

Introduction



This document summarizes hardware design knowledge required to design PCBs with target devices, along with design examples and a design checklist. It also describes the restrictions and recommendations described in its hardware manuals, as well as includes some hints to reduce troubles in hardware development.

Target Device

- RZ/V2H
- RZ/V2N

NOTE

The design of evaluation board kits might not meet items in the checklist due to internal evaluation and realizing multiplexed functions, however, we strongly recommend to design along with the checklist.

In figures in this document,  marks indicate prohibited items, and  marks indicate items to notice.

Please also refer to the following documents.

- User's Manual: Hardware
- Design Data of EV Kit
- Other Application Notes for Hardware Design

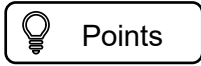
Table of Contents

1. Reset.....	4
1.1 Circuits for QRESN Input.....	4
1.2 Note for the Reset Signal with Open-Drain	5
1.3 Reset Timing.....	6
1.4 Resetting Peripheral Devices	7
1.5 Mode Pins.....	8
2. Clock	9
2.1 Input and Output Clock	9
2.2 Accuracy and Deviation	10
2.3 Crystal Oscillator Circuits	11
2.4 Clock Input for Peripheral Devices	12
2.5 General Notes on the Clock Circuit and Wiring.....	13
3. Power Supplies	14
3.1 Capacity for MPU and Peripherals	14
3.2 Capacity for VDD	16
3.3 Supply Voltage and Power Sequence	17
3.4 Noise Filter.....	18
3.5 Impedance Design.....	19
3.6 Bypass Capacitors.....	20
3.7 Selecting Ceramic Capacitors	21
3.8 Design of the Power Supply IC Circuit	22
3.9 Remote Sensing of the Output Voltage by the Power Supply IC	23
3.10 Supplying Power through Connectors.....	24
3.11 SD	25
3.11.1 Power Supply Circuit for the SD card Interface	25
3.11.2 Switching the IO Power Supply Voltage	26
3.12 Reverse Current through USB 2.0 Power Supplies	27
4. High-Speed Interfaces	29
4.1 Other Guidelines and Documents	29
4.2 DDR	30
4.2.1 Power Integrity and Signal Integrity Simulations	30
4.2.2 Pin Swapping for LPDDR4/LPDDR4X.....	31
4.2.3 Power-on Sequence for LPDDR4.....	32
4.2.4 Power-on Sequence for LPDDR4X	32
4.3 Resistor and Capacitor	33

4.4	Clock	34
4.5	Instructions in schematic for PCB Layout	35
4.6	Observing High-Speed Serial Interface Signals	36
4.7	Disturbance Noises and Static Electricity	37
5.	Others	38
5.1	Unused Pins	38
5.2	Countermeasures against Overshoots and Undershoots	39
5.3	Level Shifting	40
5.4	IO Characteristics between MPU and Connected Devices	41
5.5	Conflict between Pull-Up and Pull-Down	42
5.6	Relatively High-Speed Interfaces (about 10 to 100 MHz)	43
5.7	Notes on Signal Connection	44
5.8	Polarity of Electrolytic Capacitors	45
5.9	Programming On-Board EEPROM	46
5.10	Board to Board Connector	47
5.11	Reconfirm Operations of Re-used Circuits	48
6.	PCB Layout Design	49
6.1	Power Supply	49
6.1.1	Target Impedance	49
6.1.2	Trace Width	50
6.2	Exposed Pad	52
6.3	Digital and Analog Ground	53
6.4	Ground Shields/Guards	55
6.5	Return Path	57
6.6	High-Speed Interfaces	59
6.6.1	Differential Signals	60
6.6.2	Crosstalk	62
6.6.3	Pads for Monitoring Signals	64
6.7	General Notes	65
6.8	Countermeasures for Noise	66
7.	Check List	67
	REVISION HISTORY	72

1. Reset

1.1 Circuits for QRESN Input



- Check the IO voltage of the QRESN.

The QRESN is a reset input with Schmitt trigger, and it is on the 1.8V power domain.

- Do not set QRESN high when the MPU is powered off.

High level input for the QRESN is not supported while the power supply is turned off. Be sure to input signals according to the power sequence described in the User's Manual. The input timing differs depending on whether or not PWC is used.

- Design the pull-up for the QRESN signal according to your system requirements.
- Also check the reset signals of peripheral devices.

Some peripheral devices may require a reset at the same time as the MPU reset. In such cases, make sure that there is no problem with the voltage and timing of the reset signal of the peripheral device. Also, ensure that the reset signal is stable.

1.2 Note for the Reset Signal with Open-Drain



Points

When an open-drain output, wired OR, and voltage level shifter are used in the circuits for generating the reset signals, take care on the following points.

- Peripherals with a reset terminal with Schmitt trigger IO are recommended.
- Take care of the noise.

The effect of noise such as crosstalk is very likely to lead to the incorrect behavior of reset signals. Therefore, apply countermeasures for noise such as separating the reset signal from other signals and applying a grounded guard.

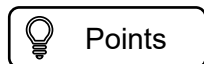
- Determine the pull-up resistor to account for stray capacitance.

The overall wiring length tends to be long because the reset signal branches to other devices and connectors at the edge of the board. As a result, the signal waveform is likely to be dulled by stray capacitance. Consider this when determining the value of the pull-up resistor to be used for the open-drain output.

- Note the pull-up/pulldown resistors scattered throughout the schematic.

If pull-up or pull-down resistors for the reset signal are scattered across multiple pages of the schematic, they may be overlooked and cause unintended operation. For schematic readability, it is recommended that pull-up and pull-down resistors be placed in a single location.

1.3 Reset Timing



- Make sure the following constraints are met for the reset timing.
 - a. The reset control timing differs depending on the boot mode and whether or not PWC is enabled. Be sure to control the reset timing according to the power supply sequence described in the User's Manual.
 - b. When PWC is enabled, if QRESN pin becomes low, the PWEN0 to PWEN2 pins become low simultaneously. This is because PWC is reset. If PWC is used to control the power supply, do not control QRESN while the LSI is starting up.

1.4 Resetting Peripheral Devices

Points

- When using the main reset signals QRESN to reset a peripheral device, take care on the following points.
 - Is power to the peripheral device turned on?
 - Are you observing the correct reset sequence for the peripheral device?
 - Is the active level of the reset signal for the peripheral devices correct?
 - When multiple peripheral devices are connected, are you sure that pull-up or pull-down resistors within the individual devices will not cause the voltage of the reset signal to be at an intermediate level?
 - Is the driving ability of the reset signals sufficient?
- When the target devices have reset pins for exclusive use with specific modules such as the DDRn_RESETN pin for DRAM and the XSPI0_RESET0N pin for the SPI Multi I/O interface, the SD0RSTN pin for the SD/MMC host interface, and so on, use these pins instead of the QRESN pin.

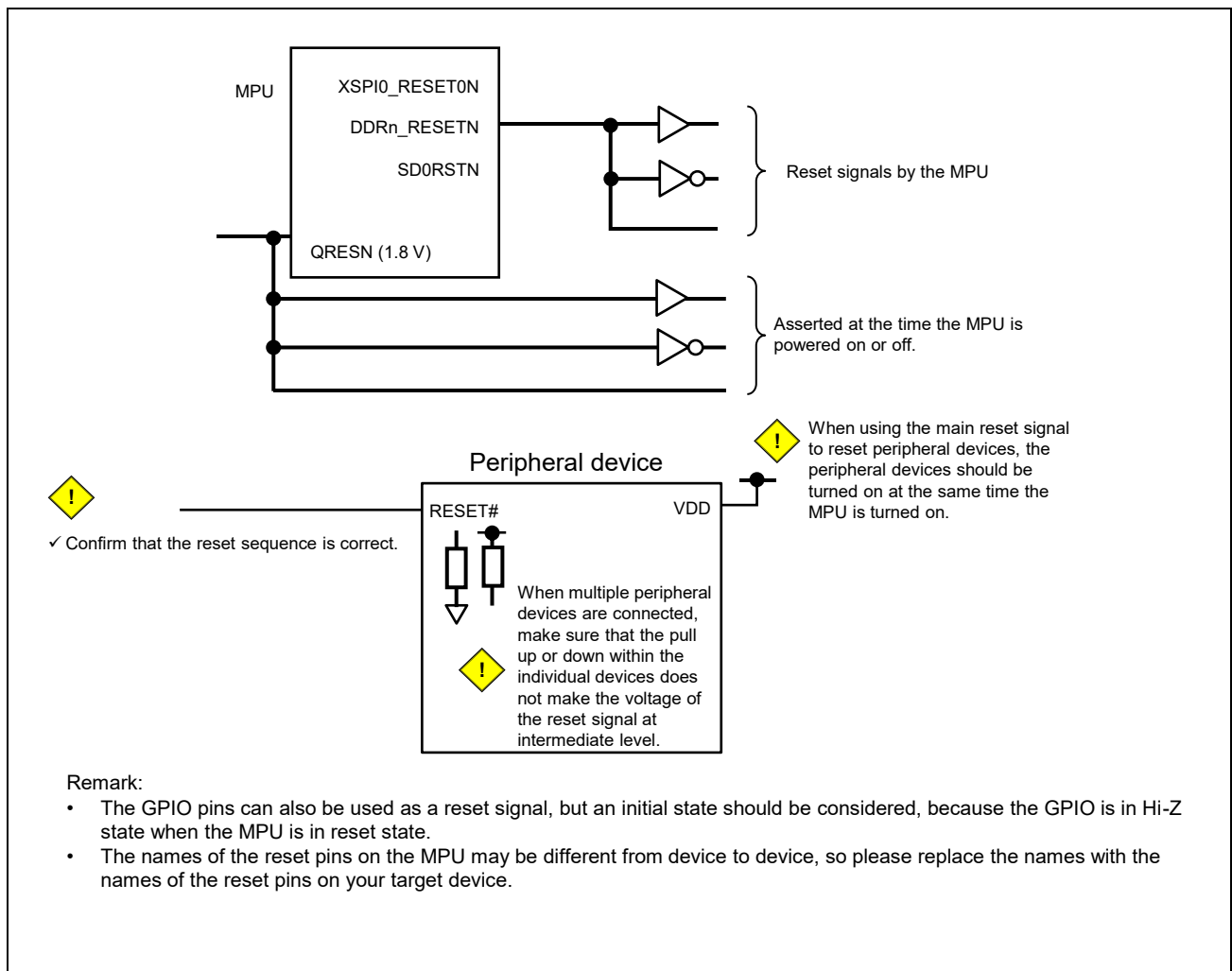


Figure 1.1 Resetting Peripheral Devices

1.5 Mode Pins



Points

The mode pins (MD_BOOTn / MD_CLKS) are used to set the initial operating mode. They are implemented with pull-up or pull-down resistors and are captured only one time after the QRESN is released, not after a software reset. The level of the mode pins should be held for a time specified as tMDH. Please refer to the User's Manual for the pin specifications.

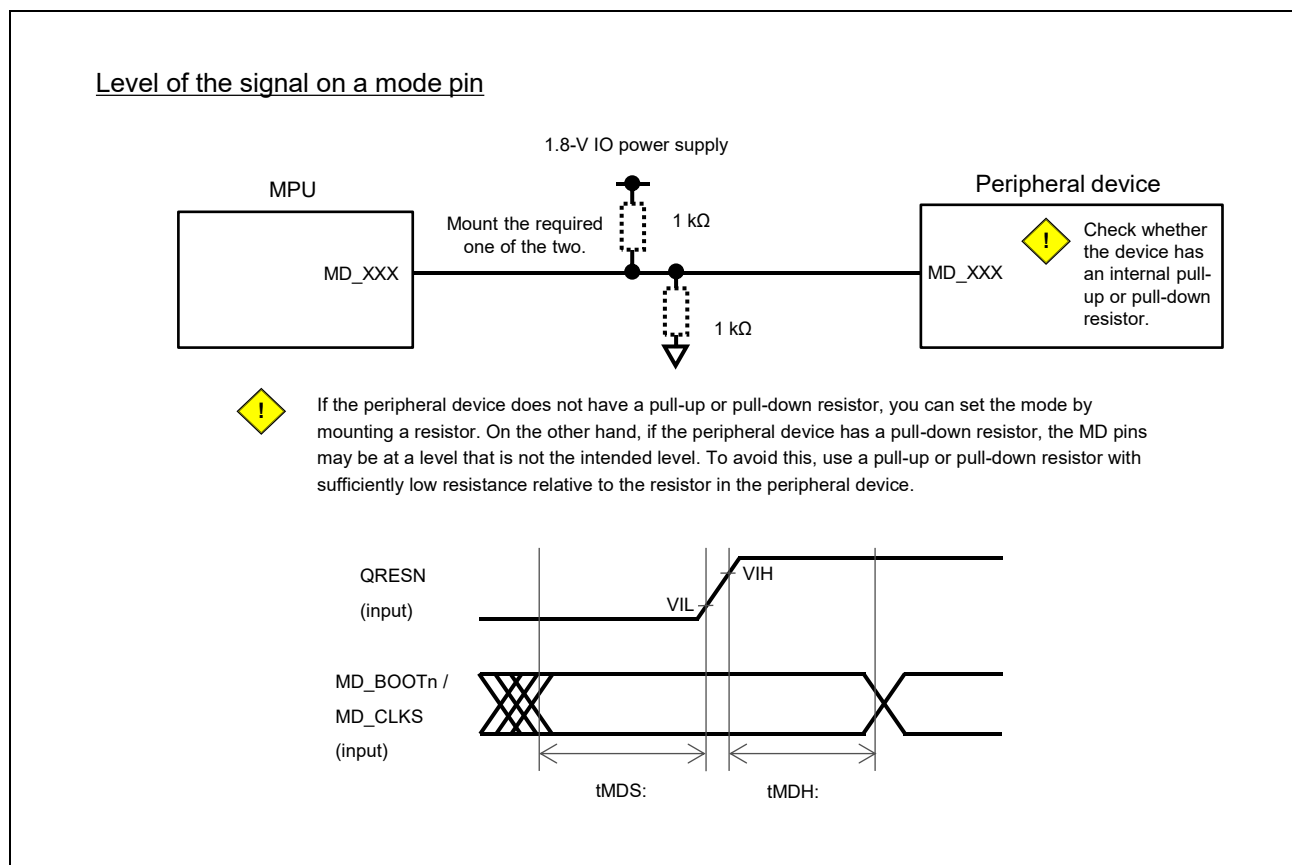


Figure 1.2 Handling the Mode Pins

2. Clock

2.1 Input and Output Clock



- The voltage of the clock should conform to the voltage of its power domain.
- The spread spectrum clock generator (SSCG) cannot be used as the input clock unless specifically stated otherwise in the hardware manual.
- Make sure that the input clock is accurate enough.

The system clock is frequency-multiplied to obtain each kind of clock. Deviation of the oscillation frequency (center frequency) of the input clock (the system clock) affects the center frequency and deviation of the DDR_CLK.

- Make sure that whether the output clock of the MPU satisfies with the requirement of the input clock of the external peripheral devices. Refer to the CPG section and Electrical Characteristics of the User's Manual.

2.2 Accuracy and Deviation



Points

- In general, the specifications of crystal resonators are based on the temperature characteristics, the degree to which oscillation may deviate from the oscillation frequency (center frequency), and so on.
- As with crystal resonators, the specifications of crystal oscillators and clock synthesizers are also based on the degree to which oscillation may deviate from the oscillation frequency (center frequency).
- When using the clock generated inside the target devices as the clock source of external peripheral devices or modules, you should be more careful for the deviation of the input clock of the MPU.
- The system clock provides the source of each kind of clock through frequency-multiplication. Any deviation in the frequency of the input clock (the system clock) thus produces deviation in the frequency of clock after multiplication.
- To design a system such that the maximum frequency of the DDR_CLK does not exceed the maximum frequency specified for the DRAM, you must select the center frequency of the system clock input in consideration of possible deviation.
- When using MIPI DSI, or USB, the jitter of the external clock should be less than the values in **Table 2.1**. On the other hand, when not using any of MIPI DSI, and USB, the jitter of the external clock should be less than 80 ps.
- We recommend using clocks with less than 50 ppm frequency deviation.

