NXP Semiconductors
User's Guide

Document Number: IMX8MNHDG
Rev. 0, 12/2019

i.MX 8M Nano Hardware Developer’s Guide

Contents

1. Overview

This document aims to help hardware engineers design and test the i.MX 8M Nano series processors. It provides examples on board layout and design checklists to ensure first-pass success, and solutions to avoid board bring-up problems.

Engineers should understand board layouts and board hardware terminology.

This guide is released with relevant device-specific hardware documentation, such as datasheets, reference manuals, and application notes. All these documents are available on www.nxp.com/imx8mnanoevk.

1.1. Device supported

This document supports the i.MX 8M Nano (14 x 14 mm package).

1.2. Essential references

This guide is supplementary to the i.MX 8M Nano series chip reference manuals and data sheets. For reflow profile and thermal limits during soldering, see General Soldering Temperature Process Guidelines (document AN3300). These documents are available on www.nxp.com/i.MX8MNANO.

1.3. Supplementary references

1.3.1. General information

The following documents introduces the Arm® processor architecture and computer architecture.

- For information about the Arm Cortex-A53 processor, see: https://www.arm.com/products/processors/cortex-a/cortex-a53-processor.php
- For information about the Arm Cortex-M7 processor, see: https://www.arm.com/products/processors/cortex-m/cortex-m7-processor.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 introduces the high-speed board design:

- Right the First Time- A Practical Handbook on High Speed PCB and System Design - Volumes I & II - Lee W. Ritchey (Speeding Edge) - ISBN 0-9741936- 0-72
- Signal and Power Integrity Simplified (2nd Edition) - Eric Bogatin (Prentice Hall)- ISBN 0-13-703502-0
- High Speed Digital Design- A Handbook of Black Magic - Howard W. Johnson & Martin Graham (Prentice Hall) - ISBN 0-13-395724-1
- High Speed Signal Propagation- Advanced Black Magic - Howard W. Johnson & Martin Graham - (Prentice Hall) - ISBN 0-13-084408-X
- High Speed Digital System Design- A handbook of Interconnect Theory and Practice - Hall, Hall and McCall (Wiley Interscience 2000) - ISBN 0-36090-2
- Signal Integrity Issues and Printed Circuit Design - Doug Brooks (Prentice Hall) ISBN 0-13-141884-X
- PCB Design for Real-World EMI Control - Bruce R. Archambeault (Kluwer Academic Publishers Group) - ISBN 1-4020-7130-2
- Digital Design for Interference Specifications - A Practical Handbook for EMI Suppression -David L. Terrell & R. Kenneth Keenan (Newnes Publishing) - ISBN 0-7506-7282-X
- Electromagnetic Compatibility Engineering - Henry Ott (1st Edition - John Wiley and Sons) - ISBN 0-471-85068-3
- Introduction to Electromagnetic Compatibility - Clayton R. Paul (John Wiley and Sons) - ISBN 978-0-470-18930-6

- Grounding & Shielding Techniques - Ralph Morrison (5th Edition - John Wiley & Sons) - ISBN 0-471-24518-6
- EMC for Product Engineers - Tim Williams (Newnes Publishing) - ISBN 0-7506- 2466-3

1.4. Related documentation

Additional literature will be published when new NXP products become available.

For the list of current documents, see www.nxp.com/i.MX8MNANO.

1.5. Conventions

Table 1 lists the notational conventions used in this document.

Table 1. **Conventions used in the document**

Conventions	Description
`Courier`	Used to indicate commands, command parameters, code examples, and file and directory names.
Italics	Used to indicates command or function parameters.
Bold	Function names are written in bold.
cleared/set	When a bit takes the value zero, it means to be cleared; when it takes a value of one, it means 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 nnnnh	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. 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	An italicized *x* indicates an alphanumeric variable.
n, m	An italicized *n* indicates a numeric variable.

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

1.6. Acronyms and abbreviations

Table 2 defines the acronyms and abbreviations used in this document.

Table 2. **Definitions and acronyms**

Acronym	Definition
ARM^{TM}	Advanced RISC Machines processor architecture
BGA	Ball Grid Array package
BOM	Bill of Materials
BSDL	Boundary Scan Description Language
CAN	Flexible Controller Area Network peripheral
CCM	Clock Controller Module
CSI	MIPI Camera Serial Interface
DDR	Dual Data Rate DRAM
DDR3L	Low voltage DDR3 DRAM
DDR4	DDR4 DRAM
DDRC	DDR Controller
DFP	Downstream Facing Port (USB Type-C)
DRP	Dual Role Port (USB Type-C)
ECSPI	Enhanced Configurable SPI peripheral
EIM	External Interface Module
ENET	10/100/1000 Mbps Ethernet MAC peripheral
EPIT	Enhanced Periodic Interrupt Timer peripheral
ESR	Equivalent Series Resistance
GND	Ground
GPC	General Power Controller
GPIO	General Purpose Input/Output
HDCP	High-bandwidth Digital Content Protection
I^{2}C	Inter-integrated Circuit interface
IBIS	Input output Buffer Information Specification
IOMUX	i.MX 8M Nano 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
LPDDR4	Low Power DDR4 DRAM
LVDS	Low-Voltage Differential Signaling
MLB	Media Local Bus
ODT	On-Die Termination
OTP	One-Time Programmable
PCB	Printed Circuit Board
PCIe	PCI Express
PCISig	Peripheral Component Interconnect Special Interest Group
PDN	Power Distribution Network
PMIC	Power Management Integrated Circuit
POR	Power-On Reset
PTH	Plated Through Hole PCB (i.e. no microvias)
RGMII	Reduced Gigabit Media Independent Interface (Ethernet)
RMII	Reduced Media Independent Interface (Ethernet)
ROM	Read-Only Memory

2. i.MX 8M Nano design checklist

This document provides a design checklist for the i.MX 8M Nano (14 x 14 mm package) processor. The design checklist tables recommend optimal design and provide explanations to help users understand better. All supplemental tables referenced by the checklist appear in sections following the design checklist tables.

2.1. Design checklist table

Table 3. **LPDDR4 recommandations (i.MX 8M Nano)**

Check box	Recommendations	Explanation/Supplemental recommendations
	1. Connect the DRAM_ZN ball on the processor (ball P2) to a 240 Ω, 1% resistor to GND.	This is a reference used during DRAM output buffer driver calibration.
	2. The ZQ0 and ZQ1 balls on the LPDDR4 device should be connected through 240Ω, 1% resistors to the LPDDR4 VDD2 rail.	-
	3. Place a 10 kΩ, 5% resistor to ground on the DRAM reset signal.	This will ensure adherence to the JEDEC specification until the control is configured and starts driving the DDR.
	4. The ODT_CA balls on the LPDDR4 device should be connected directly to the LPDDR4 VDD2 rail.	LPDDR4 ODT on the i.MX 8M Nano is command-based, making processor ODT_CA output balls unnecessary.
	5. The architecture for each chip inside the DRAM package must be x 16.	The processor does not support byte mode specified in JESD209-4B.
	6. The processor ball MTEST (ball N2), should be left unconnected.	These are observability ports for manufacturing and are not used otherwise.
	7. The VREF pin on the processor (ball P1) can be left unconnected.	The VREF signal for LPDDR4 is generated internally by the processor.
	8. It is strongly suggested to use LPDDR4 if lower power consumption is required since DLL-off mode is not supported.	The LPDDR4 can operate at low frequency without DLL-off mode.
	9. VDD_DRAM should be always on during DDR retention mode. Otherwise the data in DRAM might be lost when exiting this mode.	See Errata e50381 for detailed information.

Table 4. **DDR4/DDR3L recommendations (i.MX 8M Nano)**

Check box	Recommendations	Explanation/Supplemental recommendations
	1. Connect the ZQ(DRAM_ZN) ball on the processor (ball P2) to individual 240 Ω, 1% resistors to GND.	This is a reference used during DRAM output buffer driver calibration.
	2. The ZQ ball on each DDR4/DDR3L device should be connected through individual 240 Ω, 1% resistors to GND.	-
	3. Place a 10 kΩ, 5% resistor to ground on the DRAM reset signal.	This will ensure adherence to the JEDEC specification until the control is configured and starts driving the DDR.
	4. The processor ball MTEST (ball N2), should be left unconnected.	These are observability ports for manufacturing and are not used otherwise.
	5. Using x16 bit board to test the x8 bit DDR feature, only the controller setting is different, the PHY should train as x16 bit device.	Using x8 bit setting for initial and train on a x16 bit board, it may cause some issues, such as: • BG issue • PDA(Per DRAM Accessibility) based train issue.

Table 4. DDR4/DDR3L recommendations (i.MX 8M Nano)

Check box	Recommendations	Explanation/Supplemental recommendations
	6. DLL-off mode isn't supported, which means DDR4/DDR3L can't run in low frequency such as 100MTS.	The power consumption for low power mode in DDR4/DDR3L system will be higher compared with LPDDR4 system.
	7. VDD_DRAM should be always on during DDR retention mode. Otherwise the data in DRAM might be lost when exiting this mode.	See Errata e50381 for detailed information.

Table 5. I^{2}C recommendations

Check box	Recommendations	Explanation/Supplemental recommendations
	1. Verify the target I^{2}C interface clock rates	The I^{2}C bus can only be operated as fast as the slowest peripheral on the bus. If faster operation is required, move the slow devices to another I^{2}C port.
	2. Verify that there are no I^{2}C address conflicts on any of the I^{2}C buses utilized	There are multiple I^{2}C ports available on chip, so if a conflict exists, move one of the conflicting devices to a different I^{2}C bus. If it is impossible, use a I^{2}C bus switch (NXP part number PCA9646).
	3. Do not place more than one set of pull-up resistors on the I^{2}C lines.	This could result in excessive loading and potential incorrect operation. Choose the pull-up value commensurate with the bus speed being used.
	4. Ensure that the VCC rail powering the i.MX 8M Nano I^{2}C interface balls matches the supply voltage used for the pull-up resistors and the slave I^{2}C devices.	Prevent device damage or incorrect operation due to voltage mismatch.

Table 6. JTAG recommendations

Check box	Recommendations	Explanation/Supplemental recommendations
	1. Do not use external pullup or pulldown resistors on JTAG_TDO.	JTAG_TDO is configured with an on-chip keeper circuit and the floating condition is actively eliminated.
	2. Follow the recommendations for external pull-up and pull-down resistors given in **Table 18**.	-
	3. JTAG_MOD should be connected to ground through a resistor.	-
	4. JTAG_TMS pin must be connected with a 50ohm series resistor near the component if used or fanout. Otherwise, floating if not fanout.	-

Table 7. Reset and ON/OFF recommendations

Check box	Recommendations	Explanation/Supplemental recommendations
	1. The POR_B input must be asserted at powered up and remain asserted until the last power rail for devices required for system boot are at their working voltage. This functionality is controlled by the PMIC on EVK.	POR_B is driven by the PMIC. If a reset button is used, it should be connected to the PWRON_B pin of the PMIC instead of directly connected to POR_B pin of the CPU. When POR_B is asserted (low) on the i.MX 8M Nano, the output PMIC_ON_REQ remains asserted (high).

Table 7. **Reset and ON/OFF recommendations**

Check box	Recommendations	Explanation/Supplemental recommendations
	2. For portable applications, the ONOFF pin may be connected to an ON/OFF SPST push-button switch to ground. An external pull-up resistor is required on this pin.	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 initiate a software-controllable power-down). The connection to GND for approximate 5 seconds or more causes a forced OFF.
	3. Connect GPIO1_IO02(WDOG_B, ball AG13) to external PMIC or reset IC to repower the system except SNVS is strongly recommended.	i.MX8M Nano can't be reset by internal reset source in idle mode, repower is preferred. Some peripherals like SD3.0, QSPI also need repower during system reset.
	4. GPIO1_IO02(WDOG_B, ball AG13) is used as Cold Reset. If the PMIC enable the WDOG_B reset by default, external pull up resistor (100Kohm) and WDOG timer buffer are needed to support boundary-scan mode.	During entering boundary scan mode, WDOG_B is always low. Without the WDOG timer buffer and external 100Kohm pull up, WDOG_B will repeatedly reset 8MNANOD4-EVK when entering boundary-scan mode. See section 5.5. Boundary scan operation for more details.

