC_CH0_CLK_ROOT	Multi Mode DDR Controller	528
AHB_CLK_ROOT	AMBA Bus	132

Table 5-1. Clock roots (continued)

Clock Root Name (from CCM)	Description	Default frequency [MHz]
IPG_CLK_ROOT	Inter-packet Gap	66
PERCLK_CLK_ROOT		66
USDHC1_CLK_ROOT	Ultra Secured Digital Host Controller	198
USDHC2_CLK_ROOT		198
USDHC3_CLK_ROOT		198
USDCH4_CLK_ROOT		198
SSI1_CLK_ROOT	Synchronous Serial Interface	63.525
SSI2_CLK_ROOT		63.525
SSI3_CLK_ROOT		63.525
GPU2D_AXI_CLK_ROOT	2D Graphics Processing Unit	270
GPU3D_AXI_CLK_ROOT		270
PCIE_AXI_CLK_ROOT	PCI Express	270
VDO_AXI_CLK_ROOT	Video Data Order Adapter	270
AXI_CLK_ROOT	Advanced eXtensible Interface	270
IPU1_HSP_CLK_ROOT	IPU High-Speed Processing Clock	264
IPU2_HSP_CLK_ROOT		264

5.3 Waking the core up from stop mode

The following modules can wake the core up from stop mode.

- CAN
- ECSPI
- EIM
- ENET
- EPIT
- GPC
- GPIO
- GPT
- I2C
- KPP
- PCIE

- SDMA
- UART
- USB

Chapter 6
Avoiding Board Bring-up Problems

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.

6.1 Using a current monitor to avoid power pitfalls

Excessive current can cause damage to the board. Avoid this problem by using a current-limiting laboratory supply set to the expected typical main current draw (at most). Monitor the main supply current with an ammeter when powering up the board for the first time. You can use the supply's internal ammeter if it has one. By monitoring the main supply current and controlling the current limit, any excessive current can usually be detected before permanent damage occurs.

6.2 Using a voltage report 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 voltage values to be measured. The following guidelines help produce the best voltage 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.MX6 processor.

Ensure that the supplies that are powering the i.MX6 meet the DC electrical specifications as listed in your chip-specific data sheet.

The following table illustrates how a sample voltage report table helps detect errors. 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 6-1. Sample voltage report

Source	Net name	Expected (V)	Measured (V)	Measured point	Comment
Main	5V0	5.0	5.103	C5.1	—
3.3 V discrete reg	3V3_DELAYED	3.35	3.334	SH1	Requires LDO3 to enable
PMIC Switcher 1	VDDARM	1.375	1.377	SH2	—
PMIC Switcher 2	VDDSOC	1.375	1.376	SH3	—
PMIC Switcher 3	1V5_DDR	1.5	1.501	SH4	—
PMIC LDO1	1V8	1.8	1.802	TP9	—
PMIC LDO2	2V5	2.5	0.3	TP5	—
VREFDDR	0V75_REFDDR	0.75	0.751	C8.1	50% of 1V5_DDR
Coin Cell	3V0_STBY	3.0	3.006	TP1	—
i.MX6	VDDARM_CAP	1.1	1.114	C6.1	—
i.MX6	VDDHIGH_CAP	2.5	2.515	SH5	—
i.MX6	VDDSNVS_CAP	1.0	1.016	TP2	—

6.3 Checking for clock pitfalls

Problems with the external clocks are another common source of board bring-up issues. Ensure that all of the 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_P/N on the i.MX6 can be an advantage if low jitter or special frequency clock sources are required by modules driven by CLK1_P/N or CLK2_P/N. See the CCM chapter in your i.MX6 chip reference manual for details. If a 32.768 kHz crystal is not connected to the i.MX6, 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_P/N can be used as oscillator inputs for low jitter special frequency sources.
- CLK1_P/N and CLK2_P/N are optional.

In addition to probing the external input clocks, the internal clocks can also be checked 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.

6.4 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.MX6 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.MX6 boots. For a more detailed description about the different boot modes, see the system boot chapter of the chip reference manual.

The following figures show two examples of the power-up sequence.