Table 2.1 QEXTAL Input Clock Jitter

Module	Symbol	Parameter		Max	Unit
USB3	Rj	RMS phase jitter	12 kHz – 20 MHz	2.3	ps, rms
			2 MHz – 20 MHz	1.7	ps, rms
	Dj	Reference clock deterministic jitter	1 MHz – 100 MHz	7.4	ps, pp
			3 MHz – 20 MHz	3.7	ps, pp
MIPI DSI	Dj	Reference clock deterministic jitter	—	40	ps, pp
USB2	Dj	Reference clock deterministic jitter	—	40	ps, pp
Others	Dj	Reference clock deterministic jitter	—	80	ps, pp

2.3 Crystal Oscillator Circuits



Points

- For the load capacitance, damping resistance and feedback resistance of the crystal oscillator circuit, determine appropriate values based on the evaluation of oscillation by the selected crystal resonator manufacturer.
- For the provisional value of the load capacitance before evaluating the oscillation, select a value that meets the characteristics of the resonator.
- In evaluating the oscillation, adding a feedback resistance is relatively easy, but adding a damping resistance is difficult. Select the value of the damping resistor in consideration of adjustment in evaluating the oscillation.
- Observe the following notes for the crystal oscillator circuit, even though they are not stated in the hardware manual:
 - Bypass Capacitors
Insert laminated ceramic capacitors as bypass capacitors for each VSS/VCC pair. Mount the bypass capacitor near the power supply pins of the LSI. Use components with a frequency characteristic suitable for the operating frequency of the LSI, as well as a suitable capacitance value.
 - Notes on Using a PLL Oscillation Circuit
Keep the wiring from the PLL VDD and VSS connection pattern to the power supply pins short, and make the pattern width large, to minimize the inductance component. The analog power supply system of the PLL circuits is sensitive to noise. Therefore, system malfunction may occur by the intervention with another power supply. Do not supply the analog power supply with the same resource as the digital power supply of VDD and VCC.
 - When Using an External Crystal Resonator
Place the crystal resonator, capacitors C_{in} and C_{out} , and damping resistor R_d as close to the XIN and XOUT pins as possible. To minimize induction and thus obtain oscillation at the correct frequency, the capacitors to be attached to the resonator must be grounded to the same ground. Do not bring wiring patterns close to these components.
- Observe the following other notes for the crystal oscillator circuit:
 - Do not cross the circuit with other signal lines.
 - Keep the circuit away from lines that carry large currents.
 - Do not ground the circuit to ground patterns that carry large currents.

2.4 Clock Input for Peripheral Devices



Points

- It is also necessary to pay attention to the clock input and clock oscillator circuit of peripheral devices. Read the data sheets and manuals of the individual peripheral devices carefully to avoid improper design.
- If you intend to use the output clock of the MPU as the input clock for a peripheral device, pay attention to the voltage of the clock output (the power domain to which the clock pin belongs and the interface level), and the clock frequency and accuracy.
- Please refer to the datasheet of the peripheral devices to be used.

2.5 General Notes on the Clock Circuit and Wiring



Points

- Pay attention to the following general notes when designing the clock input and clock output circuits. In addition, important points to be considered in PCB pattern design, such as the GND shield, are helpful in circuit diagram review and evaluation after the board is completed. State the important points as notes on the circuit diagram as far as possible.
 - Properly reduce the load on the clock line to prevent malfunctions due to rounding of the clock signals on rising or falling edges. When the load is too large, take measures such as inserting a clock buffer.
 - Avoid unnecessary branches and stubs as much as possible to avoid degradation of the signal quality caused by reflection. Use continuous patterns for branching as much as possible. When the wiring requires a T branch, take measures such as inserting a clock buffer.
 - When inserting a clock buffer, be sure to consider the effect of clock delay. When delay is a problem, take measures such as using a zero-delay buffer.
 - Prepare circuits and pads in consideration of adjustment of the clock quality after the board is completed.
 - Insert a damping resistor such that the resistor is near to the outputting terminal of the clock signal
 - Placing a termination circuit such as a Thevenin termination near to the receiving terminal of the clock signal
 - Use routes that minimize variations in characteristic impedance. Measures for this include keeping routes in the same PCB wiring layer and minimizing the use of via holes in clock wiring.
 - Prevent the wiring for the clock and other signals from running side by side to avoid crosstalk noise and take measures such as applying a ground shield where appropriate.
 - Consider the layout in terms of avoiding excessive noise from switched-mode power circuits and so on.
 - In the case of circuits for selecting from among two or more clock signals, use chip jumper blocks with 0Ω resistors and use a pattern that minimizes the number of stubs.

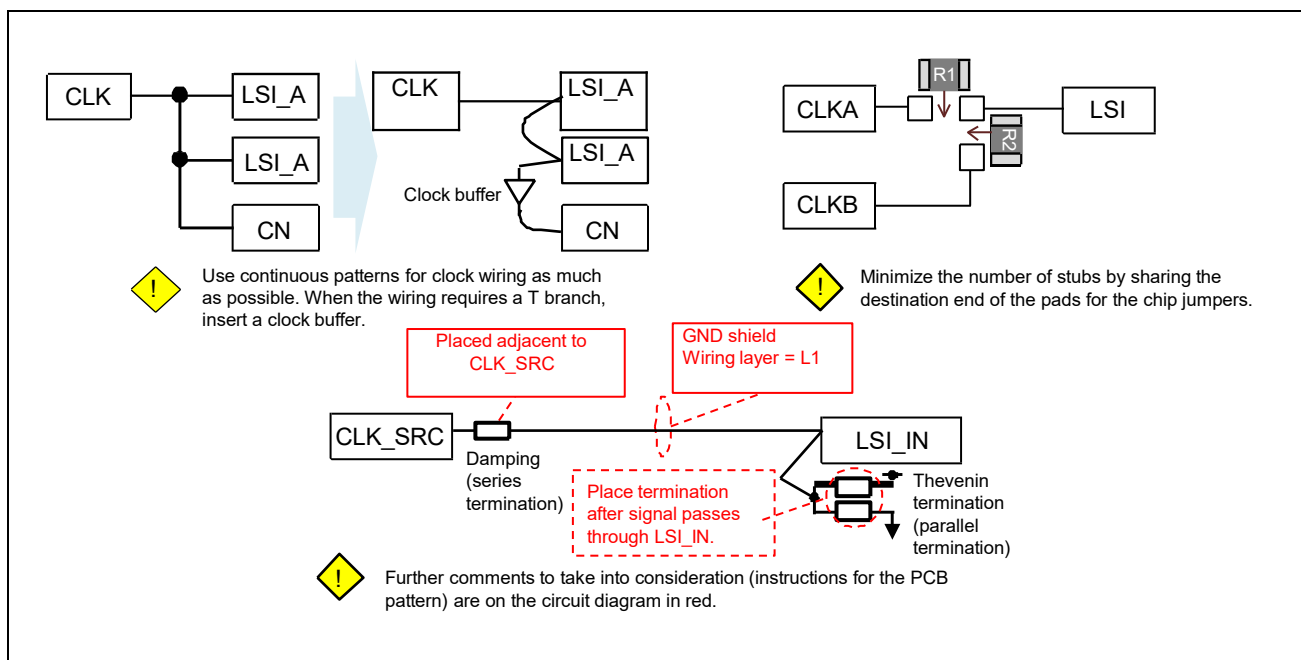


Figure 2.1 General Notes on the Clock Circuit and Wiring

3. Power Supplies

3.1 Capacity for MPU and Peripherals



Points

- In general, the system with the MPU including lots of peripherals such as SD cards, USB, memory devices like DRAMs, and communications devices, etc., requires a wide variety of power supplies. Then its power system is likely to be complex.
- The logic power supply has to be able to provide particularly large currents, so its capacity to supply power should be designed with a margin that allows for increases above the estimate of current drawn.
- For a power supply that is to be the source of multiple power supply voltages, take measures in consideration of the efficiency of each power supply IC, such as stepping down voltages in two stages.
- In order to “visualize” the possibility of power capacity being insufficient, we recommend that you create a calculation sheet, such as a power tree.
- Please refer to the datasheet for the electrical characteristics of the power supply IC and peripheral devices to be used.
- For RZ/V2H and RZ/V2N, the use of PMICs (RAA215300A2GNP#HA7 and DA9215-8D) is recommended as shown in **Figure 3.1**. However, when using CM33 boot on the RZ/V2H, please use the RAA215300A2GNP#HA2 as shown in the **Figure 3.2**, since the software package does not support PMIC control for RAA215300A2GNP#HA7.

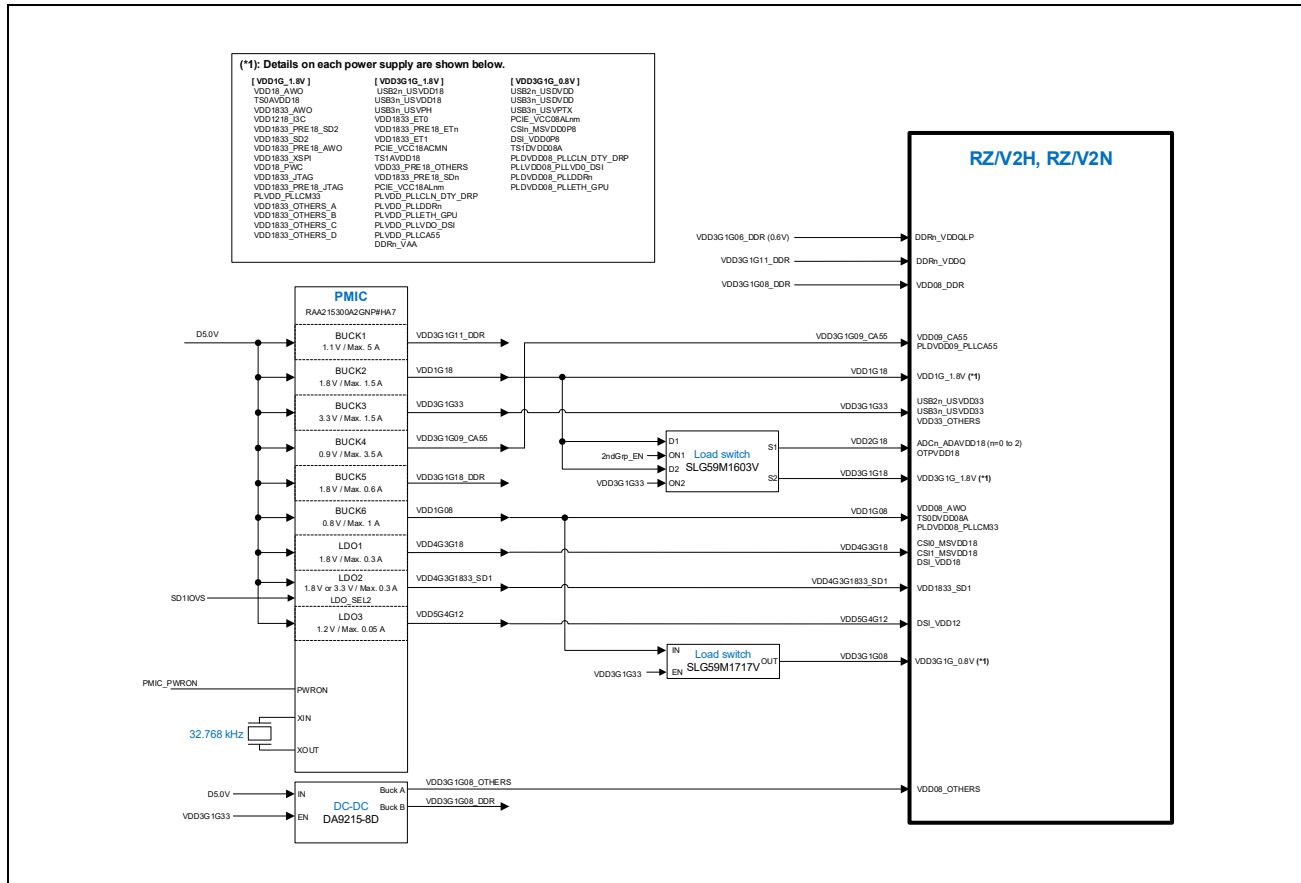


Figure 3.1 Example of a Power Tree



3.2 Capacity for VDD



Points

- When selecting a power supply IC for the logic power supply, make sure that its specifications satisfy the requirement of all the power supply of the MPU, where the specification includes such as the load regulation and load response. For example, the load regulation ability for the power supply IC should satisfy the VDD specification of the MPU.
- Simply calculating, on a desk, the maximum current drawn from the electrical characteristics, and designing the power supply capacity with a margin based on the results of calculation is desirable. However, designing a power supply that allows a margin of two or three times for the maximum current is not easy without significantly increasing costs. The following methods can be used to reduce the cost of the power supply by estimating the details of the standard power consumption.
 - Setting the power supply voltage to $TYP + \alpha$ (allowing for the accuracy of the power supply voltage) to be set for the power supply IC, rather than to the maximum value in the specification
 - Excluding (from the estimate) modules that are not to be used among the internal modules
 - Using duty ratios based on anticipated use cases rather than current drawn during full operation
 - Implementing a margin based on addition rather than multiplication (for example, using +1 A or 2 A instead of two to three times the maximum value)
- If the margin is reduced, the instantaneous peak of the current drawn might exceed the power supply capacity. The following methods can be used to compensate for this.
 - Using several capacitors with large capacitances, such as 10 to 100 μF , to absorb sudden increases in load current
 - Increasing the response of the power supply IC, by increasing the switching frequency of the power supply IC by a value of the order of several MHz, and implementing a multi-channel power supply IC configuration
 - Limiting the overcurrent protection of the power supply IC to the original maximum current + α to prevent unexpected shutdowns
- Please refer to the datasheet of the power supply IC to be used.

3.3 Supply Voltage and Power Sequence



Points

- Check that the power on and off sequence described in the electrical characteristics of the user's manual.
 - We strongly recommend drawing a timing chart of the power up and down sequences of all the power supplies including the reset system of the MPU and peripheral devices.
 - Regarding the sequence of switching power off, the before-and-after relations of items in the sequence might change depending on whether a discharge circuit is present. Therefore, also consider the effect of a discharge circuit on the sequence.
 - Make sure that the rising or falling of the supply voltage is not excessively long. Although the slope (mV/s) of the rising or falling of the power supply voltages is not specified, a monotonic increase or decrease is assumed.
- In the power supply sequence, the condition for the next power-on is that the power supply voltage must have dropped below the rated value when the power supply was turned off last time. Therefore, it should be noted that turning the power supply back on when the power supply voltage has not sufficiently dropped due to insufficient discharge is strictly a violation of the rated value.
- For the evaluation phase, we recommend configuring the initial hardware to allow adjustment for the defect in the power supply sequence. For example, use a PMIC and a MCU which change the control program of the PMIC, and prepare a pattern for a discharge circuit.
- Note the voltage tolerance of the signal pins with pull-up. Not only pins on the MPU, but also the ones on peripheral devices.

3.4 Noise Filter



Points

- Make sure to place noise filters for dedicated power rails such as the PLL. It is not recommended merging multiple power rails and filtering with one filter. Refer to the guidelines for each interface.
- The noise filter circuit of the PLL power supply on the EVK has a proven track record, so we recommend using the same components and circuits as the EVK.

3.5 Impedance Design



Power supplies that require high current, such as VDD, which is a logic power supply, and power supplies for noise-sensitive high-speed signal interfaces such as DDR require a PCB design that takes signal impedance into account.