Table 8. **USB recommendations**

Check box	Recommendations	Explanation/Supplemental recommendations
	1. The USB1_TXRTUNE (ball E19) must be connected with a 200 Ω, 1% resistor to the ground.	-
	2. The USB1_VBUS (ball F22) must be connected with a 30K Ω, 1% series resistor to 5V VBUS power.	The USB1_VBUS pin must not connect directly to the 5V VBUS voltage. This pin must be isolated by an external resistor (30K Ω, 1%) so that the USB1_VBUS pin sees a lower voltage.
	3. Route all USB differential signals with 90 Ω differential impedance.	-
	4. ESD protection should be implemented at the connector pins. Choose a low capacitance device recommended for high-speed interfaces.	This will prevent potential damages to board components from ESD.

Table 9. **FlexSPI recommendations**

Check box	Recommendations	Explanation/Supplemental recommendations
	1. Read strobe(DQS) pad should be floated or with a 10-18pF cap load to compensate SIO/SCK pins load for high speed running, if the memory device doesn't provide DQS.	There are three modes for the internal sample clock for FlexSPI read data: • Dummy read strobe generated by FlexSPI controller and looped back internally (FlexSPIn_MCR0[RXCLKSRC] = 0x0), can only reach 66Mhz operation frequency; • Dummy read strobe generated by FlexSPI controller and looped back through the DQS pad(FlexSPIn_MCR0[RXCLKSRC] = 0x1), can reach 133Mhz operation frequency. In this mode, this pin can be floated or put some cap loads on board level to compensate SIO/SCK pins load; • Read strobe provided by memory device and input from DQS pad (FlexSPIn_MCR0[RXCLKSRC] = 0x3), can reach 133Mhz operation frequency.

Table 10. **Oscillator/Crystal recommendations**

Check box	Recommendations	Explanation/Supplemental recommendations
	1. Connect a 24 MHz crystal and a 510K Ω resistor between 24M_XTALI and 24M_XTALO (balls B27 and C26).	This crystal should have ESR not greater than 80 Ω, and be rated for a drive level of at least 180 µW. Follow the manufacturer's recommendation for loading capacitance. Use short traces between the crystal and the processor, with a ground plane under the crystal, load capacitors, and associated traces.
	2. Use the 32.768 kHz clock generated by the PMIC to drive the i.MX 8M Nano RTC_XTALI input (ball A26), and connect RTC_XTALO (ball B25) to VDD_SNVS_0P8.	The voltage level of this driving clock should not exceed the voltage of the NVCC_SNVS rail, or the damage/malfunction may occur. The RTC signal should not be driven if the NVCC_SNVS supply is OFF. It can lead to damage or malfunction. For RTC V_{IL} and V_{IH} voltage levels, see the latest i.MX 8M Nano datasheet available at www.nxp.com/i.MX8MNANO.

Table 11. **Temperature sensor recommendations**

Check box	Recommendations	Explanation/Supplemental recommendations
	1. The TSENSOR_RES_EXT (ball J24) must be connected with a 100K Ω, 1% resistor to GND.	This external resistor is used for temperature calibration, the wrong resistor value will result in erroneous behavior for the temperature sensor.

Table 12. **i.MX 8M Nano power/decoupling recommendations**

Check box	Recommendations	Explanation/Supplemental recommendations
	1. Comply with the power-up sequence guidelines as described in the datasheet to guarantee reliable operations 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)
	2. Maximum ripple voltage requirements	Common requirement for ripple noise peak-to-peak value should be less than 5% of the supply voltage nominal value.
	3. If using BD71850MWV PMIC to provide power, make sure all the regulators except BUCK5/LDO4/LDO5 have output L/C components properly connected, even if unused.	Leaving any regulator except BUCK5/LDO4/LDO5 output open will lead to malfunction of the PMIC.
	4. If using PCA9450B PMIC to provide power, make sure the voltage sensing pin of each BUCK is tied to VSYS if unused.	The voltage sensing pins are R_SNSP3_CFG, R_SNSPx and BUCKxFB in PCA9450B. Leaving any BUCK output open, the PMIC will enter fault shutdown.

Table 13. **Decoupling capacitors recommendations (i.MX 8M Nano)**

Checkbox	Supply	2.2 nF	0.22 µF	1 µF	4.7 µF	10 µF	Notes
	VDD_SOC, VDD_DRAM, VDD_GPU, VDD_DRAM_PLL_0P8	-	-	11	-	3	These 4 power rails are combined together on EVK
	NVCC_DRAM	-	-	6	-	2	-
	VDD_ARM	-	-	5	-	1	-
	VDD_SNVS_0P8	-	1	-	-	-	-
	NVCC_SNVS_1P8	-	-	1	-	-	-
	VDD_24M_XTAL_1P8	-	1	-	-	-	-
	VDD_DRAM_PLL_1P8	-	-	1	-	-	-
	PVCC_1P8	-	2	-	-	-	-
	VDD_ARM_PLL_1P8, VDD_ANA0_1P8, VDD_ANA1_1P8, VDD_USB_1P8, VDD_MIPI_1P8	-	4	-	-	1	-
	NVCC_SAI3, NVCC_SAI5, NVCC_ECSPI, VDD_USB_3P3	-	4	-	1	-	-
	NVCC_JTAG, NVCC_NAND, NVCC_SAI2, NVCC_GPIO1, NVCC_I2C, NVCC_UART, NVCC_SD1, NVCC_CLK	-	3	-	-	1	-
	NVCC_SD2	-	1	-	-	-	-
	NVCC_ENET	-	1	-	-	-	-

Table 13. **Decoupling capacitors recommendations (i.MX 8M Nano)**

Checkbox	Supply	2.2 nF	0.22 µF	1 µF	4.7 µF	10 µF	Notes
	VDD_ARM_PLL_0P8, VDD_ANA_0P8, VDD_USB_0P8,	-	1	-	1	-	-
	VDD_MIPI_1P2	-	1	-	-	-	-
	VDD_MIPI_0P8	-	1	-	-	-	-
	MIPI_VREG_CAP	1	-	-	-	-	Must connect a 2.2nF capacitor between MIPI_VREG_CAP (ball D15) and GND.
	Capacitor part number used on EVK: • 2.2 nF --- GRM033R71C222KA88D • 0.22 uF --- LMK063BJ224MP-F • 1 uF --- 02016D105MAT2A • 4.7 uF --- CL05A475KP5NRNC • 10 uF --- ZRB15XR60J106ME12D						

Table 14. **Bulk/Bypass capacitors recommendations (BD71850MWV PMIC)**

Checkbox	Supply	1 µF	4.7 µF	10 µF	22 µF	Notes
	BUCK1 (VDD_SOC&DRAM&GPU)	-	-	-	2	-
	BUCK2 (VDD_ARM)	-	-	-	2	-
	BUCK6 (VDD_3V3)	-	-	-	2	-
	BUCK7 (VDD_1V8)	-	-	-	2	-
	BUCK8 (NVCC_DRAM)	-	-	-	2	-
	LDO1 (NVCC_SNVS_1V8)	1	-	-	-	-
	LDO2 (VDD_SNVS_0V8)	1	-	-	-	-
	LDO3 (VDDA_1V8)	-	1	-	-	-
	LDO6 (VDD_PHY_1V2)	-	1	-	-	-
	MUXSW_VOUT (NVCC_SD2)	-	-	1	-	-
	Capacitor part number used on EVK: • 1 uF --- 02016D105MAT2A • 4.7 uF --- CL05A475KP5NRNC • 10 uF --- GRM188R61A106KE69D • 22 uF --- C1608X5R1A226M080AC					

Table 15. **Bulk/Bypass capacitors recommendations (PCA9450B PMIC)**

Checkbox	Supply	1 µF	2.2 µF	22 µF	Notes

Checkbox	Supply	1 µF	2.2 µF	22 µF	Notes
	BUCK1 (VDD_SOC&DRAM&GPU)	-	-	2	
	BUCK2 (VDD_ARM)	-	-	2	-
	BUCK4 (VDD_3V3)	-	-	1	-
	BUCK5 (VDD_1V8)	-	-	1	-
	BUCK6 (NVCC_DRAM)	-	-	1	-
	LDO1 (NVCC_SNVS_1V8)	1	-	-	-
	LDO2 (VDD_SNVS_0V8)	1	-	-	-
	LDO3 (VDDA_1V8)	-	1	-	-
	LDO4 (VDD_PHY_1V2)	1	-	-	-
	LDO5 (NVCC_SD2)	1	-	-	-
	Capacitor part number used on EVK: • 1 uF --- 02016D105MAT2A • 2.2 uF --- C1005X5R1A225K • 22 uF --- C1608X5R1A226M080AC				

Table 16. **PCB design recommendations**

Check box	Recommendations	Explanation/Supplemental recommendations
	1. High-speed signal traces have reference plane in adjacent layer and are impedance controlled.	Controlled impedance is the key factor to have good signal integrity. Note that the reference plane can only be GND or the signal's own I/O power. Do not use other nets as reference. For DRAM, only GND reference is accepted for maintaining impedance. The power plane reference can't be as the sole impedance return.
	2. High-speed signal traces never cross gap or slot in reference plane.	Crossing gap in reference plane will cause reflection and increase crosstalk.
	3. Place at least one GND stitching via within 50 mils of signal via when switching reference planes.	GND stitching via can help keep impedance continuous and reduce via crosstalk.
	4. Appropriate delay matching is done for parallel bus.	Signals within a bus should have delay time matched to maintain timing margin.
	5. The true and complementary signal of a differential pair must have delay matched to within 1ps.	The true and complementary signal within a differential pair should have delay time tightly matched.
	6. DDR interface passed SI simulation. Alternatively, directly copy the EVK DDR layout design.	Generally, SI simulation should be performed for DDR interface that runs at 3200 MT/s to ensure stable working. If this is not feasible, just copy the EVK DDR layout design as well as the board stack-up.

Table 16. **PCB design recommendations**

Check box	Recommendations	Explanation/Supplemental recommendations
	7. Place test point on key signals to ease debugging. When placing test point on high-speed signal traces, make sure its diameter is no more than 20mil and the test point be directly placed on the trace with no stub.	Test points can bring excessive capacitance and should be carefully handled on high-speed signal traces.
	8. Decoupling capacitors are placed as close to IC power pins and GND pins as possible.	Tight routing to both power and ground is needed to provide optimum decoupling effectiveness.

2.2. JTAG signal termination

Table 18 is a JTAG termination chart showing what terminations should be placed on PCB designs.

Table 17. **Recommended JTAG board terminations**

JTAG signal	I/O type	External termination	Comments
JTAG_TCK	Input	10 kΩ pull-down	-
JTAG_TMS	Input	50ohm serial resistor	Internal pulled up to NVCC_JTAG, connected with a 50ohm serial resistor.
JTAG_TDI	Input	None	Internal pulled up to NVCC_JTAG, no external termination required
JTAG_TDO	3-state output	None	-

2.3. Signal termination for Boundary-scan

Table 18 is a signal termination chart showing what terminations should be placed on board designs to support Boundary-scan.

Table 18. **Recommended board terminations for Boundary-scan**

JTAG signal	I/O type	External termination	Comments
BOOT_MODE0	Input	Pull up	BOOT_MODE[3:0] must be at 1111 before entering Boundary-scan mode.
BOOT_MODE1	Input	Pull up	
BOOT_MODE2	Input	Pull up	
BOOT_MODE3	Input	Pull up	
POR_B	Input	Pull up	
RTC_RESET_B	Input	Pull up	

3. i.MX 8M Nano layout/routing recommendations

3.1. Introduction

This chapter describes how to assist design engineers with the layout of an i.MX 8M Nano-based system.