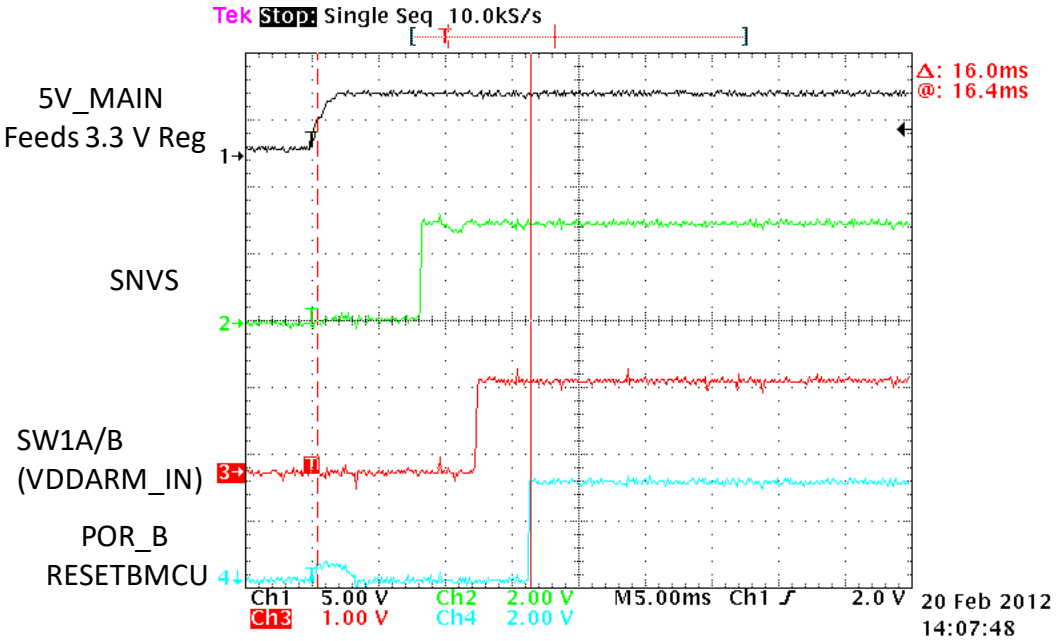


Figure 6-1. Power-up sequence example 1

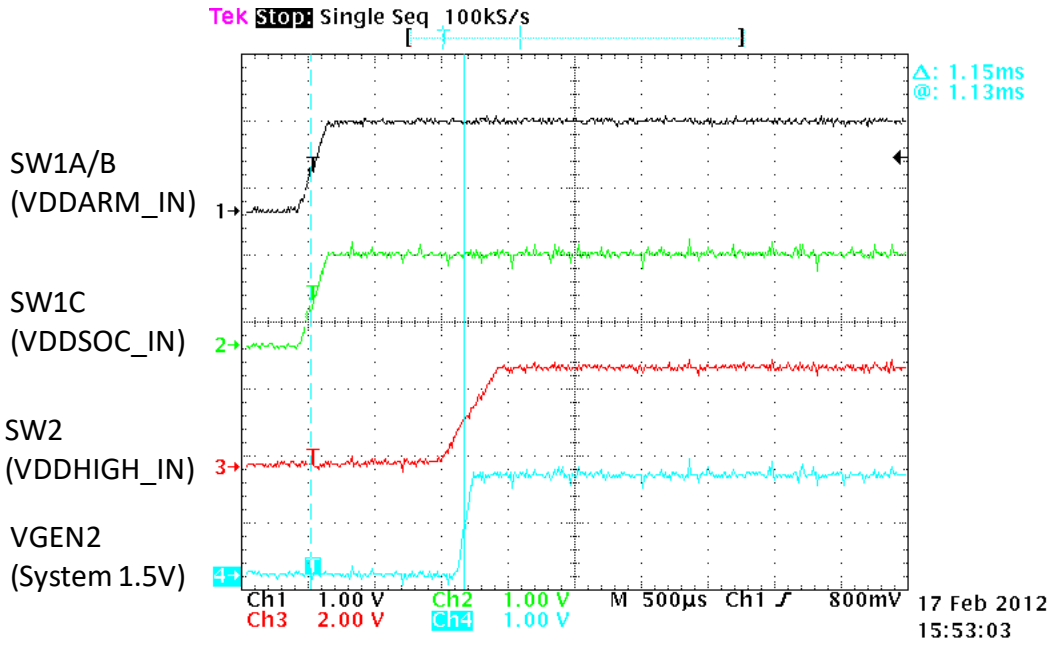


Figure 6-2. Power-up sequence example 2

6.5 Sample board bring-up checklist

Table 6-2 provides a 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 6-2. Board bring-up checklist

Checklist Item	Details	Owner	Findings & status
Note: The following items must be completed serially.			
1. Perform a visual inspection.	Check major components to make sure nothing has been misplaced or rotated before applying power.		
2. Verify all i.MX6 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.MX6 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.MX6 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.MX6 boot up. **Note:** NVCC_LVDS2V5 must be powered when using the chip DDR interface. This power input is used as the Pre-Driver power source for the DDR I/O pads.		

Table 6-2. Board bring-up checklist (continued)

Checklist Item	Details	Owner	Findings & status
3. 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.MX6 data sheet for details about power-up sequencing.		
4. Measure/probe input clocks (32 kHz, 24MHz, others).	Without a properly running clock, the i.MX6 will not function properly.		
5. Check JTAG connectivity.	This is one of the most fundamental and basic access points to the i.MX6 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.MX6 in system. The on-chip internal RAM starts at address 0090_0000h and is 256 Kbytes in density. 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.MX6 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.		
Run basic DDR initialization and test memory.	**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. It is also recommended that users recheck the schematic to ensure that the DDR memory has been connected to the i.MX6 correctly.		

Chapter 7
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

7.1 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.

7.2 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 7-1. 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 pipe (|) character
[File Name]	Yes	Name of this file. All file names must be lower case. The file name extension for an IBIS file is .ibs
[File Rev]	Yes	The revision level of this file. The specification contains guidelines for assigning revision levels.
[Date]	No	Date this file was created
[Source]	No	The source of the data in this file. Data is taken from a simulation and validated on the board.
[Notes]	No	Component or file-specific notes.
[Disclaimer]	No	May be legally required
[Copyright]	No	The file's copyright notice

Example 7-1. Header Information

```
[IBIS Ver]       4.2
[Comment Char]   |_char
[File Name]      21x21_imx6q_autmtv_003.ibs
[File Rev]       003
[Date]           Wed Mar 14 14:22:34 2012
[Source]         FSL Viper 2012.03.14
[Notes]
```

7.3 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 7-2. Component and Pin Information

Keyword	Required	Comment
[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).

Table 7-2. Component and Pin Information (continued)

Keyword	Required	Comment
[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 7-2. Component and pin information

```
[Component]         iMX6Q
[Manufacturer]      FREESCALE
[Package]
| variable       typ                 min                 max
R_pkg            0.3508997           0.0028730           0.895806
L_pkg            2.62395nH           0.07145nH           5.71558nH
C_pkg            3.89344pF           0.59645pF           29.3042pF
|
[Pin]   signal_name      model_name     R_pin      L_pin      C_pin
A2      PCIE_REXT          gpio        0.646088  4.19621nH   1.57274pF
A3      PCIE_TXM           gpio        0.615102  4.09171nH   1.58877pF
...

[Pin Mapping]    pulldown_ref    pullup_ref
A2               GND             PCIE_VPH
A3               GND             PCIE_VPH
...
[Diff Pin]  inv_pin  vdiff  tdelay_typ tdelay_min tdelay_max
A6          B6       NA        NA         NA         NA
A10         B10      NA        NA         NA         NA
A12         B12      NA        NA         NA         NA
...

[Model Selector] ddr
ddr3_sel11_ds111_mio                    DDR, 1.5V, ddr3 mode, 34 Ohm driver impedance
rgmii_sel11_ds111_mio                   DDR, 2.5V, 31 Ohm driver impedance
```

7.4 Model information

The [Model] keyword starts the description of the data for a particular buffer. Table 7-3 shows the main sets of parameters and keywords, composing the model definition.

Table 7-3. 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, typ and max pullup and power clamp data.
[Pulldown] [Pullup] [GND_clamp] [POWER_clamp]	IV information. For more details, see Section 7.4.1, " IV information."
[Ramp] [Rising Waveform] [Falling Waveform]	VT information. For more details, see Section 7.4.2, "VT information."
[Test Data] [Rising Waveform Near] [Rising Waveform Far] [Falling Waveform Near] [Falling Waveform Far] [Test Load]	VT golden model information. For more details, see Section 7.4.3, "Golden Model VT information."

7.4.1 IV information

IV information is 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.