We provide the following guidelines. Please refer to them. We strongly recommend that the target value be in accordance with these guidelines to reduce the risk of malfunction.

[DDR]

- *PCB verification guide for LPDDR4/4X*
*: Included in Spara / IBIS models (NDA required)

[High speed signal interface, DDR, VDD]

- *PCB Design Guidelines* (NDA required)

3.6 Bypass Capacitors



Points

- There are no recommended capacities for bypass capacitors for the general-purpose digital IO power supplies of the target devices; however, use the evaluation board kit as a reference to determine the appropriate number and capacity of bypass capacitors.
- It is desirable to use a single ceramic 0.1- μ F capacitor per pin of the IO power supply as a bypass capacitor. However, arrange the appropriate number of capacitors in consideration of the signal operating frequencies, the number of operations that will simultaneously be in progress, and the priority of other power bypass capacitors and filter circuits.
- If many signals are operating simultaneously, place a capacitor with a relatively large value of about 10 μ F at the end of the PCB IO power supply pattern.
- Confirm that the bypass capacitors for peripheral devices are appropriate, as well as the MPU. The recommended capacities and PCB wiring patterns might be stated in the provided data sheets and technical documents of some devices.
- Please refer to the RZ/V2H and RZ/V2N *PCB Design Guidelines* (NDA required).
- Please refer to the datasheet of the peripheral devices to be used.

3.7 Selecting Ceramic Capacitors



Points

- When a multilayer ceramic capacitor is used as a bypass capacitor, the effective capacitance decreases depending on the DC bias characteristics and temperature characteristics.
- For details, check the characteristics provided by the manufacturer of a capacitor. However, the trends are as follows:
 - The rate of decrease of the effective capacitance depends on the dielectric of the capacitor in use.
 - The effective capacitance tends to decrease with the working voltage margin for withstand voltage.
 - For capacitors with relatively high capacitance (10 μF or higher), the effective capacitance decreases significantly with the body size.
- Note that when a capacitor with a large capacitance is used to support sudden changes in the load current, the desired effect might not be fully obtained due to a decrease in the effective capacitance.
- If the decrease in the effective capacitance is a problem, also consider the capacitor characteristics: F, X5R (= the dielectric in use).
- When you select a capacitor with a small body, pay attention to the decrease in the effective capacitance. In terms of the effective capacitance, the effective capacitance of two small capacitors can correspond to that of a single larger capacitor.
- Please refer to the datasheet for the electrical characteristics of the multilayer ceramic capacitor to be used.

3.8 Design of the Power Supply IC Circuit



Points

- The datasheets and technical documents for power supply ICs usually cover the reference circuits useful for configuring power supply circuits.
- Especially for switching regulators, the power supply quality might be significantly degraded depending on the circuit constants, or oscillation might occur. Therefore, the manufacturer and model number of the components used in the provided reference circuits are amply stated in many cases.
- Verifying the selection of specific components and the margins (for phase and gain) by using a simulation tool might be possible.
- When you design power supply circuits, use the technical information and reference circuits provided by the power supply IC manufacturers to make sure that the recommended circuits and constants are observed.
- The recommended designs for PCB wiring patterns and heat dissipation might also be stated, follow the recommendations.
- Carefully confirm the specifications of the coils, FETs, diodes, and other components that need to be selected according to the current drawn in the actual use cases so that they do not violate the ratings. We also recommend writing comments regarding the confirmed characteristics on the circuit diagram to help in component selection at design reviews and in response to changes of specifications.
- Pay attention to the allowable losses (W) when inserting a shunt resistor into the power supply line for current measurement.
- Please refer to the datasheet of the power supply IC or peripheral device to be used for the electrical characteristics.
- Please refer to the datasheet for the electrical characteristics of the components used in power supply circuits and power supply IC to be used.

Reference Documents:

- IBIS/Spara simulation models for SI/PI simulations (*Interface Design Model* [NDA required])
- PCB verification guide for LPDDR4/LPDDR4X (included in the file above)

Note

For PCB design, it is recommended to utilize these models for SI/PI simulations. Even when reference board design data are employed, variations in characteristics may occur due to factors such as substrate material properties.

3.9 Remote Sensing of the Output Voltage by the Power Supply IC



Points

Recently, current for the logic power supply has increased and the power supply voltage has gotten lower. So the power supply ICs that have a function sensing the output voltage to improve the quality of a power supply have been appearing. The quality of supplying power can be maintained by placing a remote sensing pin near the load (the LSI) and canceling out IR drops in the power supply line. We provide some general notes here.

- Sense the voltage at the point as close to the load as possible. Sensing the voltage at the pin of the power supply on the MPU is preferable; however, this is often difficult.
- As with other noise-sensitive signals, take countermeasures to avoid noise, such as avoiding parallel runs of remote sensing and other signal wiring, and using ground shields.
- If a paired GND signal is provided, wire the GND signal and the remote sensing signal in a pair, and take the above precautions against noise as well.

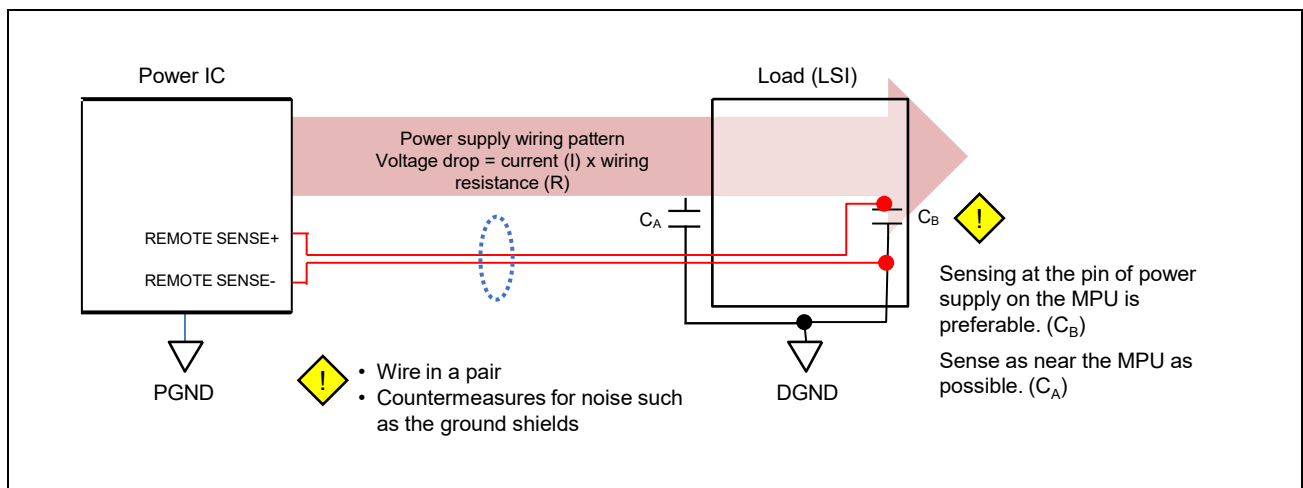


Figure 3.3 Remote Sensing Input of a Power Supply IC

3.10 Supplying Power through Connectors



Points

- When supplying power through a connector, be sure to provide sufficient power supply pins and GND pins for the maximum current value.
- Calculate the amount of current per pin from the allowable current and contact resistance of the connector pin.
- According to the amount of current near the connector, place a capacitor with a relatively large capacitance and generally consider the design in terms of making sure that changes in the power supply voltage due to changes in the current drawn will not become a problem.
- Please refer to the datasheet of the connectors to be used.

3.11 SD

3.11.1 Power Supply Circuit for the SD card Interface



Points

- When power is applied to an SD card, the inrush current may place a heavy load on the source power supply and make the system unstable.
- Depending on the card in use, the current may have large fluctuations. Determine the capacitance of a bypass capacitor for the card power supply based on actual evaluation.
- If the inrush current when the power is turned on is excessively large, you can reduce it by using a high-side switch with a soft start function.
- If the bypass capacitor placed adjacent to the SD card and SD card power connector needs to be discharged quickly when power to the SD card is switched off due to removal of the card, use a high-side switch with a discharge function. If quickly discharging the capacitor is not necessary, discharge it through a resistor with a value of about 1k Ω .
- Please refer to the datasheet of the peripheral devices to be used.

3.11.2 Switching the IO Power Supply Voltage



The MPU supports the 1.8 V and 3.3 V IO power supply voltage for SDHI.

- When dynamically switching the IO power supply voltage, make sure that the voltage does not exceed the specifications.
- In particular, when using a two-input high-side switch, be careful not to have a guard band that prevents the switches from being turned on at the same time during switching.

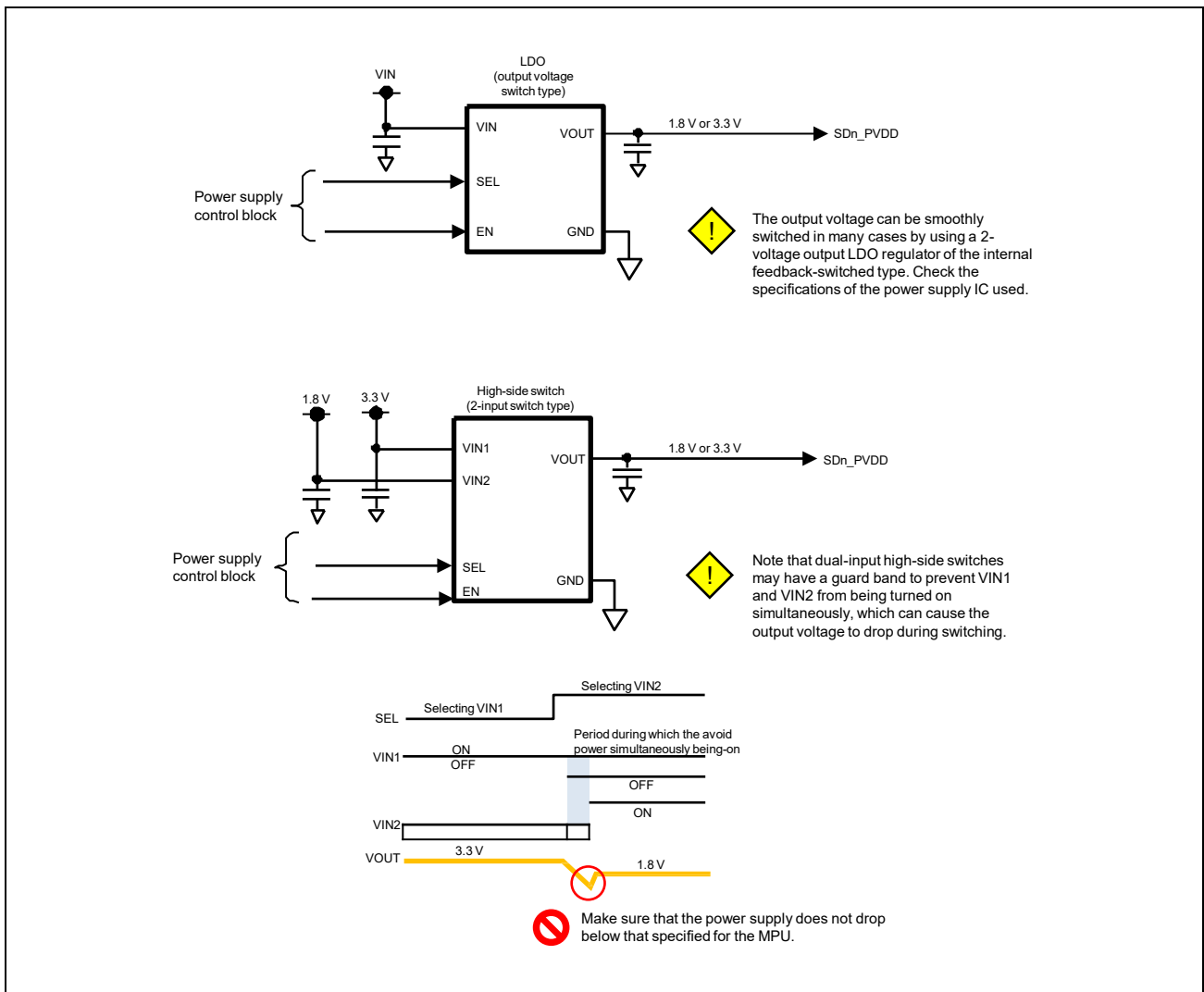


Figure 3.4 Switching the IO Power Supply Voltage of the SDHI

3.12 Reverse Current through USB 2.0 Power Supplies



The reverse current can flow in the following two cases: when the VBUS is powered while the MPU is powered off, and when the USB is in suspend mode. Please take care of them according to your case with the following examples.

A load switch or similar component should be placed between the VBUS and the USB20_VUBUSIN so that 5 V is supplied from the VBUS to the USB20_VUBUSIN after the USB power supplies such as the USB20_USVDD33, the USB20_USVDD18 and USB20_USDVDD are turned on. This prevents the reverse current via the USB20_VUBUSIN from flowing from the power supply terminal of the MPU to the power supply IC during the USB20 power supplies are turned off. (**Figure 3.5**)

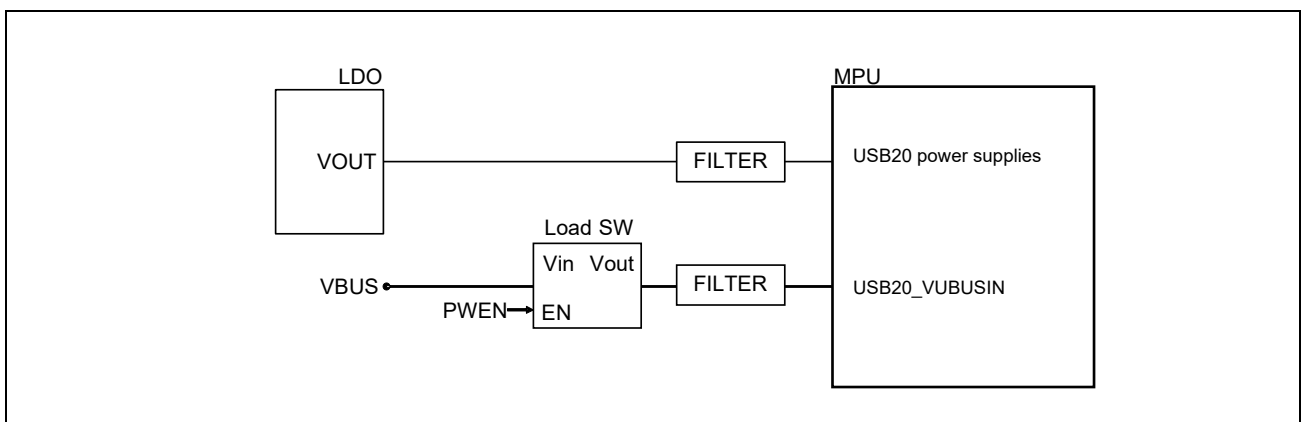


Figure 3.5 Reverse Current from the USB VBUS

If the USB2x_USDVDD 0.8 V power supply is powered by a single power supply and is not shared with other 0.8 V power supplies, a 6.2k Ω resistor should be placed between USB2_USDVDD and GND. This prevents the reverse current from flowing from the power supply terminal of the MPU to the power supply IC during USB suspend. (**Figure 3.6**).