3.2. Basic design recommendations

When using the Allegro design tool, the schematic symbol & PCB footprint created by NXP is recommended. When not using the Allegro tool, use the Allegro footprint export feature (supported by many tools). If the export is not possible, create the footprint per the package dimensions outlined in the product data sheet.

Native Allegro layout and gerber files are available on http://www.nxp.com/imx8mnanoevk.

3.2.1. Placing decoupling capacitors

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

The 0201 or 0402 decoupling and 0603 or larger 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 required by the processor. See the i.MX 8M Nano EVK layouts for examples of the desired decoupling capacitor placement.

The following list describes how to choose correct decoupling scheme:

- Place the largest capacitance in the smallest package that budget and manufacturing can support.
- For high-speed bypassing, select the required capacitance with the smallest package (for example, 0.1 µF, 0.22 µF, 1.0 µF, or even 2.2 µF in a 0201 package size).
- 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 ball of the associated package from the schematic.
- A preferred BGA power decoupling design is available on the EVK board design available on http://www.nxp.com/imx8mnanoevk. Customers should use the NXP design strategy for power and decoupling.

3.3. Stack-up and manufacturing recommendations

3.3.1. Stack-up recommendation (i.MX 8M Nano)

Due to the number of balls on the i.MX 8M Nano processor in the 14 mm x 14 mm package, a minimum 6-layer PCB stack-up is recommended. For the 6-layers on the PCB, a sufficient number of layers need to be dedicated to power on routing to meet the IR drop target of 2% for the i.MX 8M Nano CPU power rails.

The constraints for the trace width will depend on such factors as the board stack-up and associated dielectric and copper thickness, required impedance, and required current (for power traces). The stack-up also determines the constraints for routing and spacing. Consider the following requirements when designing the stack-up and selecting board material:

- Board stack-up is critical for high-speed signal quality.
- Preplanning impedance of critical traces is required.
- High-speed signals must have reference planes on adjacent layers to minimize cross-talk.
- PCB material: the material used on EVK is TU768.

3.3.2. Manufacturing recommendation (i.MX 8M Nano)

Since the i.MX 8M Nano processor uses 0.5mm-pitch BGA package, the PCB technology must meet below requirement to fully fanout all the signals of the processor using PTH(plated through holes).

- Minimum trace width: 3.2mil
- Minimum trace to trace/pad spacing: 3.2mil
- Minimum via size: 8mil-diameter hole, 16mil-diameter pad
- Minimum via pad to pad spacing: 4mil

Figure 1 shows the reference routing of the i.MX 8M Nano, PTH is ok for the fanout, HDI is not needed..

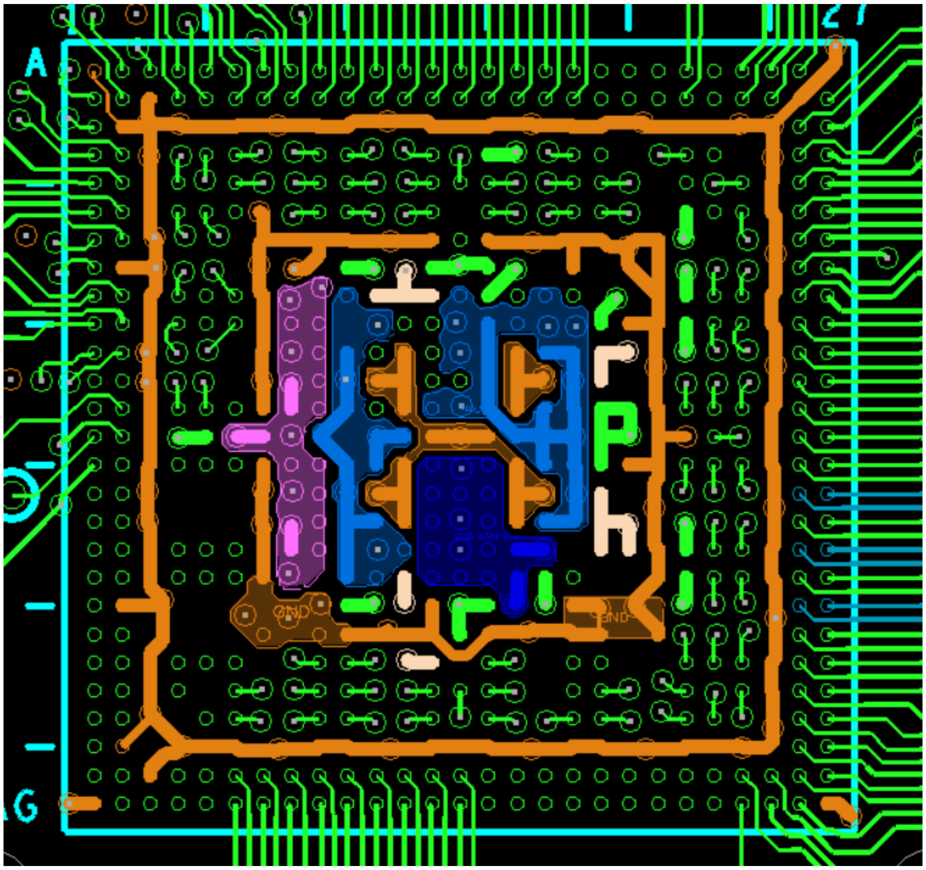


Figure 1. i.MX 8M Nano fanout routing on EVK

3.3.3. EVK PCB stack-up (i.MX 8M Nano)

Table 19 and Table 20 show stack-up of the EVK. The CPU board use 6-layer stack-up and the BB board use 8-layer stack-up.

Table 19. **8MNANOLPD4-CPU Board stack up information**

Layer	Description	Coppoer (Oz.)	Dielectric thickness (mil)
1	Signal	0.5+Plating	
	Dielectric		2.76 mil
2	GND	1	
	Dielectric		2.95 mil
3	Signal	1	
	Dielectric		25.28 mil
4	Power	1	
	Dielectric		2.95 mil
5	Power	1	
	Dielectric		2.76 mil
6	Signal	0.5+Plating	
Total thickness:	47.24(4.72/-4.72) mil		1.2(+0.12/-0.12) MM
Material:	TU768		TU768

Table 20. **8MMINI-BB Board stack up information**

Layer	Description	Coppoer (Oz.)	Dielectric thickness (mil)
1	Signal	0.5+Plating	
	Dielectric		2.717 mil
2	GND	1	
	Dielectric		4.33 mil
3	Signal	1	
	Dielectric		11.085 mil
4	Power	1	
	Dielectric		14.170 mil
5	Power	1	
	Dielectric		11.415 mil
6	Signal	1	
	Dielectric		4.33 mil
7	GND	1	
	Dielectric		2.717 mil
8	Signal	0.5+Plating	
Total thickness:	62.992(6.299/-6.299) mil		1.6(+0.16/-0.16) MM
Material:	TU768		TU768

3.4. DDR design recommendations

3.4.1. DDR connection information

The i.MX 8M Nano processor can be used with LPDDR4, DDR4 or DDR3L memory. Since these memory types have different I/O signals, there are 38 generically-named functional balls, depending on the type of memory used. See Table 21 for the connectivity of these generic balls for DDR3L, LPDDR4 and DDR4. The schematic symbol created by NXP already replaced these generic names with DDR function.

Table 21. **DDR3L/LPDDR4/DDR4 connectivity**

Ball name	Ball #	LPDDR4 function	DDR4 function	DDR3L function
DRAM_AC00	F4	CKE0_A	CKE0	CKE0
DRAM_AC01	F5	CKE1_A	CKE1	CKE1
DRAM_AC02	K4	CS0_A	CS0_n	CS0#
DRAM_AC03	J4	CS1_A	C0	-
DRAM_AC04	L2	CK_t_A	BG0	BA2
DRAM_AC05	L1	CK_c_A	BG1	A14
DRAM_AC06	F6	-	ACT_n	A15
DRAM_AC07	J5	-	A9	A9
DRAM_AC08	J6	CA0_A	A12	A12/BC#
DRAM_AC09	K6	CA1_A	A11	A11
DRAM_AC10	E4	CA2_A	A7	A7
DRAM_AC11	D5	CA3_A	A8	A8
DRAM_AC12	N4	CA4_A	A6	A6
DRAM_AC13	N5	CA5_A	A5	A5
DRAM_AC14	K5	-	A4	A4
DRAM_AC15	N6	-	A3	A3
DRAM_AC16	M1	-	CK_t_A	CK_A
DRAM_AC17	M2	-	CK_c_A	CK#_A
DRAM_AC19	N2	MTEST	MTEST	MTEST
DRAM_AC20	AB4	CKE0_B	CK_t_B	CK_B
DRAM_AC21	AB5	CKE1_B	CK_c_B	CK#_B
DRAM_AC22	W4	CS1_B	-	-
DRAM_AC23	V4	CS0_B	-	-
DRAM_AC24	U2	CK_t_B	A2	A2
DRAM_AC25	U1	CK_c_B	A1	A1
DRAM_AC26	N1	-	BA1	BA1
DRAM_AC27	R6	-	PARITY	-
DRAM_AC28	W6	CA0_B	A13	A13
DRAM_AC29	V6	CA1_B	BA0	BA0
DRAM_AC30	AC4	CA2_B	A10 / AP	A10 / AP
DRAM_AC31	AD5	CA3_B	A0	A0
DRAM_AC32	R4	CA4_B	C2	-
DRAM_AC33	R5	CA5_B	CAS_n / A15	CAS#
DRAM_AC34	T1	-	WE_n / A14	WE#
DRAM_AC35	T2	-	RAS_n / A16	RAS#
DRAM_AC36	V5	-	ODT0	ODT0

Table 21. **DDR3L/LPDDR4/DDR4 connectivity**

Ball name	Ball #	LPDDR4 function	DDR4 function	DDR3L function
DRAM_AC37	W5	-	ODT1	ODT1
DRAM_AC38	AB6	-	CS1_n	CS1#

3.4.2. LPDDR4-3200 design recommendations

TBD.

3.4.2.1. i.MX 8M Nano LPDDR4-3200 routing recommendations

TBD

3.4.2.2. LPDDR4-3200 routing example (i.MX 8M Nano)

TBD.

3.4.3. i.MX 8M Nano DDR4-2400 design recommendations

The following list provides some generic guidelines for implementing an i.MX 8M Nano design using DDR4.

1. It is expected that the layout engineer and design team already has experience and training with DDR designs at speeds of 1.2 GHz / 2400 MT/s.
2. DQ/DMI signal traces must refer to solid GND plane only. Addr/Cmd/Ctrl signal traces can refer to GND plane only or GND+VDDQ plane (when routed as strip line). Referring to VDDQ plane only is not allowed.
3. Keep edge-to-edge spacing of high-speed signal traces no less than 1.5 times the trace width to minimize trace crosstalk.
4. At a speed of 2400 MT/s, signal vias can be a significant source of crosstalk. If not properly designed, it can introduce crosstalk larger than that from the trace. To minimize via crosstalk, make sure that the number of vias on each point-to-point signal is no more than 2. Place at least one ground stitching via within 50 mils of signal via when switching reference planes to provide continuous return path and reduce crosstalk. If it is not possible to place enough ground stitching vias due to space limitation, try to make the length that the signal actually travels on the via as short as possible, as illustrated in Figure 2.