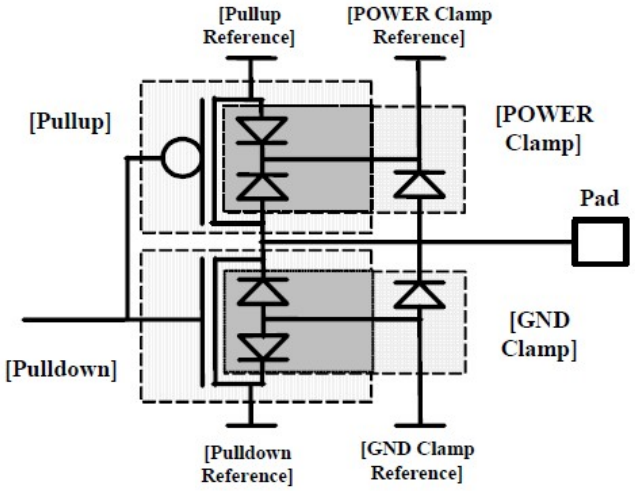


Figure 7-1. Model IV parameters' structure

7.4.2 VT information

Table 7-4. Ramp and waveform keywords

Keyword	Required	Comment
[Ramp]	Yes	Basic ramp rate information, given as a dV/dt_r for rising edges and dV/dt_f for falling edges, see the following equation. $\frac{dV}{dt} = \frac{\text{20 \% to 80\% voltage swing}}{\text{time taken to swing above voltage}}$ **Note:** The dV value is the 20% to 80% voltage swing of the buffer when driving into the specified load, R_load (for [Ramp], this load defaults to 50). For CMOS drivers or I/O buffers, this load is 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 7-3. Ramp and waveform keywords example

```
[Ramp]
| variable       typ                 min                 max
dV/dt_r  0.4627/0.3456n      0.4326/0.4568n      0.4962/0.3030n
dV/dt_f  0.4546/0.3481n      0.4272/0.3918n      0.4774/0.3569n
R_load = 0.2400k
|
[Rising Waveform]
R_fixture= 0.2400k
V_fixture= 0.0
V_fixture_min= 0.0
V_fixture_max= 0.0
|time     V(typ)              V(min)              V(max)
|
|0.0S              0.3369uV          12.4052uV           41.7335nV
|19.7866fS         0.6730uV          12.7375uV           0.3823uV
|20.8863fS         0.6917uV          12.7519uV           0.4013uV
|21.9489fS         0.7058uV          12.7657uV           0.4196uV
...
|
[Falling Waveform]
R_fixture= 0.2400k
V_fixture= 0.0
V_fixture_min= 0.0
V_fixture_max= 0.0
|time     V(typ)              V(min)              V(max)
|
|0.0S     0.7711V            0.7211V             0.8270V
|0.3334nS 0.7711V            0.7211V             0.8270V
|0.3445nS 0.7711V            0.7211V             0.8269V
...
```

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 they 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 a 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]

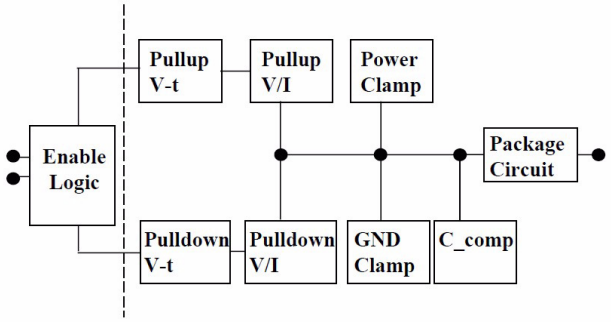


Figure 7-2. Model data interpretation

7.4.3 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 behavioral simulation results against the transistor level circuit model from which the IBIS model parameters originated.

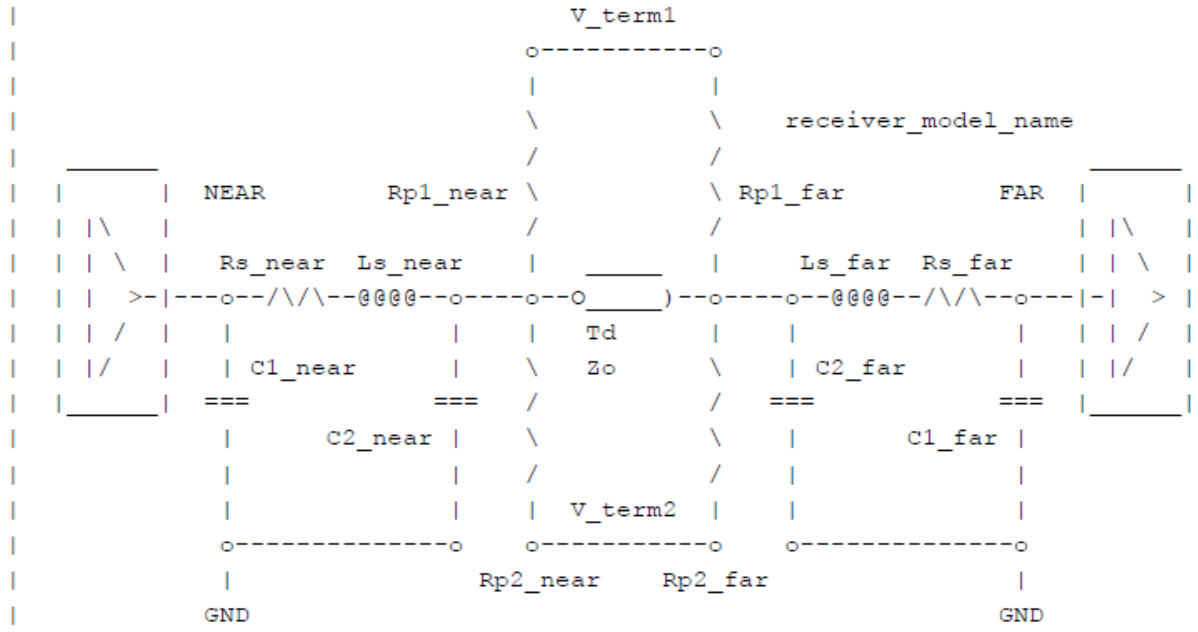


Figure 7-3. Generic test load network

The following table explains the golden waveform keywords.

Table 7-5. 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 behavioral simulation results against the transistor level circuit model from which the IBIS model parameters originated.
[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 by Figure 7-3.
[Test Load]	Yes	• Defines a test load network and its associated electrical parameters for reference by golden waveforms under the [Test Data] keyword. • If Test_load_type is Differential, the test load is a pair of the circuits shown in . If the R_diff_near or R_diff_far subparameter is used, a resistor is connected between the near or far nodes of the two circuits. • If Test_load_type is Single_ended, R_diff_near and R_diff_far are ignored.

7.5 NXP naming conventions for model names and usage in i.MX6 IBIS file

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.

7.5.1 [Model Selector] ddr

The “ddr” model type supports both the DDR and the RGMII protocol signals.

7.5.1.1 DDR [Model Selector]

“ddr” models exist for DDR3, DDR3L, and LPDDR2 protocols.

This model has the following parameters:

- DDR protocol
- DDR IO type
- Drive strength
- ODT enable/disable

The IBIS model name is composed from the parameters’ values in two ways, as follows:

- Without active ODT circuit:

  `<ddr protocol>_sel<ddr_type>_ds<drive_strength>_mio`
- With active ODT circuit:

  `<ddr protocol>odt_t<ODT_value>_sel<ddr_type>_mi`

DDR write models ("_mio" suffix) have no simulated ODT, as ODT is disabled during write. Write models' DS parameter is meaningful and changes to describe the different levels of drive strength.