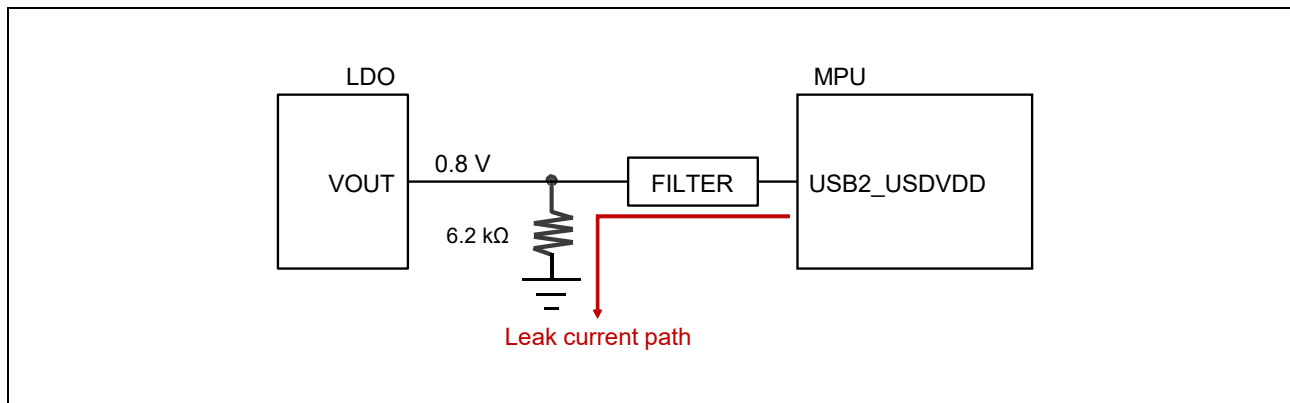


Figure 3.6 Reverse Current from USB2x_USDVDD without Branch

4. High-Speed Interfaces

4.1 Other Guidelines and Documents



Points

We are providing the design guidelines for high-speed signal interfaces (*PCB Design Guidelines* [NDA required]). Be sure to follow the guideline to verify the design.

- LPDDR4/4X interface
- USB 3.2 Gen2x1 interface
- USB2.0 interface
- PCI Express 3.0 interface (PCI Express Gen1/Gen2/Gen3)
- Camera data receiver unit (MIPI CSI-2)
- MIPI DSI interface
- 12-bit A/D converter
- Temperature sensor unit (Power supply)
- PLL (Power supply)
- OSC (Crystal)

Reference Documents:

- IBIS/Spara simulation models for SI/PI simulations (*Interface Design Model* [NDA required])
- PCB verification guide for LPDDR4/LPDDR4X (included in the file above)

Note

For PCB design, it is recommended to utilize these models for SI/PI simulations. Even when reference board design data are employed, variations in characteristics may occur due to factors such as substrate material properties.

4.2 DDR

This chapter describes the points to note regarding DDR design. For detailed design methods, please refer to the following guidelines.

4.2.1 Power Integrity and Signal Integrity Simulations

■ Make sure the DRAM model number is correct

Please make sure that the model number of the DRAM you are installing matches the model number of the DRAM you are using as a model in the pre-design simulation. If the model numbers do not match, the DRAM may not operate properly.

When installing multiple types of DRAMs with different capacities or from different manufacturers on the same PCB, please verify by simulation with all model numbers of DRAMs to be used according to the PCB verification guide. In addition, due to shrinking of silicon dies and changes in internal structure, model parameters may differ if the model number is different even for DRAMs of the same capacity from the same vendor. There is a risk in judging that there is no problem only by observing the waveform on the current PCB. Be sure to obtain a new model from the DRAM vendor and check the compatibility according to the PCB verification guide.

4.2.2 Pin Swapping for LPDDR4/LPDDR4X



The MPU's DDR interface has a signal pin swapping function and has some restrictions.

- Make sure that the following restrictions are met:
 - Address/command signals can be swapped per rank. These signals can also be swapped per channel.
 - Control signals can be swapped per rank. These signals can be swapped per channel.
 - During write leveling, the SoC monitors changes for all data signals. The PCB designers can swap data signals in the same byte lanes such as byte lanes between lane 0 and lane 1 as needed. These signals are swappable per channel.
 - The positive or negative logic of the clock and data strobe signals cannot be swapped. These signals can be swapped per channel.
 - Swapping DQS differential signals are possible as long as it is byte-by-byte. These differential pairs can also be swapped per channel.
 - DMI signals cannot be swapped with DQ signals. These signals can be swapped per channel.
 - The initialization software must contain information about pin swapping.
 - For details, see **section 3.4.3** in the *RZ/V2H (RZ/V2N) Group User's Manual: Hardware*.

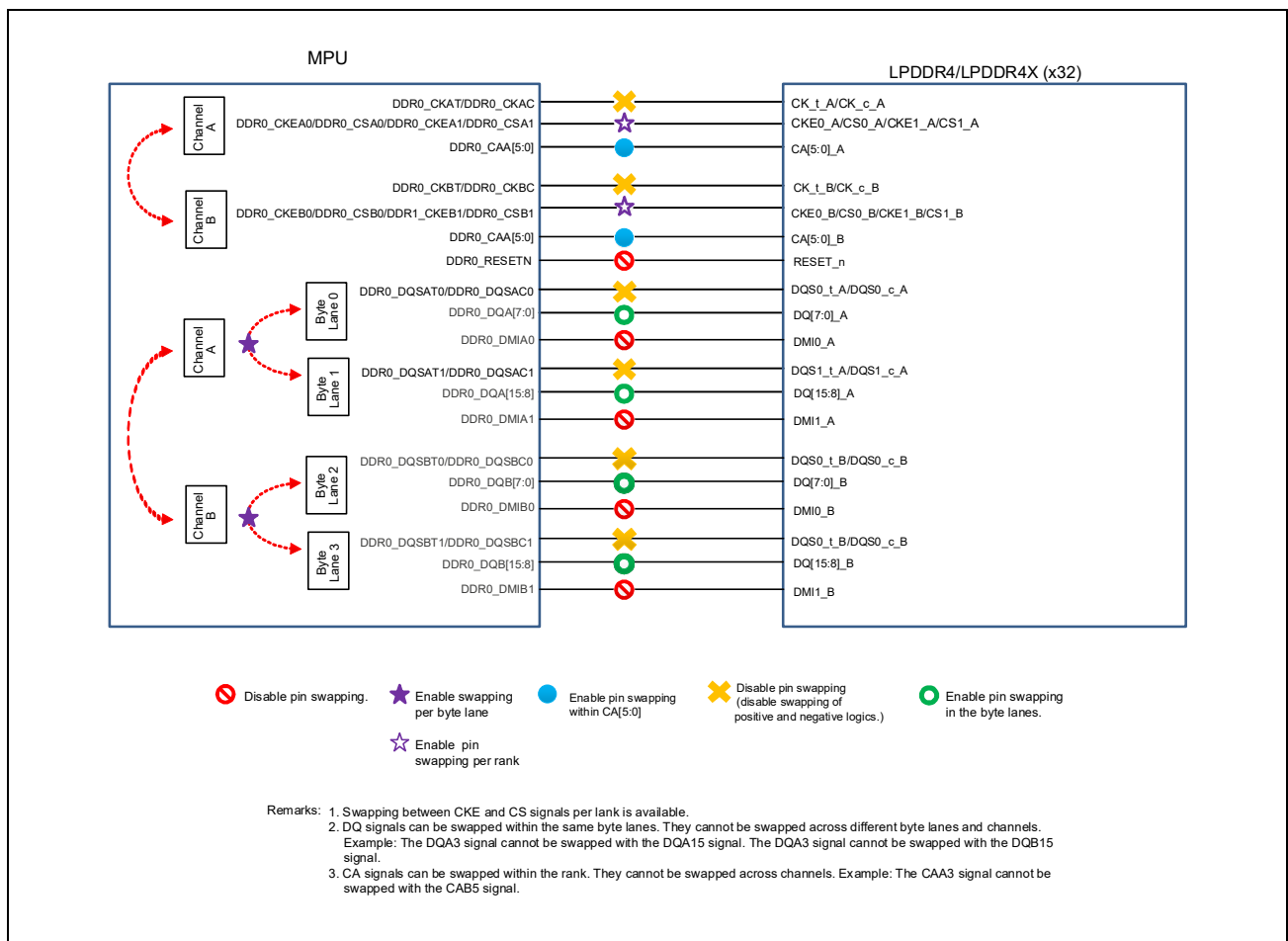


Figure 4.1 Pin Swapping between the MPU and the LPDDR4/LPDDR4X DRAM

4.2.3 Power-on Sequence for LPDDR4



Points

- LPDDR4 SDRAM has VDD1 (typ.1.8 V), VDDQ (typ.1.1 V) and VDD2 (typ.1.1 V) as power supply pins.
- The power-on sequence is defined for each power supply of LPDDR4.
- When LPDDR4 SDRAM is connected to the target device, and both the power supplies are shared with the target device, the power supply must be adjusted for the power supply interface of the LPDDR4 SDRAM.
- Please refer to JEDEC (JESD209-4) for more information on power-on order restrictions.

4.2.4 Power-on Sequence for LPDDR4X



Points

- LPDDR4X SDRAM has VDD1 (typ.1.8 V), VDDQ (typ.0.6 V), and VDD2 (typ.1.1 V) as power supply pins.
- The power-on sequence is defined for each power supply of LPDDR4X.
- When LPDDR4X SDRAM is connected to the target device, and both the power supplies are shared with the target device, the power supply must be adjusted for the power supply interface of the LPDDR4X SDRAM.
- Please refer to JEDEC (JESD209-4-1) for more information on power-on order restrictions.

4.3 Resistor and Capacitor



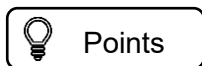
Points

- Follow the guidelines for selecting bypass capacitors. High-speed interfaces have dedicated power pins, and the *PCB Design Guidelines* [NDA required] provide recommended capacitance values for bypass capacitors for power pins.
- As with power supply decoupling, place small-capacity bypass capacitors close to the MPU's power supply pins.
- Resistors used as current references may specify not only resistance values but also accuracy and temperature coefficients, so follow the design guidelines.

4.4 Clock

- The guidelines provide requirements for external clock input, so follow the guidelines for designing. A spread-spectrum clock (SSC) cannot be connected as the external clock.
- Make sure that a clock is not input before the corresponding IO power supplies is turned on.

4.5 Instructions in schematic for PCB Layout



The guideline provides recommendations and restrictions for wiring patterns. We recommend including indications in the circuit diagram to give pattern designers a secure understanding of how recommendations and restrictions in the guidelines have been applied.

For general reference information on PCB pattern design, see **section 6, PCB Layout Design** in this guideline.

4.6 Observing High-Speed Serial Interface Signals



Points

- Note that installation of an observation pad on high-speed serial signal wiring may cause degradation in the signal quality.
 - Unnecessary stubs are constructed to set observation points.
 - Wiring runs that can only be arranged in the inner layer are brought out to the surface layer to set observation points on the surface layer.
 - Setting observation points for differential signals may lead to the symmetry of the differential wiring collapsing.
- The possibility of signal degradation increases with the signal speed, due to the addition of observation circuits. In order to improve the design quality, avoiding the unjustifiable setting of observation points is also necessary.
- High-speed signals might be distorted due to ISI or reflection deriving from observation points. In such cases, ideal waveforms such as are shown in the documents for reference might not be observed.
- In cases such as the above, you must proceed with advanced simulation of the intended observation points and the input circuits of the observation equipment by using the S parameter of the wiring and confirm the suitability by using the observation point movement function of the observation equipment.

4.7 Disturbance Noises and Static Electricity



Points

- For interfaces such as USB2.0, that can be hot-plugged or connected to external devices, insert components for countermeasures against static electricity and external common mode noise.
- Dedicated components that can handle the signal level and signal frequency of each of the interfaces and components with noise filters having an ESD protection function are also available. Select components by reference to the circuit diagram of the evaluation board kits.

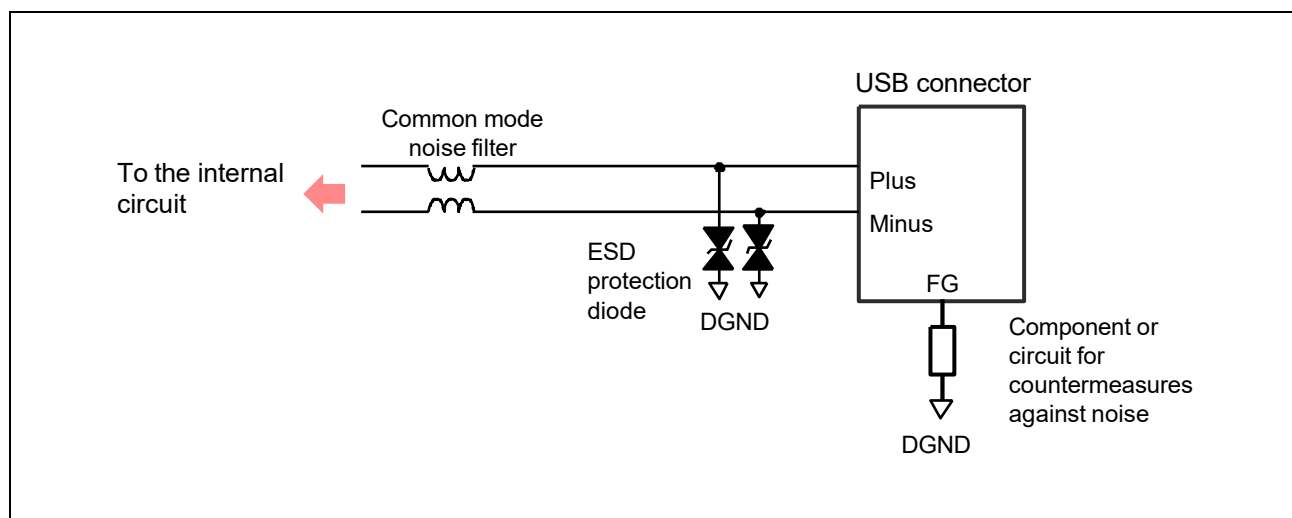


Figure 4.2 Example of Measures Against Disturbance Noises and Static Electricity

5. Others

5.1 Unused Pins



- If there are unused function pins, handle these pins in accord with the statements on the handling of unused pins in the hardware manual.

Documents for Reference

- RZ/V2H Group User's Manual: Hardware
- RZ/V2N Group User's Manual: Hardware

5.2 Countermeasures against Overshoots and Undershoots



Points

- Larger than expected signal overshoots and undershoots must be prevented from causing the device ratings to be exceeded.
- A common method of doing so is to insert damping resistors near the signal outputting ends before determining the necessity and, if one is required, value of the resistor through evaluation. However, in such cases, you must pay attention to the following points.
 - The placement of damping resistors increases the number of parts near the MPU and peripheral devices, significantly affecting the placement of other important parts.
 - The connection of damping resistors necessitates the movement of signals to the surface layer. As a result, via holes and through holes significantly affect the properties of the wiring.
- When inserting a damping resistor to prevent overshooting and undershooting, carefully determine their necessity rather than inserting them unnecessarily.
 - Obtain IBIS models of the buffers, simulate typical transmission paths such as the longest one and shortest one, and estimate the possible amounts of overshoot and undershoot in advance.
 - Reduce the amounts of overshoot and undershoot by bringing the wiring layer (surface layer/inner layer) and wiring width close to the output impedance of the buffer.
 - Check if the driving ability of the output buffers is adjustable or can be set.
 - Apply impedance matching with parallel termination at the receiving ends instead of inserting damping resistors.

Documents for Reference

- Hardware user's manuals for individual target devices
- Lists of pin functions for individual target devices

5.3 Level Shifting



Points

- Some types of IO of the target device such as SD/MMC and xSPI allow two power supply voltages.
- When the signal voltage differs between the MPU and a peripheral device, level shifters should be used and pay attention to the following points.
 - Is the IC to be used voltage tolerant?
 - Does the high-level voltage of signals and the VIH of the IC raise a problem?
 - Does the IC operate at a speed sufficient for the operating frequency of the signals?
- Pay attention to the power supply and output enable of the IC used for level shifting so that the signals do not go to the high level while power for the target device is cut off.
- In order to prevent misuse and misconnection of signals before and after level shifting, we recommend using net names that include signal voltages on the circuit diagram.