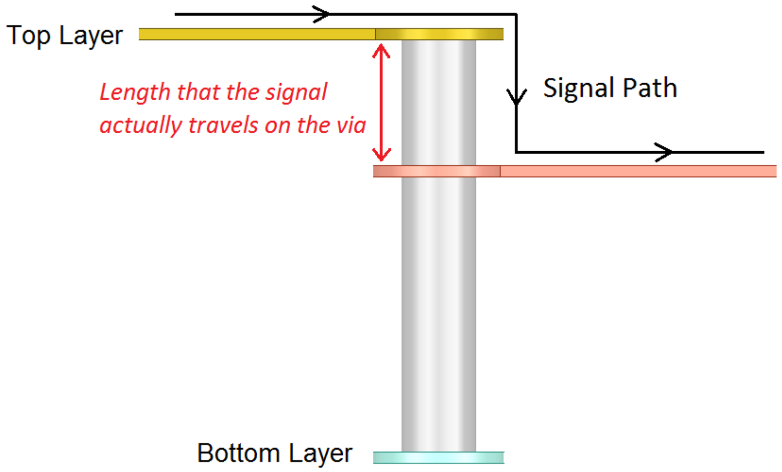


Figure 2. Length that the signal actually travels on the Via

5. CLK and DQS signal can be routed on different layers with DQ/CA signals to ease routing. When doing this, keep no less than 5 times trace width spacing from other signals.
6. Use time delay instead of length when performing the delay matching. The delay matching includes the PCB trace delay and the IC package delay. Incorporate the package pin delay into the CAD tool's constraint manager.
7. Include the delay of vias when performing delay matching. This can be realized in Allegro tool by enabling the **Z Axis Delay** in **Setup** -> **Constraints** -> **Modes**.
8. Byte swapping between upper/lower byte lane is allowed. Bit swapping within each slice/byte lane is allowed.
9. Bit swapping of Cmd/Addr/Ctrl signals is **NOT** allowed.
10. In general, the DDR4 DRAM should be placed 200 mils from the i.MX 8M Nano.
11. Enable the Data Bus Inversion (DBI) feature. It can help reduce both power consumption and power noise.

3.4.3.1. i.MX 8M Nano DDR4-2400 routing recommendations

DDR4-2400 needs to be routed with signal fly times matched shown in Table 22. The delay of the via transitions needs to be included in the overall calculation. This can be realized in Allegro tool by enabling the **Z Axis Delay** in **Setup** -> **Constraints** -> **Modes**.

An example of the delay match calculation has been shown for the i.MX 8M Nano DDR4 EVK board design in Table 23 and Table 24. This analysis is done for the DDR4-2400 implementation using the i.MX 8M Nano. In Table 23 and Table 24, the **PCB Delay** column is obtained directly from the Allegro PCB file, and the **Pkg Delay** column is the package delay obtained from Table 25.

Table 22. **i.MX 8M Nano DDR4-2400 routing recommendations**

DDR4-2400				
DDR4/DDR3L signal	**Group**	**PCB + package prop delay**		**Considerations**
		Min.	**Max.**	
CK_t/CK_c	Clock	Short as possible	500ps[1]	Match the true/complement signals within 1 ps.
A[13:0]/BA[1:0]/BG0	Address/ Command/ Control	CK_t - 10 ps	CK_t + 10 ps	
CS/RAS/WE/CAS				
CKE/ODT				
DQS0_t/DQS0_c	Byte 0 - DQS	Short as possible	CK_t + 1.0 * ^{t}CK	Match the true/complement signals of DQS within 1 ps.
DM0	Byte 0 - Data	DQS0_t -10 ps	DQS0_t +10 ps	
DQ[7:0]				
DQS1_t/DQS1_c	Byte 1 - DQS	Short as possible	CK_t + 1.0 * ^{t}CK	
DM1	Byte 1 - Data	DQS1_t -10 ps	DQS1_t + 10 ps	
DQ[15:8]				

1. For 16-bit single chip routed with point to point, CK_t/CK_c should be short as possible and much less than 500ps. 230ps is recommended.

Table 23. **DDR4 delay matching example (Addr/Cmd/Ctrl/CK signals)**

Net name	**PCB delay (ps)**	**Pkg delay (ps)**	**Comment**
DRAM_A0 U1.AD5:U2.P3	167.2	56.5	Routed on top layer, no via
	223.7		Total Net Delay
DRAM_A1 U1.U1:U2.P7	183.2	42.2	Routed on top layer, no via
	225.4		Total Net Delay
DRAM_A2 U1.U2:U2.R3	181.1	41.8	Vias are L1-> L3->L1
	222.9		Total Net Delay
DRAM_A3 U1.N6:U2.N7	202.7	22.4	Vias are L1-> L3->L1
	225.1		Total Net Delay
DRAM_A4 U1.K5:U2.N3	179.9	43.1	Vias are L1-> L3->L1
	223.0		Total Net Delay
DRAM_A5 U1.N5:U2.P8	191.9	32.0	Vias are L1-> L3->L1
	223.9		Total Net Delay
DRAM_A6 U1.N4:U2.P2	189.1	33.8	Vias are L1-> L3->L1
	222.9		Total Net Delay
DRAM_A7 U1.E4:U2.R8	167.2	54.8	Vias are L1-> L6->L1
	222.0		Total Net Delay
DRAM_A8 U1.D5:U2.R2	165.9	59.7	Vias are L1-> L6->L1
	225.6		Total Net Delay
DRAM_A9 U1.J5:U2.R7	189.4	35.8	Vias are L1-> L6->L1
	225.2		Total Net Delay
DRAM_A10 U1.AC4:U2.M3	165.6	59.5	Routed on top layer, no via
	225.1		Total Net Delay
DRAM_A11 U1.K6:U2.T2	193.7	29.1	Vias are L1-> L3->L1
	222.8		Total Net Delay
DRAM_A12 U1.J6:U2.M7	185.8	39.6	Vias are L1-> L6->L1
	225.4		Total Net Delay
DRAM_A13 U1.W6:U2.T8	193.5	31.9	Routed on top layer, no via
	225.4		Total Net Delay
DRAM_BA0 U1.V6:U2.N2	188.4	34.4	Vias are L1-> L3->L1
	222.8		Total Net Delay

DRAM_BA1 U1.N1:U2.N8	168.3	53.4	Vias are L1-> L3>L1
	221.7		Total Net Delay
DRAM_BG0 U1.L2:U2.M2	183.8	41.1	Vias are L1-> L3>L1
	224.9		Total Net Delay
DRAM_CK_C U1.M2:U2.K8	183.7	39.4	Vias are L1-> L6>L1
	223.1		Total Net Delay
DRAM_CK_T U1.M1:U2.K7	184.5	39.1	Vias are L1-> L6>L1
	223.6		Total Net Delay
DRAM_NACT U1.F6:U2.L3	176.2	45.7	Vias are L1-> L6-> L1
	221.9		Total Net Delay
DRAM_NALERT U1.R2:U2.P9	186.3	36.0	Vias are L1-> L3>L1
	222.3		Total Net Delay
DRAM_NCAS(A15) U1.R5:U2.M8	183.1	38.6	Vias are L1-> L6-> L1
	221.7		Total Net Delay
DRAM_CKE U1.F4:U2.K2	171.6	51.2	Vias are L1-> L3->L1
	222.8		Total Net Delay
DRAM_NCS U1.K4:U2.L7	186.9	35.9	Vias are L1-> L3->L1
	222.8		Total Net Delay
DRAM_NRAS U1.T2:U2.L8	171.3	51.2	Routed on top layer, no via
	222.5		Total Net Delay
DRAM_NRESET U1.R1:U2.P1	186.5	38.1	Vias are L1-> L6>L1
	224.6		Total Net Delay
DRAM_NWE(A14) U1.T1:U2.L2	179.0	43.1	Vias are L1-> L6>L1
	222.1		Total Net Delay
DRAM_ODT U1.V5:U2.K3	196.6	26.5	Vias are L1-> L3->L1
	223.1		Total Net Delay
DRAM_PARITY U1.R6:U2.T3	193.3	29.0	Vias are L1-> L3->L1
	222.3		Total Net Delay

Table 24. **DDR4 delay matching example (Byte0/Byte1 signals)**

Net Name	PCB Delay (ps)	Pkg Delay (ps)	Comment
DRAM_DMI0	137.8	57.2	Vias are L1-> L3->L1
	195.0		Total Net Delay
DRAM_DQS0_N	136.6	58.9	Vias are L1-> L6->L1
	195.5		Total Net Delay
DRAM_DQS0_P	135.7	59.0	Vias are L1-> L6->L1
	194.7		Total Net Delay
DRAM_DQ00	148.5	47.2	Vias are L1-> L3->L1
	195.7		Total Net Delay
DRAM_DQ01	152.7	43.0	Vias are L1-> L3->L1
	195.7		Total Net Delay
DRAM_DQ02	140.6	54.6	Vias are L1-> L3->L1
	195.2		Total Net Delay
DRAM_DQ03	143.5	51.7	Vias are L1-> L3->L1
	195.2		Total Net Delay
DRAM_DQ04	135.6	59.9	Vias are L1-> L3->L1
	195.5		Total Net Delay
DRAM_DQ05	136.1	58.1	Vias are L1-> L3->L1
	194.2		Total Net Delay
DRAM_DQ06	130.9	64.6	Vias are L1-> L3->L1
	195.5		Total Net Delay
DRAM_DQ07	143.4	51.4	Vias are L1-> L3->L1
	194.8		Total Net Delay
DRAM_DMI1	106.3	58.6	Routed on top layer, no via
	164.9		Total Net Delay
DRAM_DQS1_N	115.9	47.2	Routed on top layer, no via

	163.1		Total Net Delay
DRAM_DQS1_P	115.4	48.6	Routed on top layer, no via
	164.0		Total Net Delay
DRAM_DQ08	119.7	45.0	Routed on top layer, no via
	164.7		Total Net Delay
DRAM_DQ09	114.9	50.1	Routed on top layer, no via
	165.0		Total Net Delay
DRAM_DQ10	117.7	46.2	Routed on top layer, no via
	163.9		Total Net Delay
DRAM_DQ11	116.1	47.2	Routed on top layer, no via
	163.3		Total Net Delay
DRAM_DQ12	123.4	40.3	Routed on top layer, no via
	163.7		Total Net Delay
DRAM_DQ13	115.5	48.3	Routed on top layer, no via
	163.8		Total Net Delay
DRAM_DQ14	106.5	58.4	Routed on top layer, no via
	164.9		Total Net Delay
DRAM_DQ15	112.3	52.4	Routed on top layer, no via
	164.7		Total Net Delay

3.4.3.2. DDR4-2400 Routing example (i.MX 8M Nano)

Figure 3 to Figure 5 show the placement and routing of the DDR4 signals on the i.MX8M Nano DDR4 EVK board.

The CK and DQS0 signals are routed on the bottom layer to save routing space of other signal layers. Data byte lane 1 and DQS1 are routed on the top layer and data byte lane 0 are routed on layer 3, which can minimize via crosstalk to achieve enough timing margin for the 2400 MT/s high-speed signals. This is also to make the signal actually travel on the via as short as possible to minimize via crosstalk. Addr/Cmd/Ctrl signals are routed on the top layer, layer 3 and bottom layer since they have more tolerance for crosstalk.