DDR read models ("_mi" suffix) have no meaningful DS parameter, as no driving happens during read. Read models' ODT parameter is meaningful and changes to describe different levels of ODT impedance.

DDR Protocol	Selected according to the used DDR. DDR IO voltage level is selected accordingly.
DDR IO Type	Controlled by the IOMUXC_SW_PAD_CTL_GRP_DDR_TYPE[19:18] register in IOMUXC (IOMUX controller) DDR_SEL bits, to select between DDR3 & LPDDR2.
Drive strength	Controlled by bits [5:3] (DSE) of the following registers in IOMUXC (IOMUX controller): IOMUXC_SW_PAD_CTL_PAD_DRAM_SDCLK_x (2 registers) IOMUXC_SW_PAD_CTL_PAD_DRAM_CAS IOMUXC_SW_PAD_CTL_PAD_DRAM_RAS IOMUXC_SW_PAD_CTL_PAD_GRP_ADDDS IOMUXC_SW_PAD_CTL_PAD_DRAM_RESET IOMUXC_SW_PAD_CTL_PAD_DRAM_SDCKEx (2 registers) IOMUXC_SW_PAD_CTL_PAD_DRAM_SDODTx (2 registers) IOMUXC_SW_PAD_CTL_PAD_GRP_CTLDS IOMUXC_SW_PAD_CTL_PAD_DRAM_SDQSx (8 registers) IOMUXC_SW_PAD_CTL_PAD_DRAM_BxDS (8 registers) IOMUXC_SW_PAD_CTL_PAD_DRAM_DQMx (8 registers)
ODT value	Controlled by bits [18:16], [14:12], [10:8], and [6:4] in MPODTCTRL register of MMDC.

Example 7-4. [Model Selector] DDR in IBIS file

```
ddr3_sel11_ds111_mio                    DDR, 1.5V, ddr3 mode, 34 Ohm driver impedance
...
lpddr2_sel10_ds111_mio                  LPDDR, 1.2V, lpddr2 mode, 34 Ohm driver impedance
lpddr2_sel10_ds110_mio                  LPDDR, 1.2V, lpddr2 mode, 40 Ohm driver impedance
...
```

See the register description in the IOMUXC chapter in the chip reference manual for further details about this model.

7.5.1.2 RGMII

This model has the following parameters:

- RGMII voltage
- Drive strength

The IBIS model name is composed from the parameters' values as follows:

```
rgmii_sel11_ds<drive_strength>_mio
```

Voltage Level	IBIS currently supports only the 2.5 V option. 2.5 V is applied to NVCC_RGMII. No further register programing is required.
Drive strength	Controlled by bits [5:3] (DSE) of the following registers in IOMUXC (IOMUX controller): IOMUXC_SW_PAD_CTL_PAD_RGMII_TXC IOMUXC_SW_PAD_CTL_PAD_RGMII_TX_CTL IOMUXC_SW_PAD_CTL_PAD_RGMII_TDx (4 registers) IOMUXC_SW_PAD_CTL_PAD_RGMII_RXC IOMUXC_SW_PAD_CTL_PAD_RGMII_RX_CTL IOMUXC_SW_PAD_CTL_PAD_RGMII_RDx (4 registers)
IO Type	Regardless of the voltage level, he ddr_sel of IOMUXC_SW_PAD_CTL_GRP_DDR_TYPE_RGMII should always be set to '11'.

Example 7-5. [Model Selector] RGMII in IBIS file

```
rgmii_sel11_ds111_mio                    DDR, 2.5V, 31 Ohm driver impedance
rgmii_sel11_ds110_mio                    DDR, 2.5V, 37 Ohm driver impedance
rgmii_sel11_ds101_mio                    DDR, 2.5V, 45 Ohm driver impedance
...
```

7.5.2 [Model Selector] gpio

This model has the following parameters:

- Voltage level
- Drive strength
- Slew rate
- Speed

The IBIS model name is composed from parameters' values as follows:

```
gpio<voltage_level>_ds<drive_strength>_sr<slew_rate(1 bit)><speed(2 bits)>_mio
```

Voltage level	For i.MX6 chips, there are no user configurations for the voltage level because the GPIO cell senses the NVCC and auto-configures itself accordingly. The IBIS user can choose between high and low voltage by selecting a different model at [Model Selector].
Drive strength	Controlled by the DSE bits (bits [5:3]) in the IOMUXC_SW_PAD_CTL_PAD_<pad name>.
Slew rate	Controlled by the SRE bit (bit 0) in the IOMUXC_SW_PAD_CTL_PAD_<pad name>.

Speed	Controlled by the SPEED bits (bits [7:6]) in the IOMUXC_SW_PAD_CTL_PAD_<pad name>.

Example 7-6. [Model Selector] gpio in IBIS file

```
[Model Selector] gpio
gpiohv_ds111_sr111_mio                  GPIO, 3.3V,extra drive, fast slew, max frequency
gpiohv_ds111_sr110_mio                  GPIO, 3.3V,extra drive, fast slew, fast frequency
gpiohv_ds111_sr101_mio                  GPIO, 3.3V,extra drive, fast slew, medium frequency
gpiohv_ds111_sr100_mio                  GPIO, 3.3V,extra drive, fast slew, slow frequency
...
gpiolv_ds111_sr111_mio                  GPIO, 1.8V,extra drive, fast slew, max frequency
gpiolv_ds111_sr110_mio                  GPIO, 1.8V,extra drive, fast slew, fast frequency
gpiolv_ds111_sr101_mio                  GPIO, 1.8V,extra drive, fast slew, medium frequency
...
```

See the register description in the IOMUXC chapter in the chip reference manual for further details about this model.

7.5.3 [Model Selector] lvds

A single model is available for LVDS, as no configurable parameters exist for this IO model.

The LVDS model is available not only for the LVDS port signals, but also for the general purpose CLK1_x and CLK2_x, who share the same IO model.

Example 7-7. [Model Selector] lvds in IBIS file

```
[Model Selector] lvds
lvds_mio                              LVDS, Vos = 1.2V, Voh = 1.375, Vol = 1.025, Vovdd = 2.5
...
```

7.5.4 [Model Selector] mlb

The following two models are available for MLB, as no configurable parameters exist for this IO model.

Example 7-8. [Model Selector] lvds in IBIS file

```
[Model Selector] mlb
mlb_sig_data_mio                       MLB, Signal/Data Transceiver, Vod = 400, Vid = 100
mlb_clk_mi                             MLB, CLK Receiver, Vid = 100
...
```

7.5.5 [Model Selector] USB

There are eight models are available for USB.