Documents for Reference

- Hardware user's manuals for individual target devices
- Lists of pin functions for individual target devices

5.4 IO Characteristics between MPU and Connected Devices



Points

- When connecting the target device and a peripheral device, there will generally be no problem if the CMOS inputs and outputs have the same power supply voltage, even if the connections are direct at the DC level. If the AC timing also raises no problems, the connection will be problem-free. However, the DC specifications might differ slightly from device to device, so we recommended checking the AC and DC margins of both sides.
- The main external factors that may reduce the AC and DC level margins are as follows.
 - Minor differences in power supply voltage, such as where different power supply sources are used, or an FET switch is used
 - Superimposition of ground bounce and signal reflections due to simultaneous switching output (SSO) on rising and falling of the signals
 - Signal rounding due to excessive wiring length or load capacitance
 - Under-driving of the output buffers due to excessive resistive loads
 - Degradation in the signal quality due to crosstalk or ISI
- If there is a possibility that the AC and DC margins for the connected device are insufficient, consider measures such as inserting a pull-up resistor or buffer in the circuit design stage.

Documents for Reference

- Hardware user's manuals for individual target devices
- Circuit diagrams of individual evaluation board kits

5.5 Conflict between Pull-Up and Pull-Down



Points

- Many devices including the target devices might have pull-up and down resistors. Although internal pulling up and down help to reduce the number of parts, conflicts may occur when devices are connected to each other, leading to unexpected problems.
- The following is a common case. Device A has a pull-up resistor and device B has a pull-down resistor. Connecting A and B leads to the signal potential becoming intermediate.

Cases

- An intermediate potential is produced because the internal pulling up and down compete against each other.
- The internal pulling up or down competes with the pulling down or up in an external circuit.
- When a device is turned off, pulling up in another device may apply a voltage exceeding the standard, and the power that should have been turned off does not become 0 V because of external pulling up or a device-protection diode.
- In particular, pay attention to pulling up and down for use in setting the mode of the target device.
- Check the data sheets and technical documentation of each device for details to avoid conflicts.
 - The presence of internal pulling up or down, values of the resistors used, and method of control (such as in terms of always being on, and pin and register settings)
 - Interface equivalent circuit including pulling up or down
 - Behavior in special cases such as during and immediately after a reset, etc.
- The pin function table of the target device describes the information and control of the internal pulling up and down, and the pull-up or -down resistor values are described in the electrical characteristics.

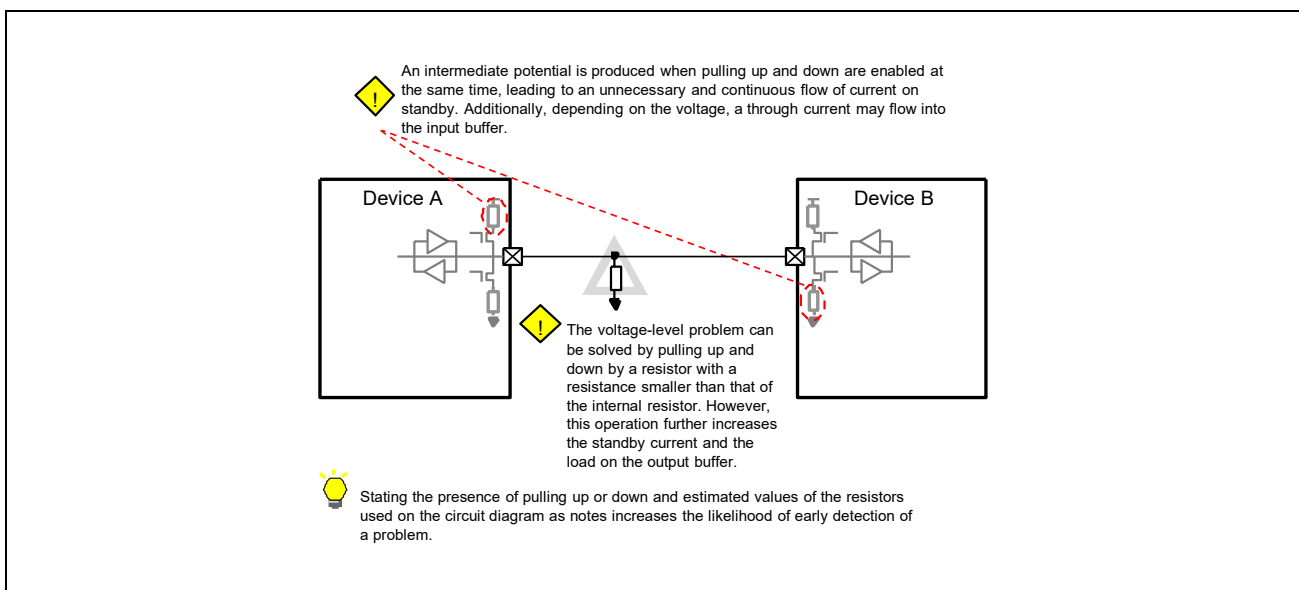


Figure 5.1 Conflict between Internal Pulling Up and Down

5.6 Relatively High-Speed Interfaces (about 10 to 100 MHz)



Points

We provide design guidelines for high-speed signal interfaces such as MIPI CSI-2, MIPI DSI, and USB, but it does not specially provide those for relatively high-speed synchronous parallel interfaces (such as Ethernet, SD/MMC, and xSPI).

- When designing a relatively high-speed interface, we recommend that you observe the following general precautions:
 - Include wiring instructions, such as equal-length wiring, in the schematic diagram to avoid problems in the layout design.
 - Route all wiring on a single layer to prevent delay differences between layers. If possible, calculate the delay from a transmission simulation in advance and instruct the designer to design an equal delay pattern.
 - Leave sufficient wiring spacing to prevent crosstalk. If possible, the wiring spacing should be three times the distance to the reference layer.
- When using a flexible cable to connect to a peripheral device, be sure to provide a return path and provide a sufficient number of ground wiring to prevent crosstalk and unnecessary radiation.

5.7 Notes on Signal Connection



Points

- Make sure that the CMOS input ports are not left open-circuit (Hi-Z).
 - If they are left open-circuit, a through current may continue to flow, leading to deterioration and destruction of elements.
 - Also pay attention to hidden open-circuit states of input ports, such as when a part or IC that is a destination for connection is not mounted.
- Confirm that output signals do not conflict between devices including the MPU.
 - If such conflict continues for a long time, elements may deteriorate or be destroyed.
 - Also, pay attention to hidden conflicts arising from special situations such as during a reset.
- Be careful when connecting ICs that operate under special conditions, such as ICs that operate with the power of an external USB bus even if the main power of the board is not on.
 - Do not drive high or pull up pins that do not have tolerance.
 - In the Renesas evaluation board kits, the USB-UART bridge IC is an LSI chip that partially operates with USB bus power.

Remarks:

Technically, the pins of the target device that are capable of pulling up and down control remain open-circuit from the time the power is turned on until reset is released or until pulling up or down is enabled by the PFC settings, if there is no external component or IC that determines the signal level.

If the reset period is short and the PFC settings proceed quickly after startup, deterioration and destruction due to heat generated by the IO through current will not occur. If you do not need the LSI to perform any processing, we consider turning off the power to be normal. However, if there is a special circumstance, such as when it is necessary to leave the power on or have a longer reset period, use an external device for additional pin handling, although this will increase the number of components.

5.8 Polarity of Electrolytic Capacitors



Points

- Incorrect polarity of the power supply capacitor may cause a serious accident such as generation of toxic gas.
- To make it easy to detect errors in implementation when designing a circuit in addition to paying attention to correct polarity of the power supply capacitor, observe the following.
 - Silk-screen markings to highlight the point
 - Devising the layout and component arrangement for uniform orientation

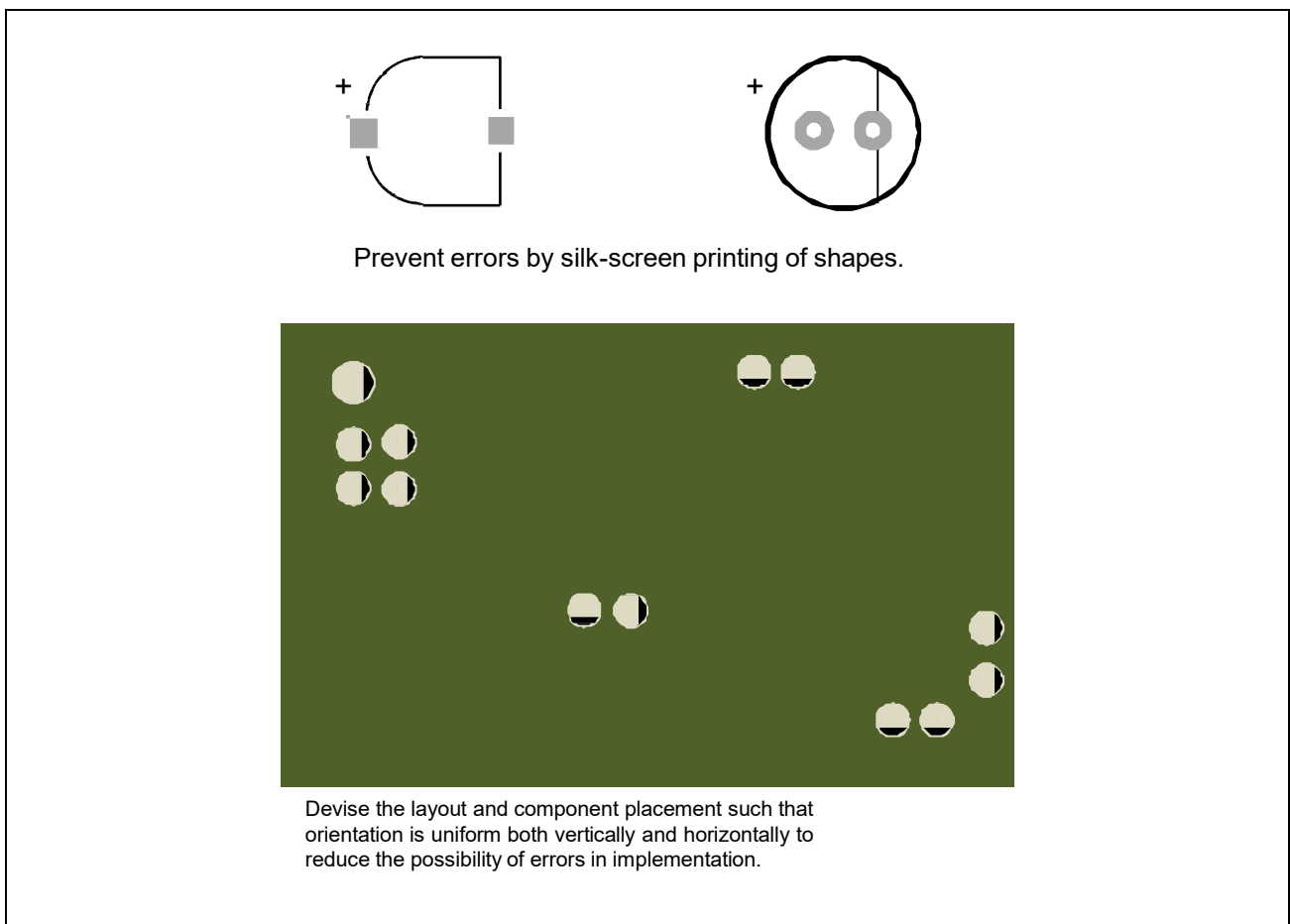


Figure 5.2 Silk-Screen Printing and Orientation of Electrolytic Capacitors

5.9 Programming On-Board EEPROM



Points

- A system configuration in which on-board EEPROM is installed for the writing and reading information such as configurations of the system power supply (PMIC), an FPGA or CPLD, MAC address, and board ID via target devices or a separate on-board controller might be useful in terms of evaluation and debugging.
- Accessing the internal SPI or EEPROM on the I2C bus when the target devices are not powered on violates the absolute maximum rating. In such a situation, the circuit must be configured so that the target devices are isolated and protected.
- In particular, you need to pay attention to devices that are not powered on, including the MPU, when programming an FPGA/CPLD that includes the system power supply (PMIC) and power control circuit.

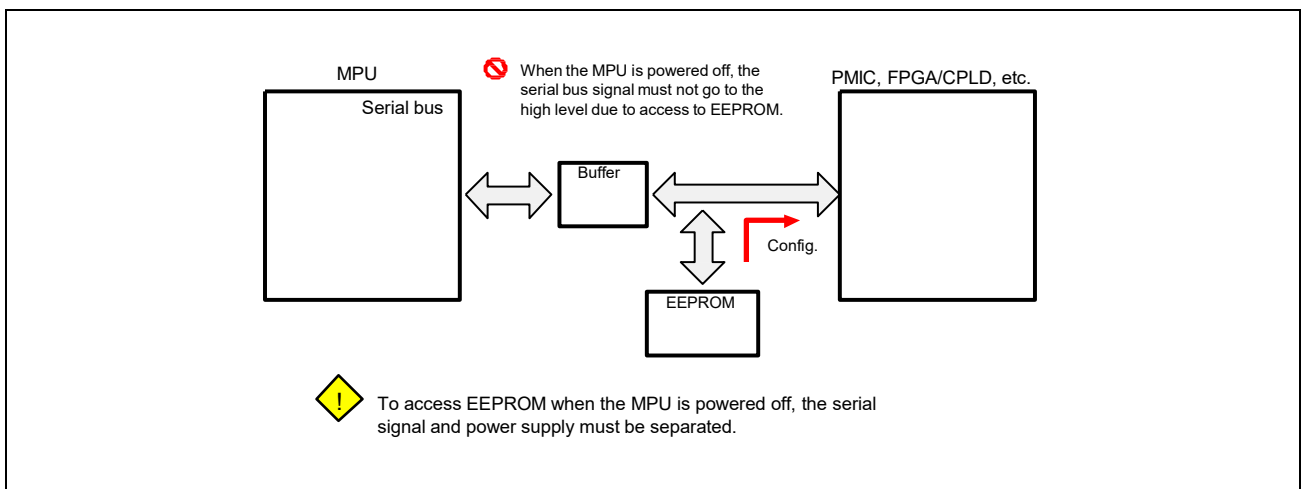


Figure 5.3 Example of a Circuit to Program the On-Board EEPROM

Document for Reference

- Circuit diagrams of individual evaluation board kits

5.10 Board to Board Connector



Note the following points when installing board-to-board connectors for peripheral devices and the expansion of memory.

- Is there any consistency in the connector pin arrangement between the main board and the expansion board?
- Is the connector pin arrangement designed so that noise such as crosstalk is not likely to be applied?
- Is there no problem with the allowable current per connector pin when power is supplied through the connector?
- Do the connectors have a sufficient number of GND pins in terms of the allowable current and reducing noise?
- Is the power supply on the main board designed to use expansion boards?
- Are the values of capacitors on the connector's boundary plane sufficient?
- When power to the target devices and peripheral devices on the main board is off, will the expansion board circuit not produce a high-level output?
- Are measures such as against the reverse insertion of connectors taken?
- Are the heights of the connectors sufficient for those of the components on the main board and expansion boards?
- Are safeguards against the connection of extension boards that might render the board inoperable sufficient?

Documents for Reference

- Circuit diagrams of individual evaluation board kits
- Datasheets or technical documents for the individual connectors and connected devices

5.11 Reconfirm Operations of Re-used Circuits



Points

- Although using circuits designed and evaluated for past hardware as design assets is common practice, assuming that a circuit is proven may delay the detection of defects and problems with the circuit.
- Confirming that points of change such as the conditions applicable to peripheral circuits and changes to the specifications of the ICs used do not raise problems is important, even for a circuit with a proven track record.
 - Have new restrictions not been added?
 - Has the revision number of the device to be used not changed?
 - Are the adapted circuits appropriate compared to application circuits described in the technical documentation?
 - Is the power connection and signal connection inconsistent due to careless copy and paste?