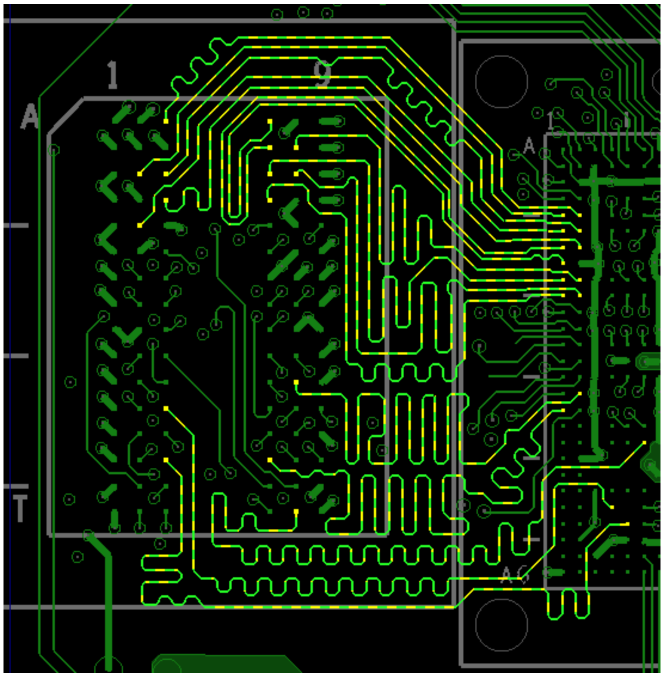


Figure 3. i.MX 8M Nano DDR4 EVK board DDR4 routing (Top Layer)

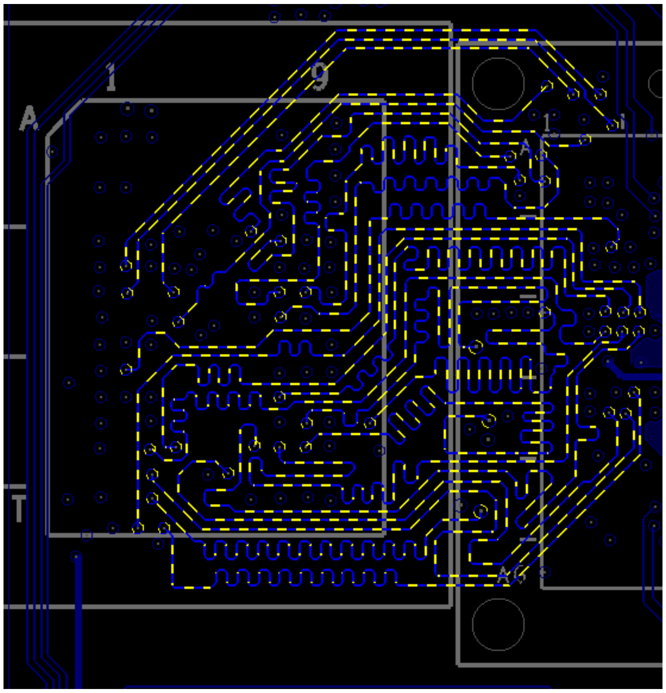


Figure 4. i.MX 8M Nano DDR4 EVK board DDR4 routing (Layer 3)

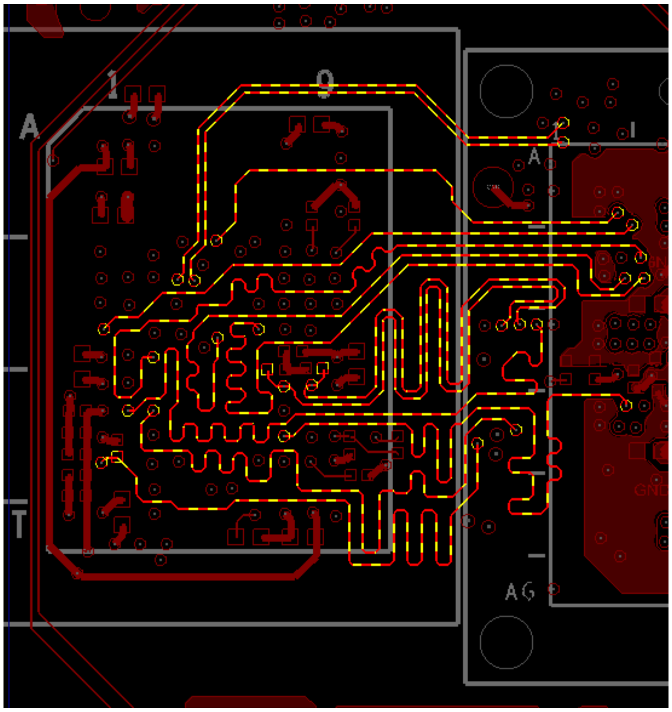


Figure 5. i.MX 8M Nano DDR4 EVK board DDR4 routing (Bottom Layer)

3.4.4. i.MX 8M Nano DDR3L-1600 design recommendations

TBD.

3.4.4.1. i.MX 8M Nano DDR3L-1600 routing recommendations

TBD.

3.4.4.2. DDR3L-1600 Routing example (i.MX 8M Nano)

TBD.

3.4.5. i.MX 8M Nano DDR SI simulation guide

The simulation architecture includes the DDR controller (i.e., the i.MX 8M Nano processor), the PCB and the DRAM device. The IBIS model for the i.MX 8M Nano processor is available from NXP. The DRAM device IBIS model must be obtained from the memory vendor.

This section describes how to check SI performance of the layout for an DDR design using the i.MX 8M Nano.

- Firstly, perform S-parameter extraction:
  - It requires a 2.5D full-wave extraction tool, such as PowerSI from Cadence.
  - Set the extraction bandwidth to 20 GHz.

    - Port reference impedance: 50 Ω for signal ports, and 0.1 Ω for power ports.
    - Coupled mode: Set the rise time to 20 ps and coupling coefficient to 1%.
- Secondly, perform time domain simulation:
    - Stimulus pattern: 500-bit random code and different pattern for each signal within the same byte.
    - Ideal power.
    - Probe at the die.
    - Simulation at slow corner (worst case).
    - Eye waveform triggered by aligning with the timing reference (DQS/CLK).

When the simulation is done, find the simulated worst eye width and compare with following requirements to see if it can pass:

- For LPDDR4-3200
    - DQ Write: Eye width @V_{REF} ±70mV should be over 227ps.
    - DQ Read: Eye width @V_{REF} ±70mV should be over 195ps.
    - Cmd/Addr/Ctrl: Eye width @V_{REF} ±77.5mV should be over 517ps.
- For DDR4-2400
    - DQ Write: Eye width @V_{REF} ±65mV should be over 355ps.
    - DQ Read: Eye width @V_{REF} ±70mV should be over 343ps.
    - Cmd/Addr/Ctrl: Eye width at threshold should be over 735ps.
- For DDR3L-1600
    - DQ Write: Eye width at threshold should be over TBD.
    - DQ Read: Eye width @641.5 ±70mV should be over TBD.
    - Cmd/Addr/Ctrl: Eye width at threshold should be over TBD.

Figure 6 shows an example of simulated eye width of DDR4-2400 DQ write.

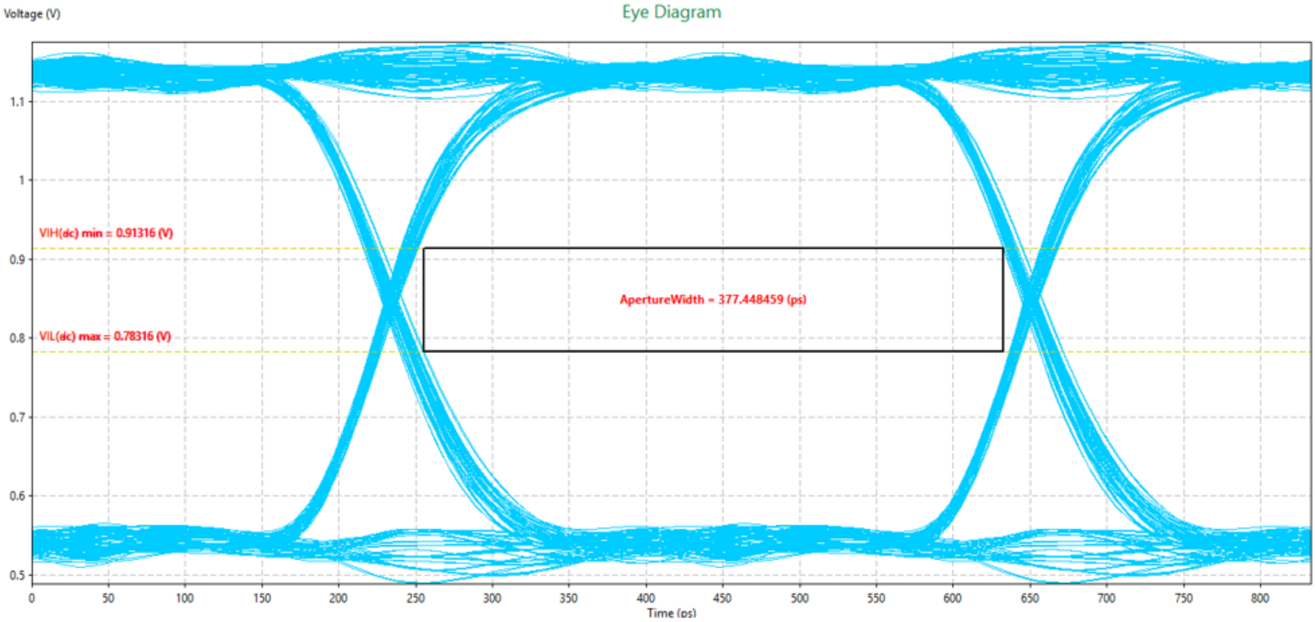


Figure 6. Example of simulated eye width

3.4.6. i.MX 8M Nano DDR package delay

When performing the required delay matching for LPDDR4/DDR4 routing, the bond wires within the i.MX 8M Nano package need to be accounted for and included in the match calculation. Table 25 lists the propagation/fly time from the die I/O to the package ball.

Table 25. **i.MX 8M Nano DDR package trace delays**

Ball Name	Delay (ps)	Ball name	Delay (ps)
DRAM_AC00	51.2	DRAM_AC32	34.3
DRAM_AC01	39.9	DRAM_AC33	38.6
DRAM_AC02	35.9	DRAM_AC34	43.1
DRAM_AC03	44.2	DRAM_AC35	51.2
DRAM_AC04	41.1	DRAM_AC36	26.5
DRAM_AC05	41.2	DRAM_AC37	35.1
DRAM_AC06	45.7	DRAM_AC38	42.1
DRAM_AC07	35.8	DRAM_ALERT_N	36.0
DRAM_AC08	39.6	DRAM_RESET_N	38.1
DRAM_AC09	29.1	DRAM_DM0	57.2
DRAM_AC10	54.8	DRAM_DM1	58.6
DRAM_AC11	59.7	DRAM_DQS0_N	58.9
DRAM_AC12	33.8	DRAM_DQS0_P	59.0
DRAM_AC13	32.0	DRAM_DQS1_N	47.2
DRAM_AC14	43.1	DRAM_DQS1_P	48.6
DRAM_AC15	22.4	DRAM_DQ00	47.2
DRAM_AC16	39.1	DRAM_DQ01	43.0
DRAM_AC17	39.4	DRAM_DQ02	54.6
DRAM_AC19	43.4	DRAM_DQ03	51.7
DRAM_AC20	51.6	DRAM_DQ04	59.9
DRAM_AC21	51.6	DRAM_DQ05	58.1

Ball Name	Delay (ps)	Ball name	Delay (ps)
DRAM_AC22	47.7	DRAM_DQ06	64.6
DRAM_AC23	40.0	DRAM_DQ07	51.4
DRAM_AC24	41.8	DRAM_DQ08	45.0
DRAM_AC25	42.2	DRAM_DQ09	50.1
DRAM_AC26	53.4	DRAM_DQ10	46.2
DRAM_AC27	29.0	DRAM_DQ11	47.2
DRAM_AC28	31.9	DRAM_DQ12	40.3
DRAM_AC29	34.4	DRAM_DQ13	48.8
DRAM_AC30	59.5	DRAM_DQ14	58.4
DRAM_AC31	56.5	DRAM_DQ15	52.4

3.4.7. High-speed routing recommendations

The following lists the routing traces for high-speed signals. The propagation delay and the impedance control should match to ensure the correct communication with the devices.

- High-speed signals (DDR, RGMII, MIPI, etc.) must not cross gaps in the reference plane.
- Avoid creating slots, voids, and splits in reference planes. Review via placements to ensure that they do not inadvertently create splits/voids (i.e., space vias out to eliminate this possibility).
- Ensure that ground stitching vias are present within 50 mils from signal layer transition vias on high-speed signals when transitioning between different reference ground planes.
- A solid GND plane must be directly under crystals, associated to components and traces.
- Clocks or strobes that are on the same layer need at least 2.5x height from reference plane spacing from adjacent traces to reduce crosstalk.
- All synchronous interfaces should have appropriate bus delay matching.
- The true and complementary signal of a differential pair must have delay matched to within 1ps.