Example 7-9. [Model Selector] usbdpdm in IBIS file

```
[Model Selector] usbdpdm
FS_driver            Full Speed USB driver
LS_driver            Low Speed USB driver
HS_driver            High Speed USB driver
hs_input             High Speed USB Input
```

```
fs_host_input        Full Speed Host Input
fs_device_dp_input   Full Speed Device dp Input
fs_device_dm_input   Full Speed Device dm Input
ls_host_input        Low  Speed Host Input
```

7.5.6 List of pins not modeled in the i.MX6 IBIS file

The following table provides a list of analog or special interface pins that are not modeled in the i.MX6 IBIS file.

Table 7-6. i.MX6 pins not supported by IBIS

ANALOG	MIPI	HDMI	PCIe	SATA
RTC_XTALI	CSI_CLK0M	HDMI_CLKM	PCIe_RXM	SATA_RXM
RTC_XTALO	CSI_CLK0P	HDMI_CLKP	PCIe_RXP	SATA_RXP
XTALI	CSI_D0M	HDMI_D0M	PCIe_TXM	SATA_TXM
XTALO	CSI_D0P	HDMI_D0P	PCIe_TXP	SATA_TXP
ZQPAD	CSI_D1M	HDMI_D1M		
	CSI_D1P	HDMI_D1P		
	CSI_D2M	HDMI_D2M		
	CSI_D2P	HDMI_D2P		
	CSI_D3M	HDMI_DDCCEC		
	CSI_D3P	HDMI_HPD		
	DSI_CLK0M			
	DSI_CLK0P			
	DSI_D0M			
	DSI_D0P			
	DSI_D1M			
	DSI_D1P			

NOTE

In i.MX6 IBIS, some of the above unsupported pins are described as "GPIO" cells. These are no more than placeholders and cannot be used for signal modeling.

7.6 Quality assurance for the IBIS models

The IBIS models are validated against the IBIS specification, which provides a way to objectively measure the correlation of model simulation results with reference transistor-level spice simulation or measurements.

Correlation	The process of making a quantitative comparison between two sets of I/O buffer characterization data, such as lab measurement vs. structural simulation or behavioral simulation vs. structural simulation.
Correlation Level	A means for categorizing I/O buffer characterization data based on how much the modeling engineer knows about the processing conditions of a sample component and which correlation metric he or she used.

All models (GPIO, DDR, LVDS, MLB) have passed the following checks:

- IBISCHK without errors or unexplained warnings
- Data for basic simulation checked
- Data for timing analysis checked
- Data for power analysis checked
- Correlated against Spice simulations

Validation reports can be provided upon demand.

7.7 IBIS usage

NXP board designers used the i.MX6Q IBIS model with the Hyperlynx tool by Mentor Graphics. The HyperLynx version used was HyperLynx v8.1.1 + Update 2.

Effective board design results achieved after loading:

- i.MX6Q IBIS model.
- Companion IC IBIS models.
- Board model in HyperLynx format.

Board simulations for various GPIO, LVDS, and DDR signals were then run.

7.8 References

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

- IBIS Open Forum (http://www.eda.org/ibis/)

  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 (http://eda.org/pub/ibis/ver5.0/)

Chapter 8
Using the Manufacturing Tool

8.1 Overview

The i.MX manufacturing tool is designed to program firmware onto storage devices such as NAND or eSD and preload the data area with media files in an efficient and convenient manner. It is intended for NXP Semiconductors customers or their OEMs who plan to mass manufacture i.MX-based products.

The application is not designed to test the devices or to diagnose manufacturing problems. Devices initialized with this application still need to be functionally verified.

8.2 Feature summary

The tool includes the following features:

- Continuous operation—operations automatically begin with the connection of a new device, and multiple operations such as update and copy can be linked together seamlessly.
- Enumeration—static-ID firmware loaded into RAM in recovery-mode prevents Windows® from enumerating every device.
- AutoPlay—various Windows® ‘pop-up’ application and status messages, such as Explorer in Windows® XP and Windows 7.

In addition, the following characteristics improve the tool’s ease of use:

- An independent process bar is set up for each physical USB port.
- The tool begins processing with the connection of the first device detected and allows users to replace each device after completion instead of needing to wait for all devices to complete.
- The tool uses color-based indicators to indicate the work status on each of the ports.
  - Blue indicates the device is being processed.
  - Green indicates the device was successfully processed and that the programmed device can be replaced with a new one independent of the of the device’s progress.
  - Red indicates the device failed to process.

8.3 Other references

For more detailed information about the manufacturing tool, see the following documents included in the manufacturing tool release package. Contact your local NXP sales office for assistance obtaining documents if needed:

- For detailed information about how to use the manufacturing tool, see *Manufacturing Tool V2 Quick Start Guide*.

- For detailed information about how to script the processing operations of the manufacturing tool, see the *Manufacturing Tool V2 UCL User Manual*.
- For information about how to generate the manufacturing tool firmware for Linux and Android, see *Manufacturing Tool V2 Linux or Android Firmware Development Guide*.
- For the change list and known issues, see *Manufacturing Tool V2 Release Notes*.

Chapter 9
Using BSDL for Board-level Testing

9.1 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.

9.2 How BSDL functions

A 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 a 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 that 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.

9.3 Downloading the BSDL file

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

9.4 Pin coverage of BSDL

Each pin is defined as a port within the BSDL file. You can open the file with a text editor (like Wordpad) 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.

9.5 Boundary scan operation

The boundary scan operation is controlled by:

- BOOT_MODE0, BOOT_MODE1, 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 JTAG 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 the fusemap tables.

Table 9-1. 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 (Development)
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

9.6 I/O pin power considerations

The boundary scan operation uses each of the available device pins to drive or read values within a given system. Therefore, the power supply pin for each specific module must be powered in order for the IO buffers to operate. This is straightforward for the digital pins within the system.

NOTE

BSDL was only tested at 1.8 V.

SATA and PCIe are not digital interfaces, but these modules provide built-in support for the IEEE 1149.6 extension for AC testing of their pins. Therefore, these modules must also be powered when utilizing a scan chain that contains the pins from these modules, or the scan chain does not function properly.

Chapter 10
Using the RMII Interface

10.1 Overview

In the Ethernet RMII interface, the RMII_REF_CLK is input into the ENET module through a GPIO pad, regardless of whether the clock is provided from an internal or external source. This chapter provides supporting instructions on how to treat this GPIO pad at the PCB level.

NOTE

This chapter only covers the required hardware and register settings. Modifications to the Ethernet driver or its initialization code are beyond its scope. For this information, see your BSP documentation.

NOTE

This chapter does not apply to the i.MX 6DualPlus/6QuadPlus, additional internal muxing was added to remove this GPIO pad dependency. Refer to the i.MX 6DualPlus/6QuadPlus Reference Manual for further details.