6. PCB Layout Design

6.1 Power Supply

6.1.1 Target Impedance



- Design power supplies to meet target impedance.

Target impedance is shown for some power supplies. The target value is only a desirable value in the pattern design of the power supply and does not guarantee that it will not fail. However, designs that greatly exceed the target values can cause failures. Therefore, it is recommended to design patterns within the target values.

Table 6.1 Guidelines Describing Target Impedance

Power Supply	Related Guideline
Logic power supplies (VDD)	<i>PCB Design Guidelines</i> [NDA required]
DRAM I/O power supplies (DDR_VDDQ)	PCB verification guide for LPDDR4/4X
USB	Spara models [NDA required]
MIPI CSI-2	
MIPI DSI	
PCIe	

6.1.2 Trace Width



Current of several amperes may be drawn through power supply pins. For such power supplies, take care with the widths of wiring patterns between the power supplies and the LSI.

- The width of the power supply trace should be designed based on the expected current load.
- When the wiring patterns for power supplies are such that wiring runs are drawn from a power supply IC to multiple layers through via holes, take the maximum current into account in ensuring a sufficient number of via holes.
- On the target devices, logic power supply pins which draw relatively large amounts of current are arranged around the center of the package. Take care that the wiring patterns for the power supplies are not narrowed or divided into parts due to clearance for via holes to be used in bringing signal lines out from the pins on the periphery of the package. Examine the widths of power supply patterns in terms of the clearance for via holes being removed from the conductor widths.
- Narrow wiring patterns for power supplies lead to the generation of heat on the board itself. The combination of the generation of heat by the MPU itself and narrow patterns leads to further lowering of the power supply voltage due to the DC resistance vs. temperature characteristic of copper. This also makes countermeasures for heat more expensive.
- When a power supply IC has a remote-sensing pin, placing the sensing pin closest to the MPU (the source of the power consumption) is advantageous from the viewpoint of cancelling the voltage-dropping effect of the power supply pattern.

Pattern width:

The per-square resistance of copper foil with a thickness t of 35 μm at 20°C is approx. 0.5m Ω . When such a wiring pattern for a power supply is 20-mm long, the DC resistance of the pattern is approx. 10m Ω . In this case, if a 5-A current is drawn, the voltage drops by approx. 50 mV.

The resistance of copper has a positive temperature coefficient. The temperature of the wiring pattern for a power supply changing from 20°C to 100°C will increase the resistance by approx. 1.35 times.

When $t = 35 \mu\text{m}$, rough estimates of the allowable currents for limiting the rise in temperature to 10°C and 20°C are 1 A and 3 A, respectively, per mm of width.

Via hole:

The thickness of the plating metal of a standard via hole is from 10 μm to 20 μm . A via hole must be suitable for the current that flows through it according to the same considerations as described for pattern width above. A rough estimate of the allowable current is 0.5 A for a via hole 0.5 mm in diameter and 0.3 A for a via hole 0.3 mm in diameter.

When the width of a power supply pattern is reduced to bring signal lines out as shown in the figure at right, examine the width of the power supply pattern with the parts occupied by the via holes and the clearance between them removed.

Ensure sufficiently wide paths for the flow of current from each power supply through countermeasures such as taking the placement of via holes into account, as in the example at right. If doing so is impossible, consider other measures such as using two layers of wiring for the power supply or exchanging the PCB design for a build-up board that has no via holes.

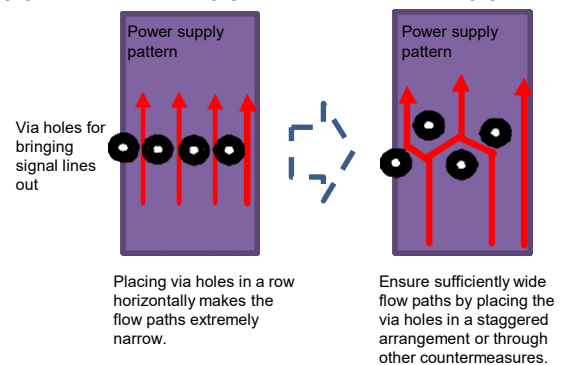


Figure 6.1 Width of Wiring Patterns and Via Holes for Power Supplies

6.2 Exposed Pad



Points

- Some ICs such as power supply ICs have exposed pads to improve heat dissipation.
- The exposed pad may not be defined as an LSI pin in some cases. To avoid forgetting the need for connection to the specified destination such as the ground or a power supply, apply countermeasures such as adding pin names for exposed pads to the circuit diagram.
- The majority of exposed pads are intended for the dissipation of heat. So that the generated heat is propagated to the inner layer along paths with low thermal resistances, apply countermeasures such as using more via holes leading to grounds in the inner layer than would be usual.
- If a recommended wiring pattern for the LSI chip covers the wiring pattern around an exposed pad, follow the recommendation.

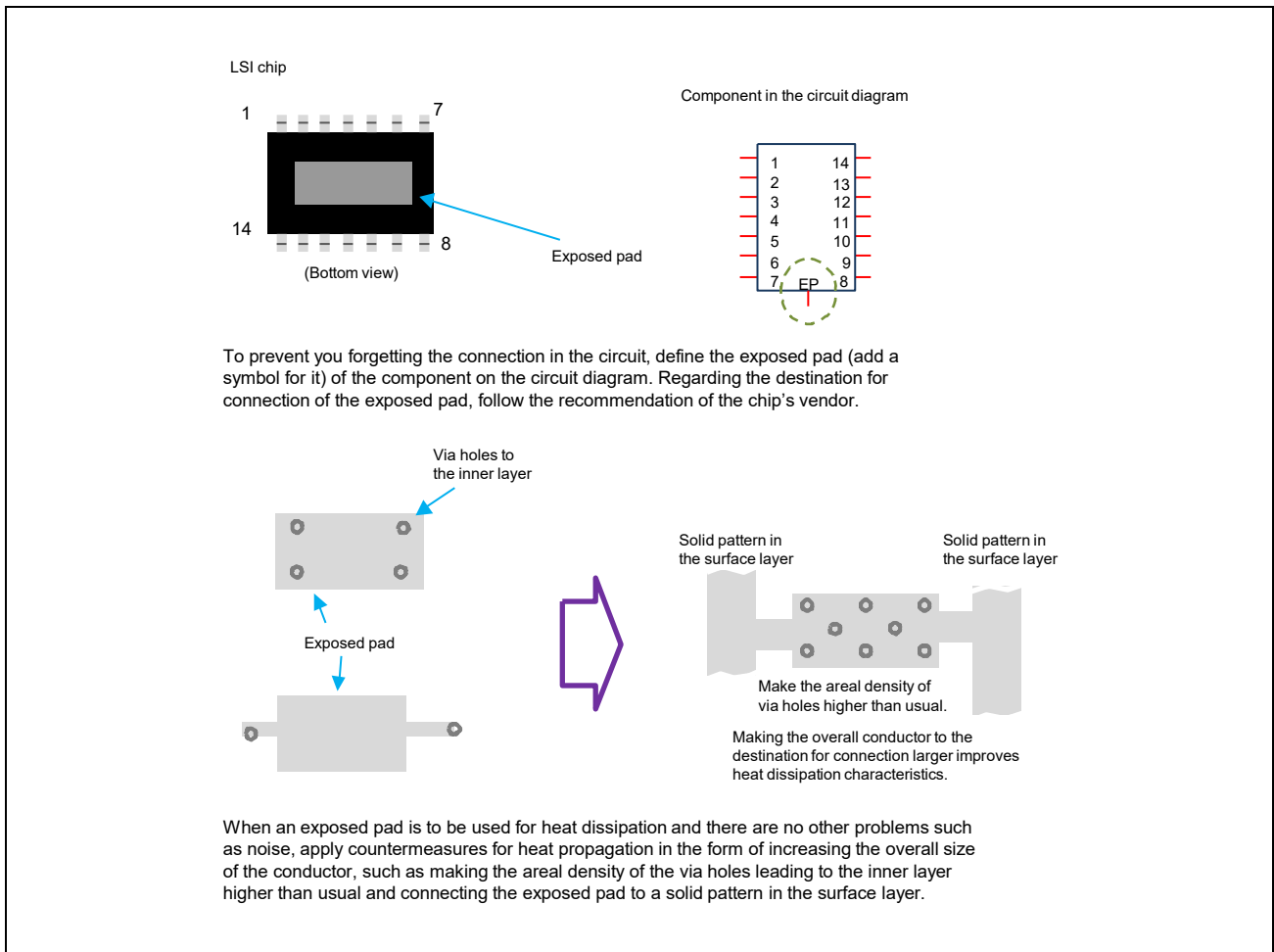


Figure 6.2 Handling Exposed Pads

6.3 Digital and Analog Ground



Points

- Analog grounds are sensitive to noise, so take particular care to avoid the application of noise from digital or signal grounds, which are relatively noisy. This is why separating analog grounds from digital and signal grounds and only connecting them at a single point are recommended in most cases. Avoid the analog ground not being connected or being connected to multiple points in the layout of patterns on a PCB by stating the one-point connection as a note on the circuit diagram.
- As the ground around a switching power supply IC draws a large amount of current in many cases, the ground is usually separated as a power ground. Most power supply IC makers provide a recommended layout of patterns on PCBs. Clearly state that the recommended layout is present on the circuit diagram.
- When connecting a frame ground for USB memory or an SD card, etc. with a digital or signal ground, take appropriate measures such as a one-point connection through noise-suppressing components. If a connector vendor provides a recommended layout of patterns on PCBs, follow the vendor's recommendation.
- When connecting a chassis ground and a PCB, take the application of external noise such as electro-static discharge (ESD) into account in the pattern design. Though the way of connecting a chassis ground and digital or signal ground differs from case to case, including non-connection, ensure the following points: noise must run through paths with relatively low impedance and be earthed; and the above paths do not include digital or signal grounds for the LSI chip that is the main component.

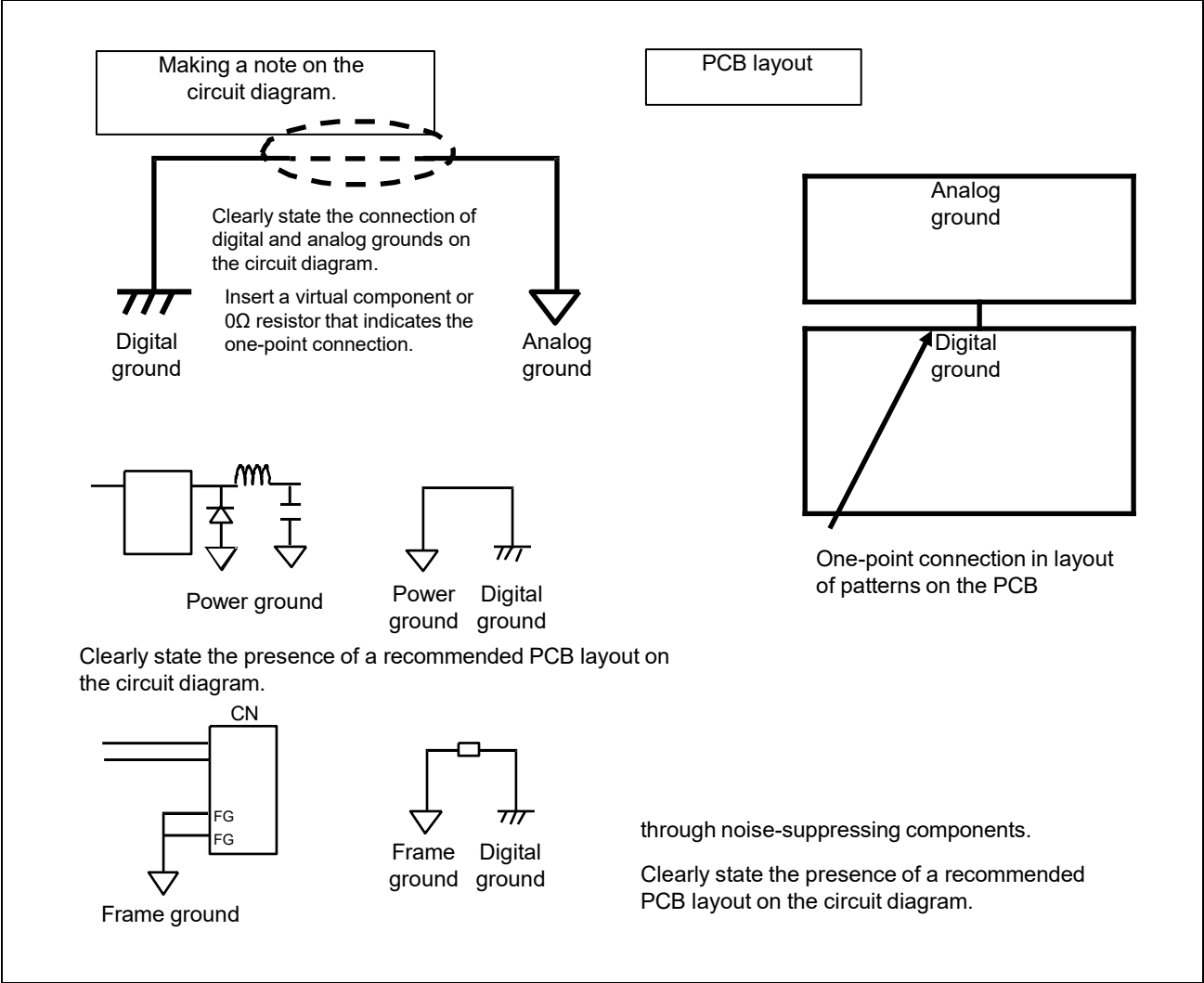


Figure 6.3 Connecting Digital Grounds to Other Grounds

6.4 Ground Shields/Guards



Points

- Applying ground shields to clock, reset, or faint analog signals may be desirable or necessary in many cases.
- Ground shields are only effective if the wiring is appropriate. Inappropriate wiring may lead to the ground shields having the opposite effect to that which was intended.
 - Be sure to connect both sides of a ground shield to the ground plane in the inner layer to prevent their being left open-circuit.
 - When the ground shield wiring runs are long, as well as a single via hole leading to the ground layer from both sides, place a via hole every 5 to 10 mm.
 - The width of the ground shield lines must be at least the same as that of the signal line, and ideally at least as wide as the diameter of the via holes leading to the ground plane.
- Care is also required in the placement of solid patterns of ground in unused areas of the PCB.
 - If this prevents the preparation of via holes leading to the inner ground, do not proceed with the application of a solid pattern of ground.
 - For a large solid pattern of ground, place one via hole leading to the ground layer roughly every 10 mm.
- Especially in the case of applying ground shields to high-speed signals, take care with regard to the uniformity of the shields.
 - When applying a ground shield to a high-speed signal, take care on the following points: do not change the spacing between the signal and ground shield along their paths; do not divide the ground shield into parts due to via holes; and as far as possible, do not allow variation in the wiring widths of a ground shield.
- When applying ground shields to differential wiring, take care with regard to the point that the ground shields being too close to the differential wiring is likely to have a strong effect on the differential combination.
 - In the wiring of a ground shield, its clearance must be at least twice the spacing of the wiring for the differential signals.