3.4.8. Reset architecture/routing

A reset button may be connected to PWRON_B pin of the PMIC (BD71850MWV) or PMIC_RST_B pin of the PMIC PCA9450B for development purposes. This allows all voltages to be put to their initial default power-on state when depressing the reset button.

Pressing the reset button causes the PMIC to trigger a cold reset event. This will cause all the power supplies except for the SNVS domain to be OFF. During this time, the POR_B driven by the PMIC will also keep asserted (low). This state will keep several hundred milliseconds to provide enough time for the power supplies to be completely powered down, and then the power supplies will start to ramp up again in defined sequence. When all the power supplies have reached their operating voltages, POR_B will be de-asserted, and the CPU may begin booting from reset.

3.5. Trace impedance recommendations

Table 26 is a reference when you are updating or creating constraints in the PCB design tool to set up the impedances/trace widths.

Table 26. **Trace impedance recommendations**

Signal group	Impedance	PCB manufacturer tolerance (+/-)
All single-ended signals, unless specified	50 Ω Single-ended	10%
DDR DQS/CLK signals	85 Ω Differential	10%
USB differential signals	90 Ω Differential	10%
Differential signals, including Ethernet, MIPI (CSI and DSI)	100 Ω Differential	10%

3.6. Power connectivity/routing

Delivering clean, reliable power to the i.MX 8M Nano internal power rails is critical to a successful board design. The PCB PDN should be designed to accommodate the maximum output current from each SMPS into the i.MX 8M Nano supply balls. Table 27 lists the design goals for each high-current i.MX 8M Nano power rail. Combining VDD_SOC with VDD_GPU and VDD_DRAM together, the max current will not exceed the PMIC BD71850 or PCA9450B regulator maximum as all three can never operate at the Maximum simultaneously.

Table 27. **i.MX 8M Nano maximum current design levels**

Supply input	i.MX 8M Nano Max current (mA)
VDD_ARM	2200
VDD_SOC	1000
VDD_GPU	800
VDD_DRAM	800
NVCC_DRAM	1000

3.6.1. i.MX 8M Nano power distribution block diagram

There are companion PMICs that provide a low-cost and efficient solution for powering the i.MX 8M Nano processor, for example, ROHM BD71850.

The default output of BUCK8 is 1.1 V, which is for LPDDR4 NVCC_DRAM. You can modify the voltage to 1.2 V for DDR4, and 1.35 V for DDR3L by programming PMIC in SPL code before the U-Boot or kernel image is loaded onto DDR. This function has been fully verified, so you can use the **ONE** PMIC part for all kinds of DDR memories.

Figure 7 shows a block diagram of the power tree of the NXP i.MX 8M Nano EVK board. It uses a single BD71850MWV PMIC to power ON rails of the processor.

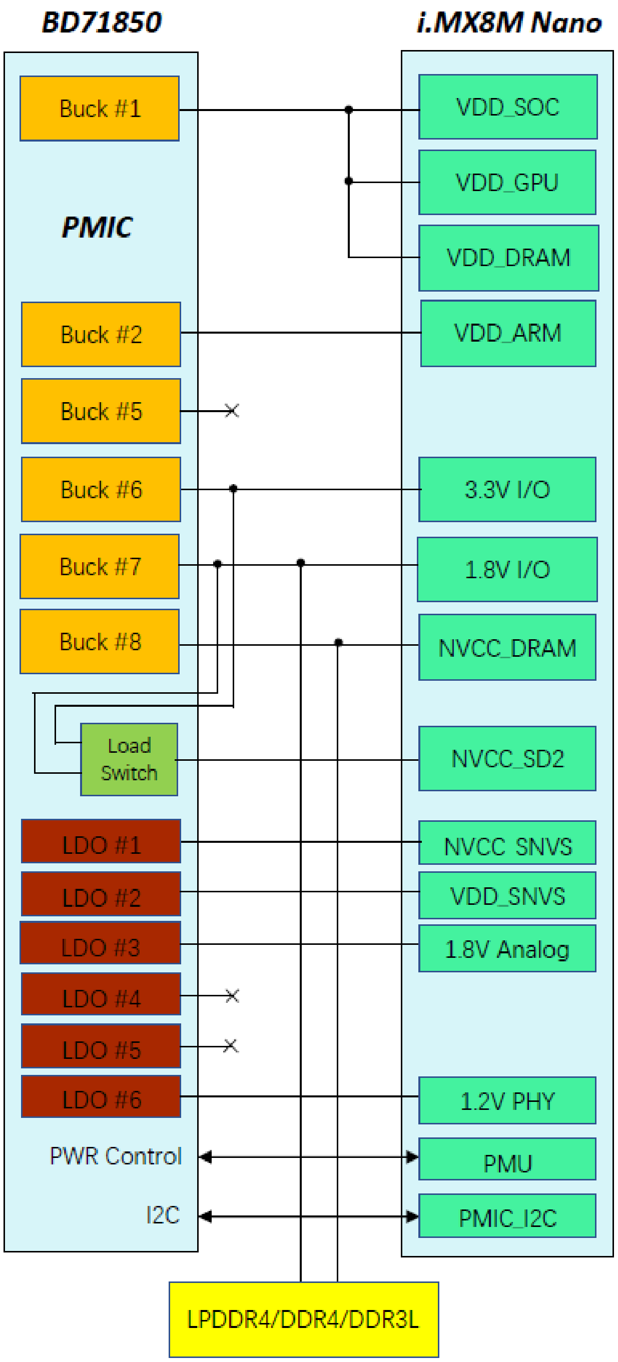


Figure 7. i.MX 8M Nano development platform power distribution block diagram

3.6.2. Power routing/distribution requirements

The designing for a good Power Delivery Network (PDN) is complicated. It includes:

1. Choose a good PCB stack-up (adequate Cu thicknesses, and layer assignments/utilization).

2. Optimize the placement and routing of the PDN. This includes good placement of the decoupling capacitors and connecting them to the power ground planes with as short and wide a trace as possible (as the increased inductance of a longer etch will degrade the effectivity of the capacitor). Use the number/placement of capacitors on the NXP development platforms.
3. Optimize DC IR drop. This involves using very wide traces/plane fills to route high-current power nets and ensure an adequate number of vias on power net layer transitions. Neck down of fill areas should be minimized and current density minimized. The maximum DC IR drop on a board should be 2% (preferably 1%) of the voltage rail (i.e., on a 1.1V rail, the maximum voltage drop should be less than 0.022 V, preferably less than 0.011 V). See **Table 28** for the DC IR drop requirement.
4. AC impedance check – the target impedance at different frequencies should be below specified values. See **Table 29** for the impedance targets vs. frequency for specified power rail for the i.MX 8M Nano PCB design.

Table 28. **i.MX 8M Nano DC IR drop requirements**

Supply input	Nominal voltage (V)	Max current (mA)	IR drop target	Corresponding power path resistance requirement (mΩ)
VDD_ARM	0.85/0.95/1.0	2200	<2%	<7.7
VDD_SOC&DRAM&GPU	0.85/0.95	2000	<2%	<8.5
NVCC_DRAM	1.1	1000	<2%	<22

Table 29. **i.MX 8M Nano PDN target impedance**

Supply Input	< 20 MHz (mΩ)	20 - 100 MHz (mΩ)
VDD_ARM	33	154
VDD_SOC&DRAM&GPU	12	58
NVCC_DRAM	17	78

3.7. USB connectivity

The i.MX 8M Nano provides one complete USB2.0 interface and the following configurations (or any subset) are supported:

- Dedicated host or device using Type-A connector or Type-B connector;
- Dual role using Type-C connector.

To implement a USB Type-C interface (UFP, DFP, or DRP), external hardware must be added to manage the two configuration channel IOs (CC1 and CC2) as well as monitor the plug orientation.

See the NXP development platform schematic for an example USB Type-C implementation.

3.8. Unused input/output terminations

3.8.1. i.MX 8M Nano unused input/output guidance

For the i.MX 8M Nano, the I/Os and power rails of an unused function can be terminated to reduce overall board power. **Table 30** lists connectivity examples for unused power supply rails and **Table 31** list connectivity examples for unused signal contacts/interfaces.

Table 30. **i.MX 8M Nano unused power rail strapping recommendations**

Function	Ball name	Recommendation if unused
MIPI-CSI & MIPI-DSI	VDD_MIPI_1P8, VDD_MIPI_1P2, VDD_MIPI_0P8	Leave unconnected[1]
USB	VDD_USB_3P3, VDD_USB_1P8, VDD_USB_0P8	Leave unconnected[2]
GPU	VDD_GPU	Leave unconnected
Digital I/O supplies	NVCC_CLK, NVCC_ECSPI, NVDD_ENET, NVCC_GPIO1, NVCC_I2C, NVCC_JTAG, NVCC_NAND, NVCC_SAI2, NVCC_SAI3, NVCC_SAI5, NVCC_SD1, NVCC_SD2, NVCC_UART, NVCC_SNVS_1P8, PVCC0_1P8, PVCC1_1P8, PVCC2_1P8	All digital I/O supplies listed in this table must be powered under normal conditions whether the associated I/O pins are in use or not, and associated I/O pins need to enable pull in pad control register to limit any floating gate current.

1. These balls supply both MIPI-CSI and MIPI-DSI interfaces and must be connected/powered if either is used.
2. These balls supply the USB interface(USB1) and must be connected/powered if any USB port is used.

Table 31. **i.MX 8M Nano unused signal strapping recommendations**

Function	Ball name	Recommendation if unused
MIPI-CSI	MIPI_CSI_CLK_P/N, MIPI_CSI_Dx_P/N	Tie all signals to ground
MIPI-DSI	MIPI_DSI_CLK_P/N, MIPI_DSI_Dx_P/N, MIPI_VREG_CAP	Leave unconnected
USB1	USB1_VBUS, USB1_DN/DP, USB1_ID, USB1_TXRTUNE	Leave unconnected

4. Avoiding board bring-up problems

4.1. Introduction

This chapter describes how to avoid mistakes when bringing up a board for the first time. The recommendations below consist of basic techniques for detecting board issues and preventing/locating the three issues encountered: power, clocks, and reset.

4.2. Avoiding power pitfalls -Current

Excessive current can damage the board. Use a current-limiting laboratory supply set to the expected 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 there is. By monitoring the main supply current and controlling the current limit, any excessive current can be detected before permanent damage occurs.

Before the board test, you can ohm out the board power rails to the ground to verify that there are no short circuits. Then, you can power on the board and there will not be any damage to the board and/or components.

4.3. Avoiding power pitfalls -Voltage

To avoid incorrect voltage rails, create a basic table called a voltage report prior to board bring up/testing. The table helps to validate that all the supplies are reaching the expected levels.

To create a voltage report, list the following:

- 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

Determine the best measurement location for each power supply to avoid a large voltage drop (IR drop) on the board. The drop causes inaccurate voltage values. The following guidelines help produce the best voltage measurements:

- Measure closest to the load (in the case of the i.MX 8M Nano processor).
- Make two measurements: the first after initial board power-up and the second while running a heavy use-case that stresses the i.MX 8M Nano processor.

Ensure that the i.MX 8M Nano power supply meets the DC electrical specifications as listed in the chip-specific data sheet. See **Table 32** for a sample voltage report table.

NOTE

This report table is for the i.MX 8M Nano EVK board. Sample voltage reports for customer PCBs will be different from this, depending on the Processor and Power Management IC (PMIC) used and the assignment of the PMIC power resources.