10.2 Configuring the RMII signal connections

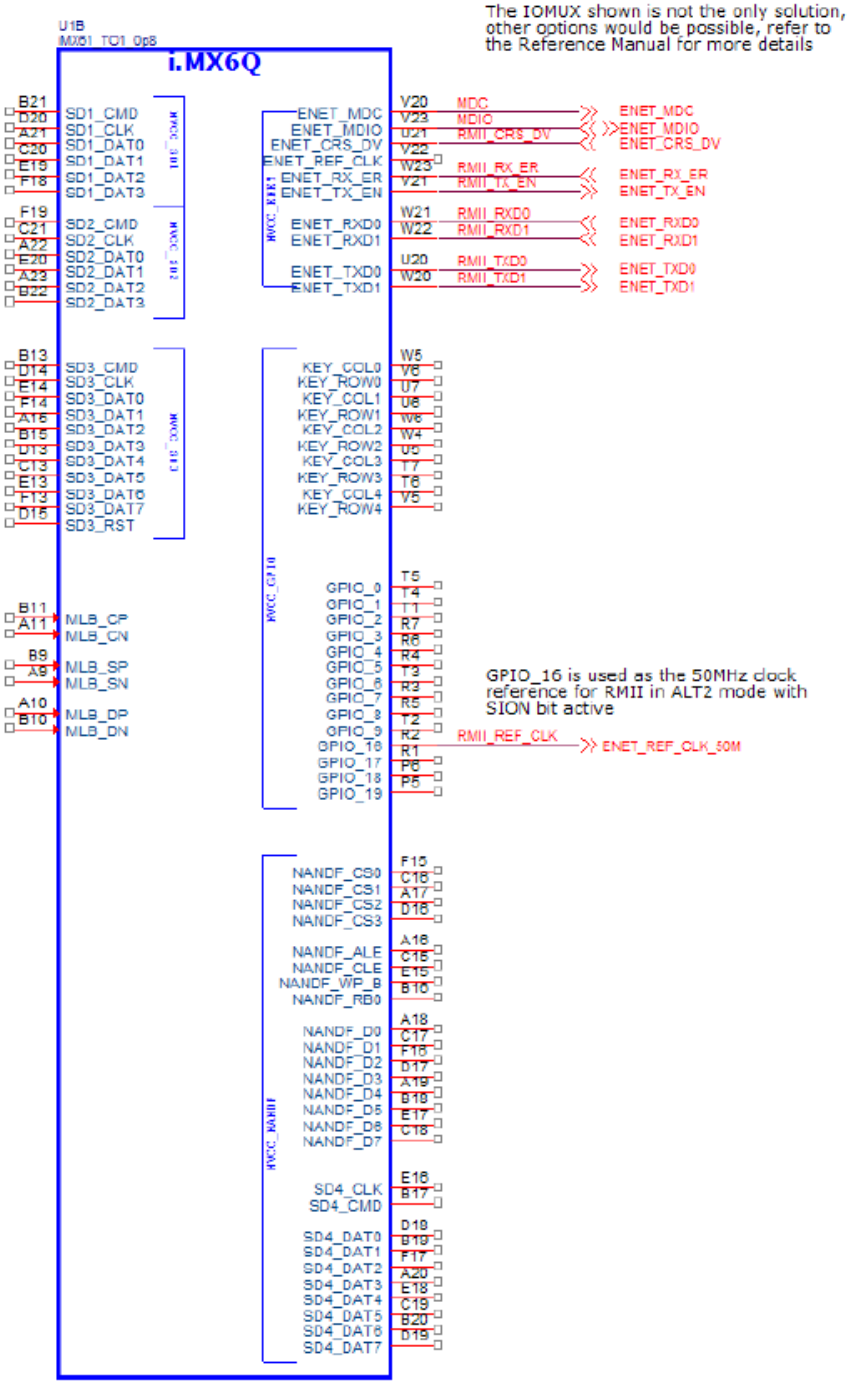


Figure 10-1. Reference schematic, part 1 of 2

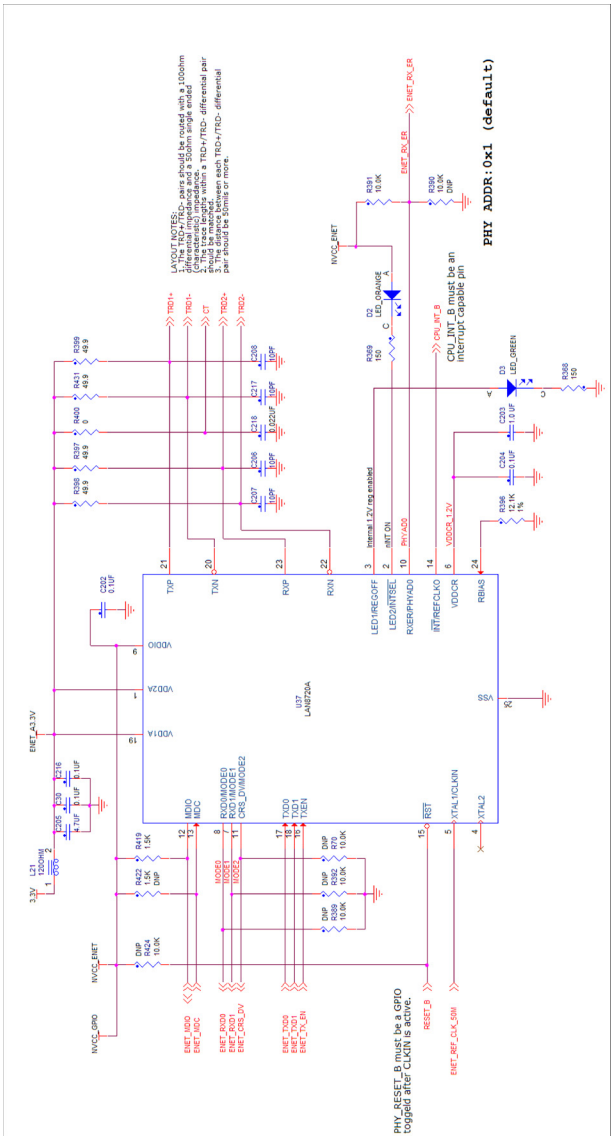


Figure 10-2. Reference schematic, part 2 of 2

A reference schematic shows the connections required to use the RMII interface. These signal connections are generally self-explanatory or explained in the chip reference manual. However, there are some required modifications.

10.3 Generating the reference clock

The Ethernet MAC needs to have a reference clock, which can be generated in one of the following three ways:

- On chip clock generator
- By an external oscillator

- By the RMII PHY

Note that the pin labeled “ENET_REF_CLK” in Figure 12-2 is only required by the full MII interface. It is not used by the RMII interface.