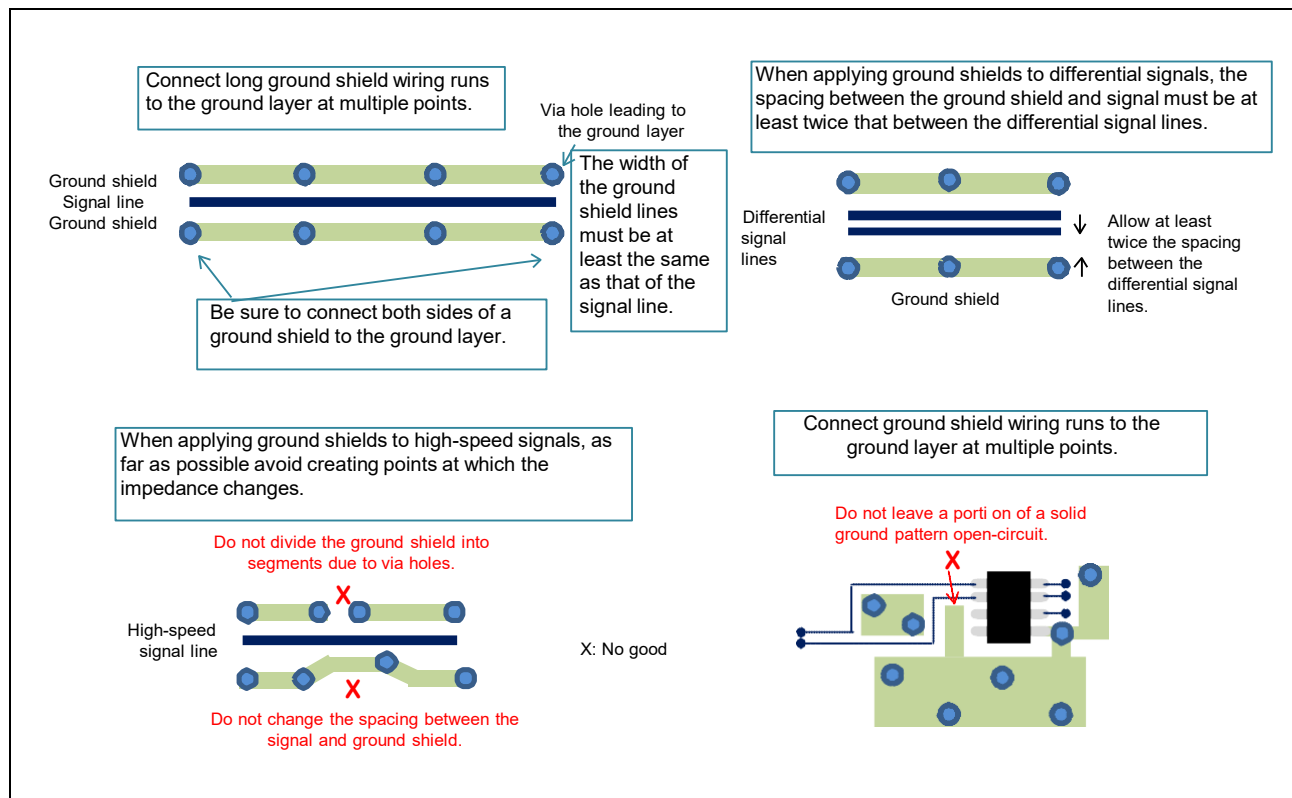


Figure 6.4 Ground Shields/Guards

6.5 Return Path



Points

- A single-ended signal is propagated from a driver to a receiver through signal wiring, and a return current flows from the receiver to the driver at the same time. In such cases, the path along which the return current flows is referred to as a return path. The ground or power supply layer placed closest to the signal wiring serves as the return path in most cases. To retain signal quality and prevent problems before they arise, take the return paths into account in pattern design.
 - In most cases, return paths are just below the signal wiring.
 - The higher the speed of a signal, the more important taking return paths into account in pattern design to retain the signal quality becomes.
 - Take care that arranging via holes to change bus signal wiring runs from one layer to another does not create slits along the return paths.
 - Take care slits in the ground layer or clearance for use with via holes arranged horizontally do not divide the return paths into parts.
 - When a reference layer (ground layer) for use as a return path is changed, place a ground via hole in an appropriate location to avoid a detour from the original return path.
 - If insufficient via holes leading to the reference layer (ground layer) are present near locations where LSI signals are brought out or bus signals are propagated from one layer to another, etc., the flow of the return currents will be concentrated in a particular via hole, leading to the generation of noise such as crosstalk from the return paths. To avoid this, place a ground via hole in every one of several signal wiring runs near locations where signals are propagated from one layer to another.
 - For wiring patterns for circuits that are sensitive to noise such as crystal oscillation circuits, pay attention to the return paths of signals as well as the wiring patterns in the circuit diagram, such as in terms of parallel runs of signal wiring.

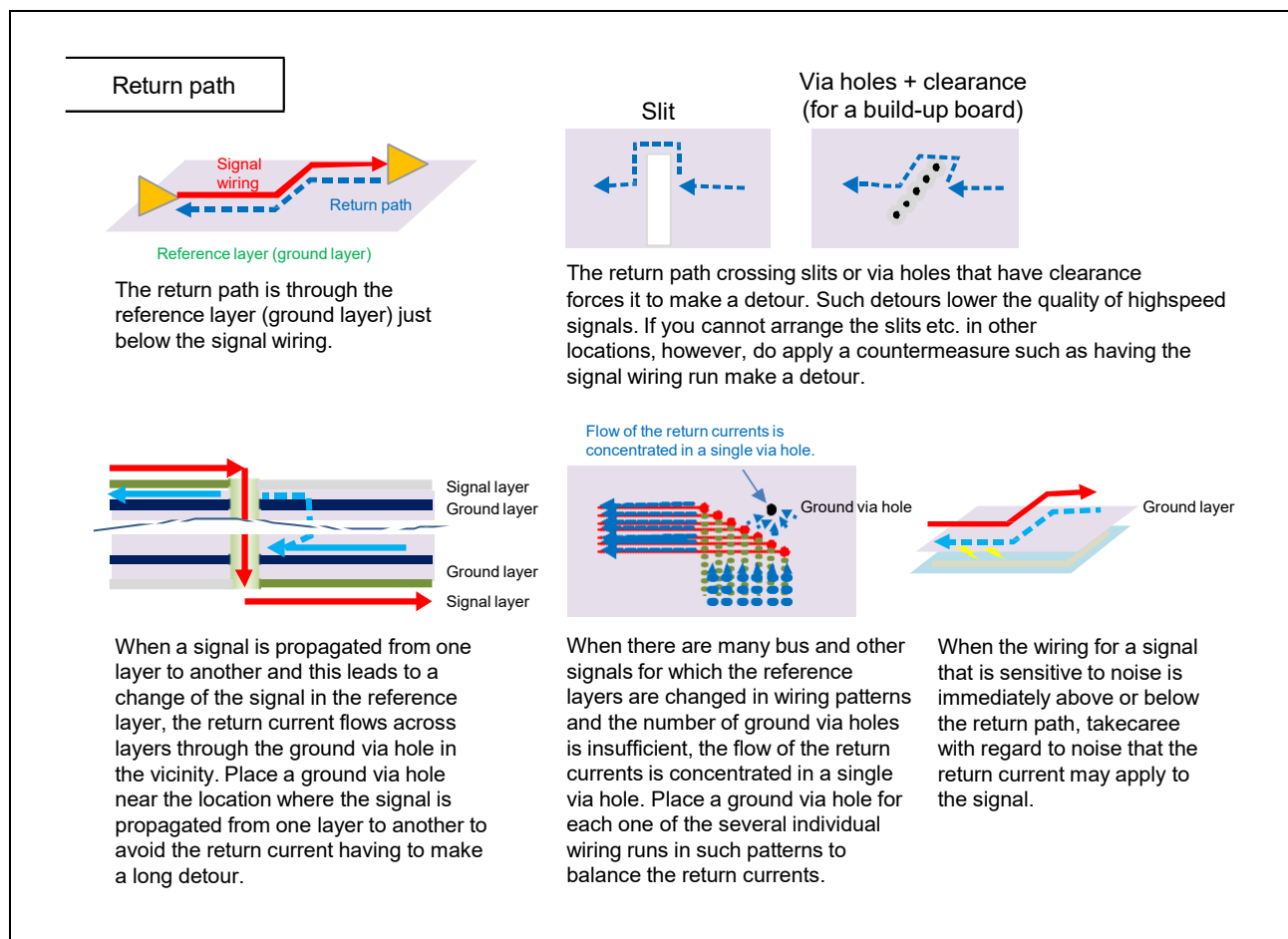


Figure 6.5 Ensuring Return Paths

6.6 High-Speed Interfaces



- Please refer to the guidelines in the following table for high-speed signal interfaces.

Guidelines for board development regarding the DDR, USB, MIPI CSI-2, DSI and PCIe are separately available and listed in the following table. Each guideline covers constraints, instructions, and guidance on the design of the related patterns on a PCB. Refer to these guidelines in pattern design. The guidelines also state reference documents from vendors other than Renesas and recommend referring to them. As these documents may also cover constraints, instructions, and guidance, refer to them together with the guidelines from Renesas.

Table 6.2 Guidelines for High-Speed Signal Interfaces

Interface	Guideline
LPDDR4, LPDDR4X	<i>PCB Design Guidelines</i> [NDA required]
	IBIS / Spara models [NDA required]
	<i>PCB verification guide for LPDDR4/4X</i>
USB	<i>PCB Design Guidelines</i> [NDA required]
MIPI CSI-2 and MIPI DSI	IBIS / Spara models [NDA required]
PCIe	

6.6.1 Differential Signals



Points

- The target devices have several high-speed differential signal interfaces such as the USB modules.
 - In differential interfaces in general, positive and negative signals are paired. This allows reduction of the signal amplitude and increases the immunity to common mode noise, thus allowing for higher-speed transfer.
 - Wiring patterns for pairs of differential signals are required to make the best use of differential transfer.
 - To avoid any problems arising later, consider modelling interface components such as components to be inserted and connectors in advance, and confirming the signal quality such as in terms of the deterioration of open eye patterns through simulation.
1. Wiring of differential pairs
 - Design the wiring patterns for pairs of differential signals while basically maintaining the required isometry and parallelism without changes in the width and spacing of the wiring.
 - As far as possible, that is, unless this presents a difficulty in terms of bringing the signal lines out or inserting any components for use as countermeasures for noise, place the wiring pairs for differential signals in the same layer.
 - When the locations where the signal lines are brought out lead to differences in electrical length, as far as possible adjust the lengths by a measure before the location. Methods of adjustment include changing the direction in which the signals are brought out. Do not use meandering wiring.
 - When a bend in a signal wiring run leads to a difference in electrical length, adjust the lengths to remove the difference by a measure as close as possible to the location where the difference was produced.
 - Place components such as those for use as countermeasures for noise, via holes for use in interlayer routing, and ground shields as close to symmetrically as possible relative to the wiring runs of differential pairs.
 - When you cannot avoid using meandering wiring to adjust the timing relative to that of other signals, ensure enough spacing of the meandering wiring relative to that of the other wiring for the differential pair.
 2. Differential impedance
 - Adjust the differential impedance to be roughly estimated as $\pm 10\%$ relative to the characteristic impedance specific to the given interface unless otherwise specified in the guideline or other documents for the given interface.
 - Remove the ground layer immediately under the wiring for differential pairs as required to adjust the impedance. Take care that the wiring runs of differential pairs do not cross slits in the ground plane for use as the reference layer.
 - When running the wiring for a differential pair from one layer to another, place a ground via hole near the via hole for the differential pair to suppress the generation of uneven impedances.
 3. Taking peripheral wiring patterns into account
 - When ground shields are not to be applied, the wiring for a differential pair and another signal must be separated from each other by at least five times the width of the signal wiring.
 - When ground shields are to be applied, the wiring for a differential pair and for the ground shield must be separated from each other by at least twice the width of the signal wiring.

- When bending signal wiring runs, do not bend them through right angles but by no more than 45° (or at least 135°).
- When you cannot avoid using wiring patterns that lead to deterioration of the signal integrity (SI) of differential signals such as for test points, use as few as is possible.

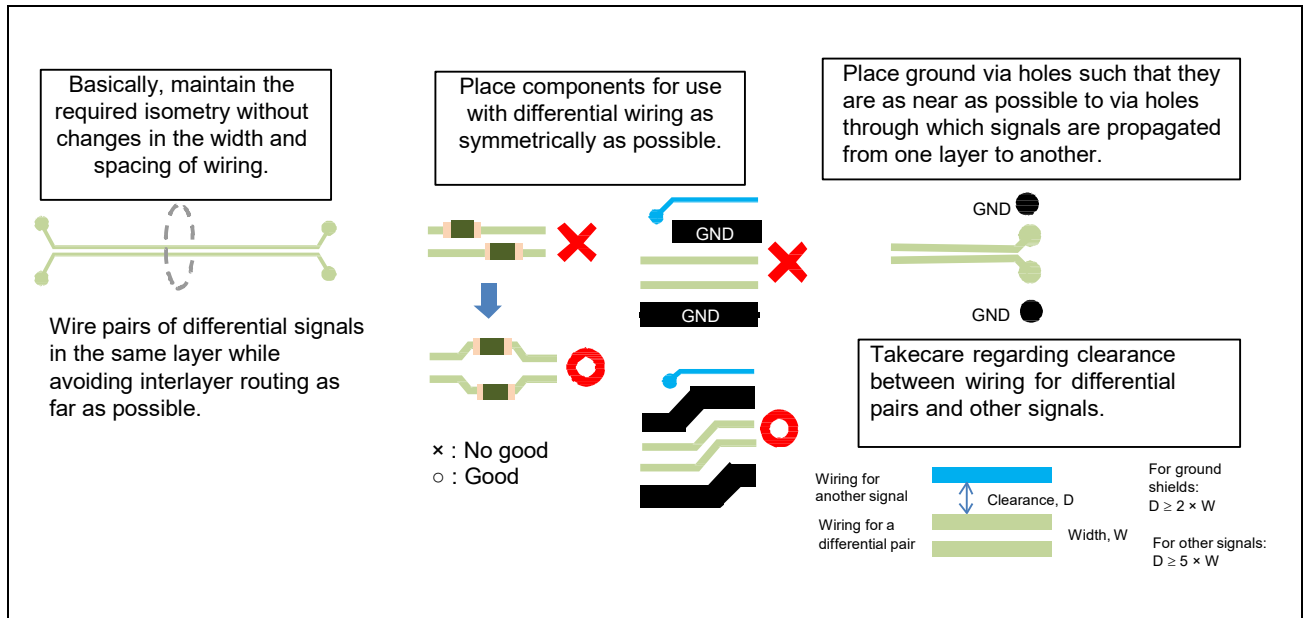


Figure 6.6 Designing Wiring Patterns for Differential Signals

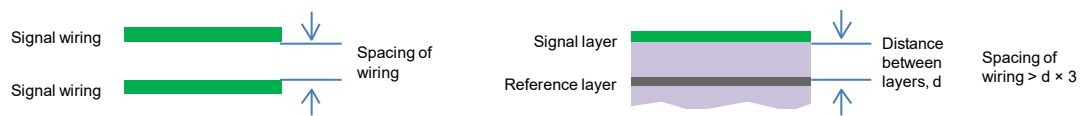
6.6.2 Crosstalk



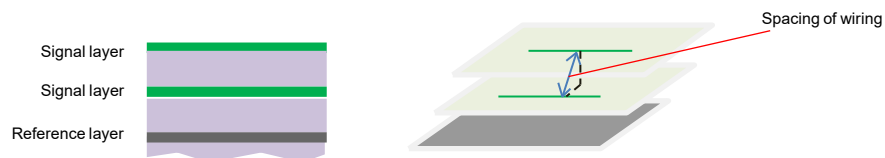
Points

Especially in the case of wiring patterns for high-speed signals, deterioration of signal quality due to crosstalk affects timing design, and may also strongly affect signals that are sensitive to noise such as analog signals. Typical countermeasures for crosstalk are given below.