Table 32. **Sample voltage report table**

Source	Net name	Expected (V)	Measured (V)	Measure point	Comment
DC jack input	VSYS	5	-	TP51	Main supply for board
BD71850_BUCK1	VDD_SOC&DRAM&PU_0V9	0.85/0.95[1]	-	TP22	-
BD71850_BUCK2	VDD_ARM_0V9	0.85/0.95/1.0[2]	-	TP23	-
BD71850_BUCK6	VDD_3V3/NVCC_3V3	3.3	-	TP25	-
BD71850_BUCK7	VDD_1V8/NVC_1V8	1.8	-	TP26	-
BD71850_BUCK8	NVCC_DRAM_1V1	1.1/1.2/1.35[3]	-	TP27	-

BD71850_LDO1	NVCC_SNVS_1V8	1.8	-	TP28	-
BD71850_LDO2	VDD_SNVS_0V8	0.8	-	TP29	-
BD71850_LDO3	VDDA_1V8	1.8	-	TP32	-
BD71850_LDO6	VDD_PHY_1V2	1.2	-	TP31	-
BD71850_MUXSW	NVCC_SD2	3.3/1.8	-	TP33	Can be either under SW

1. The default output voltage of BD71850 BUCK1 is 0.8 V. The software will change it to 0.85 V for nominal mode, 0.95V for overdrive mode in SPL before DDR initialization.
2. The default output voltage of BD71850 BUCK2 is 0.9 V. The software will change it to 0.85 V for 1.2 GHz operation, 0.95 V for 1.4 GHz, 1.0 V for 1.5 GHz.
3. 1.1V for LPDDR4, 1.2V for DDR4, 1.35V for DDR3L, BD71850_BUCK8 default output voltage is 1.1V. Software will change it to the required value in SPL before DDR initialization

4.4. Checking for clock pitfalls

Problems with the external clocks are another board bring-up issue. Ensure that all the clock sources are running as expected.

The 24M_XTALI/24M_XTALO, and the RTC clocks are the main clock sources for 24 MHz and 32.768 kHz reference clocks. Although not required, the use of low jitter external oscillators to feed CLK1_P/N can be an advantage if low jitter or special frequency clock sources are required by modules driven by CLKIN_1/2. See the CCM chapter in the i.MX 8M Nano chip reference manual for details.

When checking crystal frequencies, using an active probe is recommended to avoid excessive loading. A passive probe might inhibit the 24 MHz oscillators from starting up. Use the following guidelines:

- RTC clock is running at 32.768 kHz.
- 24M_XTALI/24M_XTALO is running at 24 MHz (used for the PLL reference).

4.5. Avoiding reset pitfalls

Follow these guidelines to ensure that you are booting correctly.

- During initial power-on while asserting the POR_B reset signal, ensure that 24 MHz and 32.768 kHz clock is active before releasing POR_B.
- Follow the recommended power-up sequence specified in the i.MX 8M Nano data sheet.
- Ensure the POR_B signal remains asserted (low) until all voltage rails associated with bootup are ON.

The BOOT_MODE[3:0] balls and internal fuses control boot. For a more detailed description about the boot modes, see the system boot chapter in the chip reference manual.

4.6. Sample board bring-up checklist

The checklist incorporates the recommendations described in the previous sections. Blank cells should be filled in during the bring-up.

Table 33. **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 powering ON.		
2. Verify all i.MX 8M Nano voltage rails	Confirm that the voltages match the data sheet's requirements. Be sure to check voltages as close to the i.MX 8M Nano as possible (like on a bypass capacitor). This reveals any IR drops on the board that could cause issues later. Ideally, all the i.MX 8M Nano voltage rails should be checked, but see guidance below for important rails to check for the i.MX 8M Nano.		
	VDD_SNVS, NVCC_SNVS, VDD_SOC, VDD_ARM, VDD_DRAM, NVCC_DRAM are particularly important voltages, and must fall within the parameters provided in the i.MX 8M Nano data sheet.		
3. Verify power-up sequence	Verify that power on reset (POR_B) is deserted (high) after all power rails have come up and are stable. See the i.MX 8M Nano data sheet for details about power-up sequencing.		
4. Measure/probe input clocks (32.768 kHz, 24 MHz, others)	Without proper clocks, the i.MX 8M Nano will not function correctly.		
5. Check JTAG connectivity	This is one of the most fundamental and basic access points to the i.MX 8M Nano to allow the debug and execution of low level code, and probe/access processor memory.		
Note: The following items may be worked on in parallel with other bring-up tasks.			
Access internal RAM	Verify basic operation of the i.MX 8M Nano in system. The on-chip internal RAM starts at address 0x0090 0000 and is 128 Kbytes in density. Perform a basic test by performing a write-read-verify operation to the internal RAM. No software initialization is required to access internal RAM.		
Verify CLKO outputs (measure and verify default clock frequencies for desired clock output options) if the board design supports the probing of clock output balls.	This ensures that the corresponding clock is working and that the PLLs are working. This step requires chip initialization, for example, via the JTAG debugger, to properly set up the IOMUX to output clocks to I/O balls and to set up the clock control module to output the desired clock. See the chip reference manual for more details.		
Measure boot mode frequencies. Set the boot configure 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 connectivity of signals between the i.MX 8M Nano and boot device and that the boot mode signals are properly set. See the "System Boot" chapter in the chip reference manual for details for boot mode configurations.		
Run basic DDR initialization and test memory	1. Assuming the use of a JTAG debugger, run the DDR initialization and open a debugger memory		

Table 33. **Board bring-up checklist**

Checklist item	Details	Owner	Findings &Status
	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. Users should recheck the schematic to ensure that the DDR memory has been connected to the i.MX 8M Nano correctly.		

5. Using BSDL for Board-level Testing

5.1. 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.

5.2. 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.

5.3. 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.

5.4. 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.

5.5. Boundary scan operation

When using the BSDL file to force the i.MX8M Nano enter the boundary scan mode on the EVK board, it needs a special setup sequence as below:

- BOOT_MODE0(G26)&BOOT_MODE1(G27)& BOOT_MODE2(C27)& BOOT_MODE3(D26) should be high, set the SWITCH SW1101[1-4]: 1111, then power up the board.
- Load the instruction and data codes to the chip TAP controller by JTAG port to enter boundary-scan mode.
    - Soft reset the JTAG TAP controller, go to Test-Logic Reset state.
    - Go to Shift-IR state, load IR Instruction = b10000.
    - Go to RUN-TEST-IDLE sate.
    - Go to Shift-DR state, load DR data = b00111000
    - Go to RUN-TEST-IDLE state
    - End task.

  Refer to **Figure 8** for the TAP controller state, TMS is used to determine the next state of the TAP controller, the serial instructions and data are received by the test logic at TDI. The value shown adjacent to each state transition in this figure represents the signal present at TMS at the time of a rising edge at TCK.

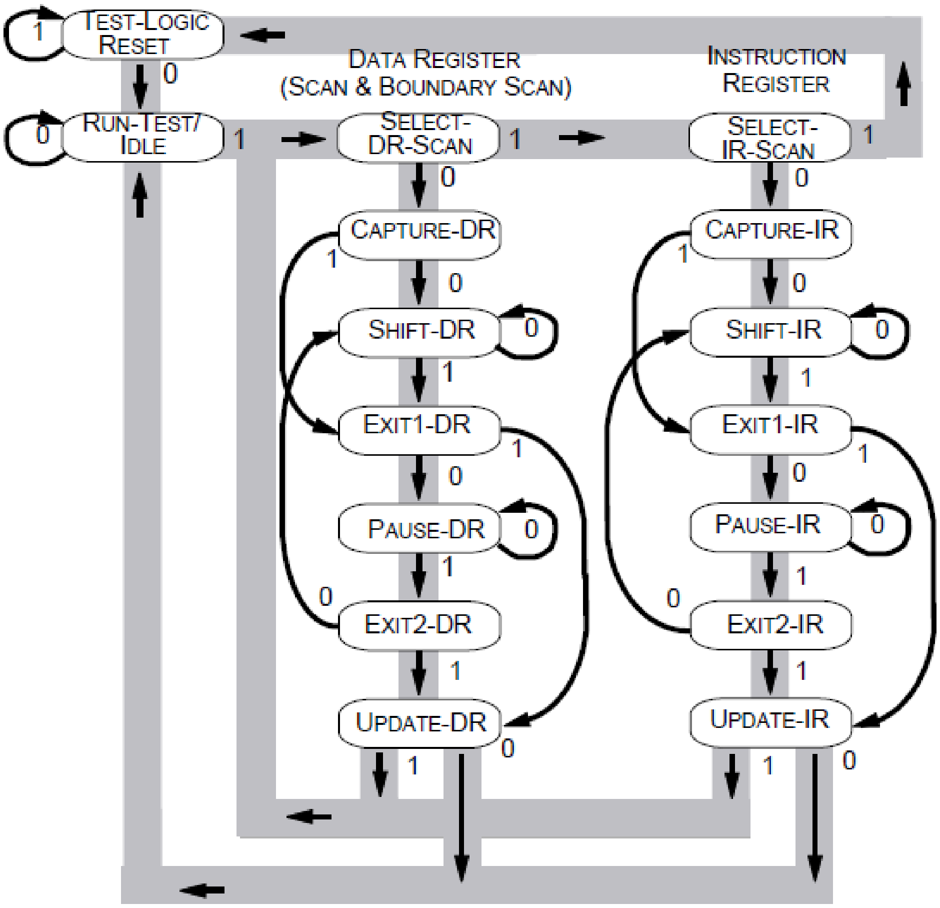
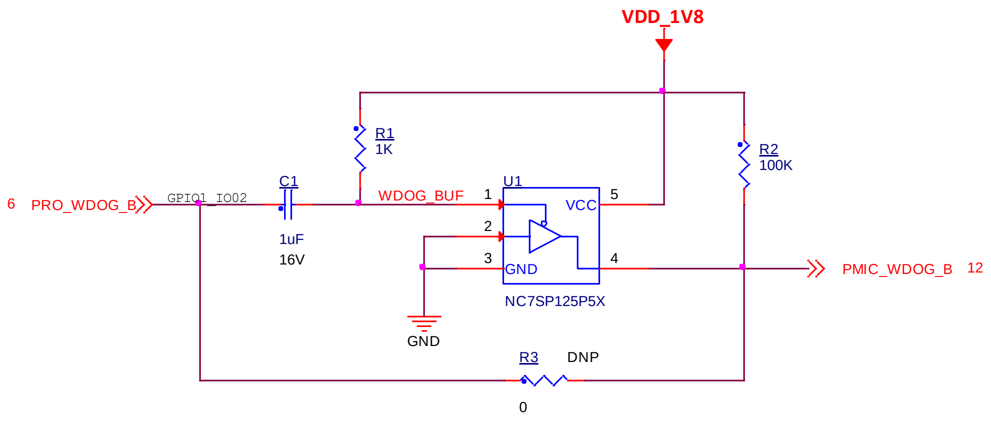


Figure 8. TAP controller state diagram

NOTE

1. GPIO1_IO02 is used as WDOG_B to toggle the PMIC. When entering boundary-scan mode, this pin is always low which will repeatedly reset the PMIC and prevent the system boot up normally. So the PMIC WDOG_B pin can’t be connected to GPIO1_IO02 and should be pulled up to 100Kohm resistor during boundary scan test. Or use the WDOG timer buffer circuit as below.



If the boundary-scan test is not required or the PMIC disable WDOG_B reset by default, GPIO1_IO02(WDOG_B, ball AG13) can be connected to the external PMIC directly without this circuit.

2. In the BSDL file, XTALI_24M(B27)/XTALO_24M(B25) “PIN Name” does not match the datasheet because the JTAG1149 specification does not allow a number as the first character. The “PIN Name” is an inversion in the BSDL file.

5.6. 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.

6. Thermal Considerations

6.1. Introduction

This chapter introduces basic thermal considerations that need to be considered, when designing an i.MX 8M Nano processor-based system. PCBs should be designed with the thermal requirements factored in early as only remedial actions are possible after. Factoring thermal management at the end of the design cycle will increase the cost of the overall design and delay productization. This section provides a few key design considerations to improve the thermal management of the final i.MX 8 based system/product.

The Thermal Design Power (TDP), represents the maximum sustained power dissipated by the processor, across a set of realistic applications. The activity profile of the application can have a significant impact on the thermal management techniques used and on the TDP.

If the customer application requires high performance for extended periods and/or if the product is required to function in high ambient temperatures, the usage of passive thermal management techniques such as a heatsink becomes necessary. For very high ambient operating environments active thermal management techniques such a cooling fan or forced convection may also be required in addition to the heatsink.