10.4 Generating the reference clock on chip

There are two possible pins that can either source or sink the reference clock: GPIO_16 and RGMII_TX_CTL. The GPIO_16 pin is the preferred choice because it has the advantage of being in a high voltage IO domain, which means it can be used at the standard 3.3 V IO voltage levels. RGMII_TX_CTL should only be used if pin function loadings are such that GPIO_16 is unavailable.

10.4.1 Using the GPIO_16 pin to generate the reference clock

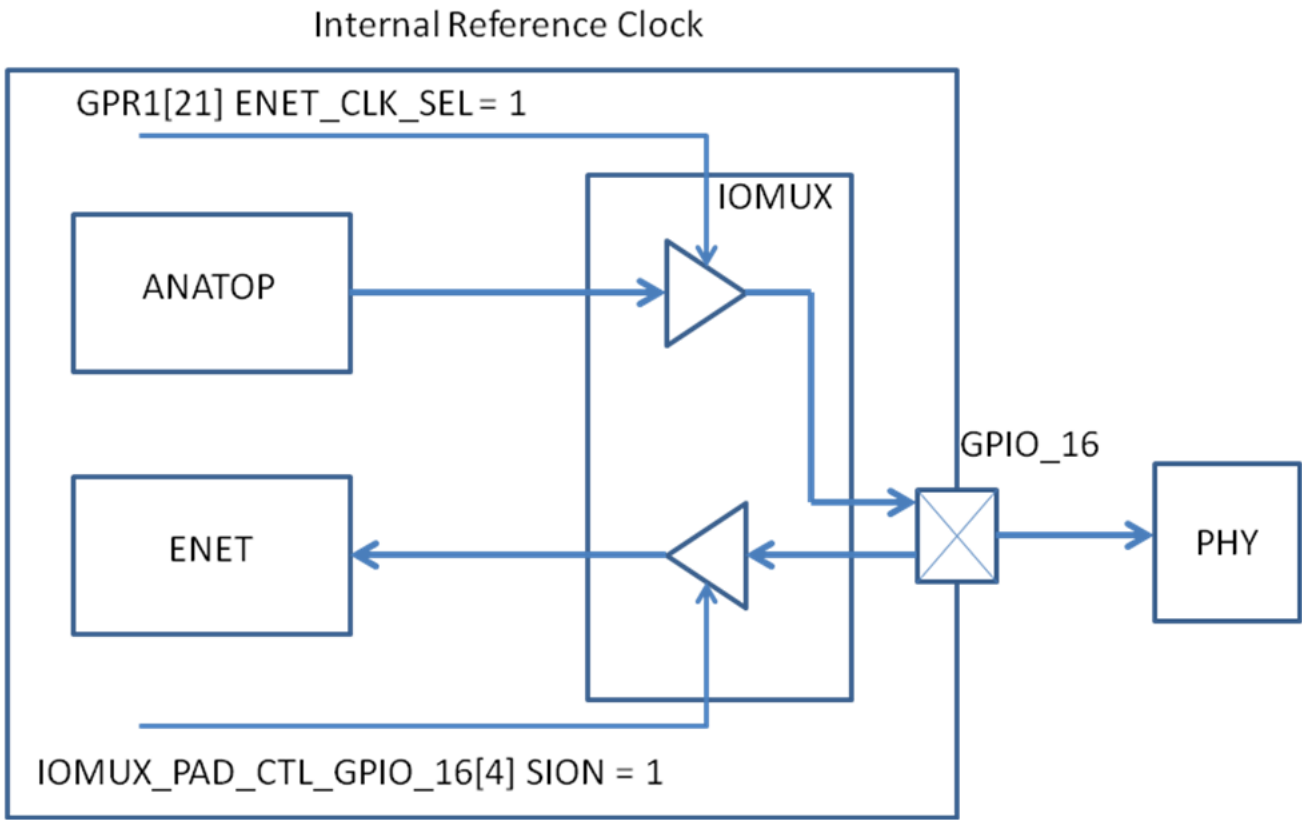


Figure 10-3. Using the GPIO_16 pad

Note that the block labeled “ANATOP” is really the Analog Ethernet PLL. See your chip reference manual for its control register figure. Bits 1–0 of CCM_ANALOG_PLL_ENETn control the frequency fed to the Ethernet MAC and should be set to 01b to obtain 50 MHZ.

To use GPIO_16 as the RMII reference clock source, use the following:

- Set mode to ALT2 (MUX_MODE[2:0] = 010).

- Set the SION bit. Note that this is not required because the function setting controls the signal path, but it is good practice as it reminds the user that the clock needs to fed back into the Ethernet MAC.
- For the internal clock case, set GPR1[21].

GPR1[21] controls the actual clock source.

- When cleared, it obtains the ENET Tx reference clock from a pad (external OSC for both external PHY and internal controller).
- When set, it obtains the ENET Tx reference clock from ANATOP (loopback through pad) and sends out to the external PHY.

The Daisy Chain register also needs to be set correctly to force the input to use the right pin. Note that there is a subtle difference between the i.MX 6Quad/6Dual families and the i.MX 6DualLite/6Solo families that affects the correct setting:

- For i.MX6Quad/6Dual—To use GPIO_16, set DAISY[0] = 1. If DAISY[0] is left at its reset value (0b), RGMII_TX_CTL will be used instead. See the Select Input Register (IOMUX_ENET_REF_CLK_SELECT_INPUT) in the IOMUXC chapter of the i.MX 6Dual/6Quad Reference Manual (IMX6DQRM), available at www.nxp.com.
- For i.MX 6Solo/6DualLite—To use GPIO_16, leave the value of DAISY[0] at is reset value (0b). To use RGMII_TX_CTL, set DAISY[0] = 1.

For further information, see the “DAISY (IOMUXC_ENET_REF_CLK_SELECT_INPUT)” section in the “IOMUX Controller (IOMUXC)” chapter of your chip reference manual. Note that while you can use the default pad settings as shown in the “SW_PAD_CTL (IOMUXC_SW_PAD_CTL_PAD_GPIO16)” section in the IOMUX controller chapter, it may be desirable to set the Slew Rate Field, SRE[0] = 1).

10.4.2 Using RGMII_TX_CTL to generate the reference clock

RGMII_TX_CTL is in the RGMII IO voltage domain, which has a maximum voltage of 2.5 V. Therefore, to use RGMII_TX_CTL, you must use a level shifter to match the RMII PHY voltage levels. This is why GPIO_16 is preferred.