For most of the applications it may be feasible to consider the PCB as the only heat dissipation media, providing that good design practices are followed.

6.2. PCB Dimensions

The dimensions of the PCB directly affect its capability to dissipate the heat. Typically, more than 80% of the heat generated by a high-power component is dissipated through the system board, when no other thermal solution is implemented. The bigger the board is, the larger the surface area through which heat can spread away from the source component and also can be transferred more efficiently into free space.

NXP conducted PCB sensitivity simulation showed that a 50% reduction in PCB x-y dimensions results in an increase of between 44-65% in package thermal resistance due to the loss in conductive volume to dissipate heat. System designers hence need to be careful when designing smaller form factor boards that have multiple high-power components.

6.3. Copper Volume

Increasing the heat dissipation (reducing thermal resistance) can also be achieved by increasing the metallization in the system board. PCBs are made up of copper and dielectric material, with the copper being orders of magnitude more thermally conductive. Copper volume influences the heat capacity of the board. With higher copper volume the board can accept more heat, so a i.MX 8 processor-based system can operate in high performance state (or near the max TDP) for longer time periods. The copper volume can be increased by increasing the dimensions of the board, by addition of ground layers or by increasing the thicknesses of the layers on which power and ground planes are located. Please refer to **Table 19** and **Table 20** for stack-up of the EVK.

6.3.1. PCB Material Selection

As previously discussed PCB material selection is extremely important for systems with high speed routing. Thermal properties should also be considered in selecting PCB materials for multilayer designs in which the system is expected to endure excessive short-term thermal steps. Specific attention should be paid to the fact that thermal properties of dielectrics are often different in horizontal and vertical directions.

Material characteristics such as the Coefficient of Thermal Expansion (CTE) should be considered. The CTE describes how a material changes dimension with temperature. Ideally, a PCB material's CTE should be closely matched to copper, which is about 17 ppm°C. CTE is a concern because as the PCB expands during heating, it can elongate plated via holes and cause fracturing. If the CTE is closely matched to copper, expansion of the PCB material and copper will be more uniform and the plated via holes will be more robust during thermal cycling.

6.4. Thermal Resistance

Reducing the thermal resistance close to the die and package is mandatory for good thermal performance. The actual semiconductor die dimensions are relatively small compared to the size of a typical PCB, which results in a very high heat flux in the die, package and its immediate vicinity. Therefore, thermal resistance encountered early in the thermal path causes a large temperature gradient. The most effective place to focus resources to reduce thermal resistance is where the thermal gradient is the highest. To efficiently dissipate the heat through the board, thermal resistance between the SoC and the board needs to be minimized. This can be achieved by utilizing all the ground pads of the component and using underfill with good thermal conductivity properties.

6.4.1. Heat Spreaders

Thermal resistance can be reduced when a heat spreader is mounted on the top of SoC package using a Thermal Interface Material (TIM) with good thermal conductivity properties (thermal paste). If the heat spreader is also thermally connected with the PCB, an alternative route for the heat is created, reducing the global thermal resistance. Spreading the heat at the beginning of the thermal path not only reduces the thermal resistance near the source component, but it also provides a broader area to further disseminate the heat.

The type of heat spreader to be used is dependent on the customers’ application available enclosure space and budget considerations. Graphite heat spreaders are quite common as they match the thermal performance of copper in two directions (x, y), at a lower weight and cost. The high in-plane (basal) thermal conductivity results in spreading and evening out of the hot spots Due its low cost the area that the graphite heat spreader covers, could be potentially larger covering all heat generating components on the system board.

6.4.2. Thermal Vias

Using a continuous low thermal impedance path from the processor to ambient conditions is important and a low thermal resistance has to be maintained throughout the PCB. Any small break in the low impedance path is highly detrimental. System designers should provide redundant thermal paths where possible. This can be achieved by adding an appropriate amount of thermal vias to connect all the ground planes together and allow the heat to spread uniformly through other layers of the PCB. System designers should allocate enough plated through vias around the ground and power balls of the i.MX processor and other heat generating components.

6.5. Power Net Design

Modern power electronics devices can have very low on-resistances. It’s quite possible that the PCB traces and connector pins that feed current to these devices contribute more ohmic losses to the system than the power transistors do. Such heating may be avoidable if traces are up-sized. Reducing trace ohmic losses may be the least expensive way to reduce the design's total power dissipation. Trace width calculators, which also predict trace temperature rise, are readily available on the internet. Using over-sized power transistors is a way to cut total power and subsequent heat dissipation.

6.6. Component Placement

The i.MX 8 processor should always be placed away from edges in the center of the PCB so that heat can effectively spread in all directions. Placing the device on the edge or even on the corner of the PCB significantly reduces heat transfer from the device and dissipation capabilities of the to the PCB, as the heat cannot efficiently spread in the directions where the edges are present. This eventually results in local hot spots and rapid heating of the source component.

In addition, the processor should be mounted on the top side of the PCB, away from the chassis walls. System designers should place heat generating components as far apart as possible to reduce thermal coupling effects. The thermal gradient is high near a power dissipating device, so even small amounts of separation help reduce thermal coupling.

A NXP conducted PCB sensitivity simulation showed that a non-centered bare i.MX 8 FCBGA package on the PCB will cause approximately 8-10% increase in junction temperature due to uneven heat propagation. This study highlights the need for centered component placement.

6.7. PCB Surroundings

The surroundings of the PCB also influence the efficiency of heat transfer from the board into free space (air). There should be enough clearance from the top and bottom sides of the PCB. If narrow gaps are created, air flow is significantly limited, resulting in accumulation of hot air in the gap. The board cannot therefore effectively transfer heat in such areas. Also, the casing should be designed in a way that natural air convection could be utilized to improve heat transfer.

If a narrow gap at the bottom side cannot be avoided (quite common for System on Modules - SOMs), it should be considered to fill the gap under the i.MX processor by thermally conductive gap filler. To further improve heat transfer, exposed copper pads should be added to the base board at the mounting spot of the filler.

6.7.1. Air Flow Considerations

Heat convection is more efficient for a vertically mounted board. Remember that components above heat producing devices run hotter than those below. If the board is to be horizontally mounted, place heat generating components on the PCB’s topside, if possible. A thermal plume (the chimney effect) forms more readily on a board’s topside and it helps disperse heat.

- Consider the system level air flow and air mover placement in the enclosure
- Avoid sub-optimal component placement that might hinder airflow or natural convection
- Avoid placing tall or bulky devices in the air flow path
- Avoid routing circuitry in an area where mounting holes would need to go
- Plan to make space for the thermal management solution early in the system design phase and consider the complete board and packaging form factor (enclosure)

6.8. Thermal Simulations

As illustrated in this section, thermal management is a very complex discipline with numerous variables that need to be considered. In order to determine whether the system is capable of stable operation (no thermal runaways) in the given use case or to identify potentially overlooked issues, thermal simulations have to be performed.

NXP can provide FloTHERM simulation models for i.MX 8 series processor family and strongly encourages customers to perform thermal simulations using these models in their form factor designs and specific use cases to get a holistic system thermal design & identify possible thermal bottlenecks. Thermal simulations become increasingly important in small form factor designs and operation in high ambient temperatures.

6.9. Software optimization

Software based power and thermal management techniques can be very effective in reducing the need for more elaborate active or passive thermal management solutions and add little or no additional cost to the system design. Attention should be paid to the required system performance and power requirements, as lower the i.MX 8 processor power consumption lower the heat generated by the processor.

The i.MX 8 series incorporates several low-power design techniques, to meet requirements of low-power design, while sustaining high performance operation. The activity profile of the customer application can have a significant impact on the thermal management techniques used and on the TDP. Carefully defining the system's worst case operating conditions can be an effective way to reduce power and thermal dissipation.

- System designers should utilize and enable all software power management techniques available for the i.MX 8 Series
- The SoC voltages and core frequencies of modules should be kept at the minimum specified levels and scaled dynamically with respect to the current performance demands of the application where possible
- The processor should enter low power modes under certain use cases whenever possible
- All unused power rails should be turned off from the PMIC and power gate unused domains if possible
- All unused module clocks should be turned off (Dynamically handled by NXP Linux BSP)
- Customers are encouraged to use the latest Linux BSP GA release available on nxp.com, that leverages the i.MX 8 processor power management features and incorporates various Linux software power management techniques

6.10. The Thermal Checklist

NXP recommends using the checklist below as a high-level guide for designing an optimal thermal management solution for your end product:

Table 34. **Thermal checklist**

Item	Activity	Check
1	Determine the TDP (Thermal Design Power)	
2	Determine the Activity Profile (use case dependent)	
3	Determine the product form factor constraints (orientation, x, y & z limits etc.)	
4	Determine the environmental operating conditions (ambient temperature, airflow regime - Forced or Natural Convection)	
5	Determine the Tj for the i.MX 8 device to use (Auto, Industrial, consumer, package lidded etc.)	
6	Factor in board design considerations early (PCB layers, metallization, layout, component placement)	
7	Run thermal simulations to determine the best thermal management approach using form factor design and use cases	
8	Investigate adding heat spreading techniques, heatsinks to alleviate thermal bottlenecks	
9	Enable all software power management techniques which can minimize power consumption (less power, less heat)	
10	Consider lower power memory and other system components, or retarget use case	

7. Revision history

Table 35. **Revision history**

Revision number	Date	Substantive changes
0	12/2019	Initial release

How to Reach Us:

Home Page:
nxp.com

Web Support:
nxp.com/support

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

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

While NXP has implemented advanced security features, all products may be subject to unidentified vulnerabilities. Customers are responsible for the design and operation of their applications and products to reduce the effect of these vulnerabilities on customer’s applications and products, and NXP accepts no liability for any vulnerability that is discovered. Customers should implement appropriate design and operating safeguards to minimize the risks associated with their applications and products.

NXP, NXP, the NXP logo, NXP SECURE CONNECTIONS FOR A SMARTER WORLD, COOLFLUX, EMBRACE, GREENCHIP, HITAG, I2C BUS, ICODE, JCOP, LIFE VIBES, MIFARE, MIFARE CLASSIC, MIFARE DESFire, MIFARE PLUS, MIFARE FLEX, MANTIS, MIFARE ULTRALIGHT, MIFARE4MOBILE, MIGLO, NTAG, ROADLINK, SMARTLX, SMARTMX, STARPLUG, TOPFET, TRENCHMOS, UCODE, Freescale, the Freescale logo, AltiVec, C-5, CodeTEST, CodeWarrior, ColdFire, ColdFire+, C-Ware, the Energy Efficient Solutions logo, Kinetis, Layerscape, MagniV, mobileGT, PEG, PowerQUICC, Processor Expert, QorIQ, QorIQ Qonverge, Ready Play, SafeAssure, the SafeAssure logo, StarCore, Symphony, VortiQa, Vybrid, Airfast, BeeKit, BeeStack, CoreNet, Flexis, MXC, Platform in a Package, QUICC Engine, SMARTMOS, Tower, TurboLink, and UMEMS,EdgeScale, EdgeLock, eIQ, and Immersive 3D are trademarks of NXP B.V. All other product or service names are the property of their respective owners. Arm, AMBA, Arm Powered, Artisan, Cortex, Jazelle, Keil, SecurCore, Thumb, TrustZone, and μVision are registered trademarks of Arm Limited (or its subsidiaries) in the EU and/or elsewhere. Arm7, Arm9, Arm11, big.LITTLE, CoreLink, CoreSight, DesignStart, Mali, Mbed, NEON, POP, Sensinode, Socrates, ULINK and Versatile are trademarks of Arm Limited (or its subsidiaries) in the EU and/or elsewhere. All rights reserved. Oracle and Java are registered trademarks of Oracle and/or its affiliates. The Power Architecture and Power.org word marks and the Power and Power.org logos and related marks are trademarks and service marks licensed by Power.org.

© 2019 NXP B.V.

Document Number: IMX8MNHDG
Rev. 0
12/2019