To use RGMII_TX_CTL, set the following:

- In the Daisy Chain register, ensure DAISY[0] is cleared; note that this is its default setting after reset.
- Set RGMII_TX_CTL pad mux register to ALT7.
- If desired, set the SION bit as discussed in Section , “There are two possible pins that can either source or sink the reference clock: GPIO_16 and RGMII_TX_CTL. The GPIO_16 pin is the preferred choice because it has the advantage of being in a high voltage IO domain, which means it can be used at the standard 3.3 V IO voltage levels. RGMII_TX_CTL should only be used if pin function loadings are such that GPIO_16 is unavailable..”
- The RGMII_TX_CTL pad control register, IOMUXC_SW_PAD_CTL_PAD_RGMII_TX_CTL, does not have a slew rate control bit. Slew rate can be controlled by judicious choice of output drive strength in the DSE field, bits 5:3

10.5 Using an external clock

The reference clock can be generated externally by the Ethernet PHY or an external oscillator. The following figure shows a possible configuration.

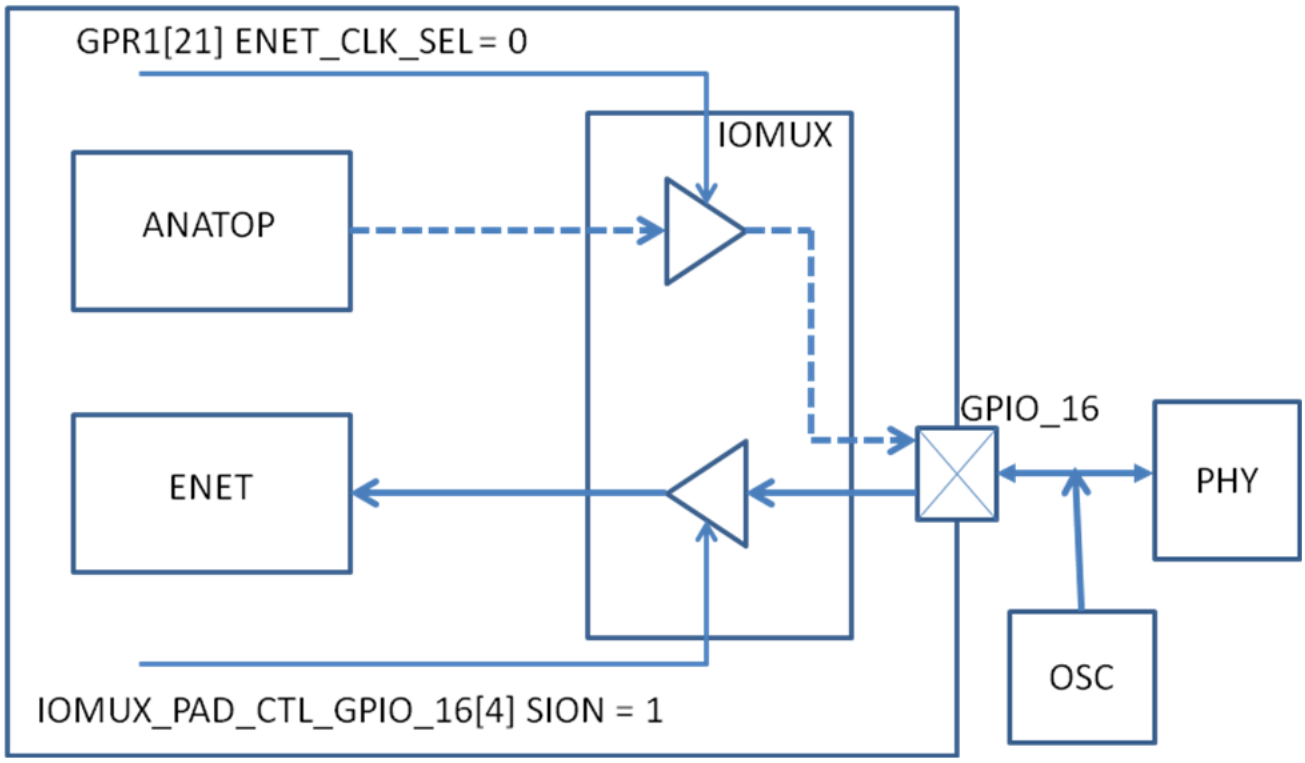


Figure 10-4. External clock configuration (external oscillator shown)

This configuration is almost identical when using an external oscillator or the RMII PHY to supply a clock. The only required modification to an RMII PHY instead of the external oscillator is to clear GPR1[21] (GPR1[21] = 0) to select the external clock input. All other settings remain the same.

Appendix A
Revision History

This table provides the revision history for this document:

Table A-1. Document Revision History

Revision number	Date	Substantive Changes
4	02/2019	• Table 2-1. DDR recommendations: Added row regarding 64-bit LPDDR2 design • Table 2-6. Power and decouple recommendations: Added row regarding compliance with LDO setpoints • Removed figure "Clock supplied by Ethernet PHY. Input at FEC_TX_CLK pin". The figure does not apply to the products covered in this document.
3	05/2018	• Added footnote to Table 2-1.
2	10/2016	• Changed reference to AN3298 in Section 1.3, “Essential reference” to instead refer to AN3300. • Updated row 4 of Table 2-1. • Added new row 5 to Table 2-1 (old row 5 now row 6). • Added new row 7 to Table 2-1. • Updated row 3 of Table 2-2. • Updated row 9 of Table 2-6. • Added row 10 to Table 2-6. • Updated row 7 of Table 2-7. • Updated row 1 of Table 2-8. • Updated all of Table 2-9. • Updated row 2 of Table 2-12. • Updated title of column 1 in Table 2-15. • Updated row 4 (USB2.0) of Table 2-20. • Added footnote to row 9 (SATA) of Table 2-21. • Updated footnote to column 3 of row 9 (SATA) of Table 2-21. • Updated Length - Min values for Byte Group 1–8 in Table 3-3.

Table A-1. Document Revision History (continued)

Revision number	Date	Substantive Changes
1	06/2013	• Added preface to book; Chapter 1, “About This Book” • Chapter 2, “Design Checklist”: - Table 2-1, “DDR recommendations,” recommendation 2, first bullet: Changed “0.1 F” to “0.1 μF.” - Table 2-1, “DDR recommendations,” recommendation 3: Regarding LPDDR2, changed from “DRAM_RESET pull is not necessary” to “DRAM_RESET should be left unconnected.” - Standardized nomenclature for pin names and signal names. For further details on recent standardization of signal names for the i.MX 6 series, see the *i.MX 6 Series Standardized Signal Name Map* (EB792). • Section 9.6, “I/O pin power considerations”: Added note specifying that BSDL was only tested at 1.8 V. • Updated and added content to Table 2-6, Table 2-7 and Table 2-9 • Added new row to table Table 2-10 • Updated third column of all rows in Table 2-13 • Updated row 5 of table Table 2-14 • Corrected references throughout the book • Added footnote to figure Figure 3-5. • Added footnote to Table 4-1 • Updates to Figure 12-3. and Figure 12-4.
0	10/2012	Initial release.