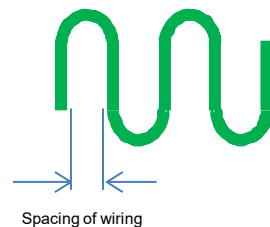
- For wiring patterns for single-ended signals, make the spacing between wiring runs as wide as possible, and take care not to arrange multiple wiring runs in parallel over long stretches.
- When multiple signal wiring runs are arranged in parallel over a long stretch in the case of the routing of buses or otherwise, apply countermeasures for crosstalk such as inserting ground signal wiring once every few wiring runs.
- Also, take care not to arrange wiring runs in parallel over long stretches in layers just above or below the wiring for high-speed signals.
- When using meandering wiring to adjust timing, take care that the meandering wiring itself is arranged correctly in terms of, e.g., the spacing.
- In pattern design where major adjustments of timing are required, verify how crosstalk affects the timing with the resulting wiring patterns through simulation to confirm that the patterns do not create a problem.
- When guidelines define constraints regarding wiring patterns such as spacing, follow the guidelines.
- When adjacent wiring is for signals that are sensitive to noise, ensure sufficient spacing, including for the respective return paths. Moreover, apply countermeasures such as the use of ground guards.



The reference spacing of signal wiring is set according to the distance from the solid ground layer or other layer that is just below the signal layer and serves as the reference layer and to the frequencies of the signals. For wiring patterns for high-speed and clock signals, the standard spacing is at least three times the distance from the reference plane in most cases.



The reference spacing of signal wiring is set according to the distance from the solid ground layer or other layer that is just below the signal layer and serves as the reference layer and to the frequencies of the signals. For wiring patterns for high-speed signals, the standard spacing is at least three times the distance from the reference plane in most cases.



The concept mentioned above also applies to wiring patterns using meandering wiring for timing adjustment. Ensure that spacing between adjacent wiring runs is sufficient.

Figure 6.7 Taking Suppression of Crosstalk into Account in the Design of Wiring Patterns

6.6.3 Pads for Monitoring Signals



Points

When placing pads for monitoring high-speed signals, take care on the following points.

- Place the signal pad on the signal line so that the stub lengths are as short as possible.
- Place a ground pad near the monitoring point if this is possible. When placing a ground pad in a ground shield, take care that slits in the shield and so on do not change the impedance.
- When the target for monitoring is a pair of differential signals, place the components including ground pads for use in monitoring symmetrically relative to the wiring runs of the differential pair.
- For monitoring a high-speed signal, select a monitoring point that suits the aims of the monitoring, such as close to the receiving terminal.
- When you proceed with simulation to check signal integrity (SI), do so with the inclusion of a pad for use in monitoring. Note that the signals monitored from the pad and at the receiving terminal will not perfectly match. The simulation of monitoring waveforms on the pad may be used to extrapolate those at the receiving terminal in later evaluation of the waveforms.

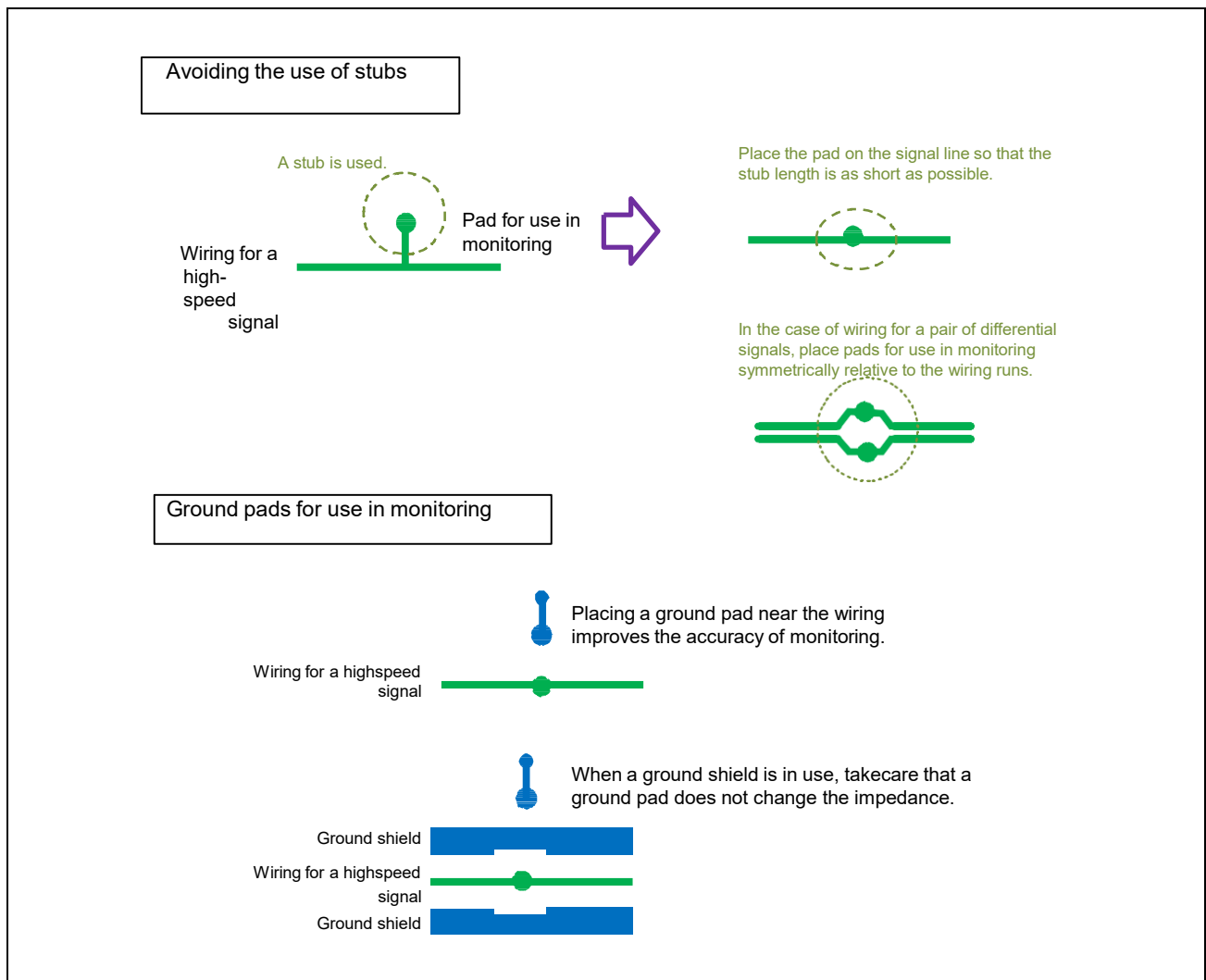


Figure 6.8 Wiring Patterns for Use in Monitoring High-Speed Signals

6.7 General Notes



Points

- General notes on designing wiring patterns for signals are given below.

When the wiring patterns for a circuit can be predicted to have a relatively strong effect on the signal quality, we recommend stating this as a note on the circuit diagram to aid in avoiding wiring patterns that are not as intended.

- For wiring patterns for relatively high-speed signals, avoid the use of stubs or branching of the wiring and make the wiring lengths as short as possible.
- For wiring patterns for clock signals, take care that elements such as reflections due to stubs or branches of the wiring and waveform rounding on rising or falling edges due to long wiring runs do not lead to excessive deterioration in the quality of the clock signals.
- When stubs are used in wiring of circuits for mode settings, external pulling up or pulling down, and filtering, make the stubs as short as possible.
- When bending wiring runs is not avoidable, do not bend them through a right angle but through 45° to avoid sudden changes in the impedance of the wiring. Especially for wiring patterns for high-speed signals, apply countermeasures such as rounding the wiring runs at the corners or using curved wiring to retain the signal quality.
- As far as possible, bring signal lines out near an LSI chip to take reflections of the signals at via holes into account. Moreover, keep the main wiring in the same layer.
- For wiring patterns for parallel synchronous signals such as in routing for a bus, use the same layer and employ isometric wiring.
- When using different layers for wiring patterns for parallel synchronous signals such as in routing for a bus is not avoidable, especially when an inner layer and the surface layer are used, simulate signal propagation and confirm that the delay times are the same on the bus as a whole. Adjust the wiring if this is not the case.
- For a circuit that has a record of unsuccessful operation, we recommend applying countermeasures for improving the monitoring performance in confirming its operation such as mainly using the surface layer for wiring patterns or the setting of test points.
- For wiring patterns for a crystal oscillation circuit, take care on the following points in general:
 - place the wiring patterns such that they are as near as possible to the LSI chip;
 - place the wiring patterns such that they are not too close to other patterns, including the digital ground, in the layer where the wiring patterns are placed and those above and below it;
 - as far as possible, remove the ground layer just below wiring patterns;
 - place the wiring patterns such that they are not too near to wiring for input or output; and apply load capacitance with the connection to the same ground.
- For wiring patterns for a crystal oscillation circuit, if the LSI vendor provides a recommended layout of patterns on a PCB, follow the recommendation.
- When designing PCBs, ensure that heat generation is properly considered. In particular, for small-sized boards, take thermal dissipation into account by increasing the number of GND layers and thermal vias. For the thermal designs, refer to *Thermal Design Guide*.

6.8 Countermeasures for Noise



Points

- The inappropriate placement of components that serve as countermeasures for noise may prevent them being sufficiently effective and may even lead to their having the opposite effect to that which was intended.
- Even if the connection information such as for parallel circuits is the same as that in the net list, take care in terms of the order of connection and the method of connection to other circuits with grounds or power supplies possibly being important.
- State constraints and conditions on pattern design as notes on the circuit diagram as far as possible to avoid wiring patterns that are not as intended.
- Reference documents such as datasheets or design guidelines for the given LSI chip cover the recommended layout of patterns and constraints on the methods of connection. Follow the recommendations and apply the constraints.

Examples

- Place a common mode noise filter or component that serves as a countermeasure for electro-static discharge (ESD) nearest to its potential entry point, that is, the connector for use in connecting an external device.
- Place a damping resistor such that the resistor is as near as possible to the sending terminal, while placing a parallel terminating resistor such that the resistor is as near as possible to the receiving terminal.
- Connect a parallel or differential terminator after a wiring run passes through an input point to the IC to avoid the use of stubs as far as possible.
- Place a current reference resistor adjacent to the LSI chip to avoid lengthening the wiring run.
- Place bypass capacitors with a smaller capacitance nearer the given LSI power supply.
- Connect wiring runs for the reference voltage and remote sensing for a switched-mode power supply through a smoothing capacitor to a location where the ripple generated in the power supply line will be sufficiently small.
- For wiring patterns for filter circuits such as resistor–capacitor (RC) filters, connect the components in the order that provides the greatest effectiveness. Specifically, in a case where an RC filter is used to reduce noise in an IC input, connect the wiring run to the IC input pin through a resistor and capacitor, in that order.
- Connect a filter circuit to ground at one point if doing so makes it more effective.

7. Check List

Table 7.1 Check List

#	Item	Point to Check	Section in the Application Note for Reference		Check
			Section	Sub section	
1	Reset	Check the IO voltage of the QRESN.	1	1.1	
2	Reset	Do not set QRESN high when the MPU is powered off.	1	1.1	
3	Reset	Make sure the QRESN signal is pulled up. We recommend the pull-up resistor no greater than 10kΩ. Determine the pull-up resistor to account for stray capacitance.	1	1.1 1.2	
4	Reset	Also check the reset signals of peripheral devices. Peripherals with a reset terminal with Schmitt trigger IO are recommended.	1	1.1 1.2	
5	Reset	Take care of the noise. Ex. Ground shield etc.	1	1.2	
6	Reset	Note the pull-up/pulldown resistors scattered throughout the schematic.	1	1.2	
7	Reset	Make sure the constraints are met for the reset timing.	1	1.3	
8	Reset for peripherals	When QRESN is used to reset peripheral devices, is the target peripheral device turned on?	1	1.4	
9	Reset for peripherals	Are the voltage and active sense of reset input for the target peripheral device correct?	1	1.4	
10	Reset for peripherals	Is the reset input for the target peripheral device set in consideration of the possible presence of an internal pull-up or pull-down resistor?	1	1.4	
11	Reset for peripherals	Is the driving ability of the reset signal sufficient?	1	1.4	
12	Reset for peripherals	Is QRESN connected to specific peripheral devices for which targeted devices have dedicated reset pins (such as XSPI0_RESETO_N, DDRn_RESETN, and SD0RSTN)?	1	1.4	
13	Reset	When using GPIO as a reset for peripheral devices, make sure that the initial state of GPIO is Hi-Z.	1	1.4	
14	Reset	Make sure that the reset sequence is not a problem by comparing the power up/down timing diagram.	3	3.3	
15	MD pin setting	Did you check whether the connected device has an internal pull-up or pull-down resistor to maintain correct operation when the MD pin signal returns to its primary function?	1	1.5	
16	MD pin setting	Are the external pull-up and pull-down resistance values for setting the MD pin selected in consideration of the internal pull-up and pull-down resistors of targeted devices and peripheral devices?	1	1.5	
17	MD pin setting	Does the peripheral device malfunction due to external pull-up or pull-down for setting the MD pin?	1	1.5	
18	MD pin setting	Is the MD pin setting confirmed to keep the level for 100 ns or less after QRESN = High change?	1	1.5	
19	QXTAL/QEXTAL	Check the frequency. Only 24 MHz is supported.	2	2.1	
20	Audio clock	Is the voltage of the clock input appropriate?	2	2.1	
21	Clock input	Check the requirement of the accuracy and deviation of the clock input.	2	2.2	
22	Clock for High-speed Interface	Is the clock input signal for the high-speed serial interface from a non-SSC type oscillator?	4	4.4	
23	X'tal circuit	Do you include patterns for a damping resistor or feedback resistor that might be necessary for evaluating the oscillation of a crystal resonator?	2	2.3	
24	X'tal circuit	Do the circuit constants for the crystal resonator match the constants of the crystal resonator in use?	2	2.3	

#	Item	Point to Check	Section in the Application Note for Reference		Check
			Section	Sub section	
25	X'tal circuit	Have you considered the notes in the circuit design guideline for the crystal resonator in use?	2	2.3	
26	X'tal circuit	Is the crystal resonator free of oscillation in the unwanted overtone mode?	2	2.3	
27	X'tal circuit	Have you confirmed that the selected crystal resonator manufacturer supports the evaluation of oscillation?	2	2.3	
28	Others related to clock	Did you pay equal attention to the clock input to peripheral devices and targeted devices?	2	2.4	
29	Others related to clock	Did you pay equal attention to the connection of crystal resonators to peripheral devices and targeted devices?	2	2.4	
30	Others related to clock	Did you follow the general notes in the guidelines for the connection of clocks to targeted devices and peripheral devices?	2	2.5	
31	Others related to clock	Did you include instructions on ground shielding, such as in the wiring layers for PCB pattern design, in the circuit diagram for clock connections and clock wiring?	2	2.5	
32	Power supply	Have you checked the power requirements for each power supply and that the power circuits have sufficient supply capacity? We recommend drawing a power tree.	3	3.1	
33	Power supply	In the development stage, it is recommended that the power supply capacity be designed with a margin to allow for the possibility that current consumption may be higher than expected.	3	3.1 3.2	
34	Power supply	In particular, use power supply ICs with sufficient margin for logic power supplies.	3	3.1 3.2	
35	Power supply	When lowering the estimated current for the logic power supply according to a use case, have you confirmed that the use case is reasonable?	3	3.1 3.2	
36	Power supply	Have you placed a sufficient number of 0.1 to 10 μ F capacitors in the power supply circuit to handle sudden surges in the logic power supply?	3	3.2	
37	Power supply	If you are using a switched-mode power supply for the logic power supply, have you confirmed that the switching frequency can follow the variations in load?	3	3.8	
38	Power supply	Does the circuit configuration conform to the power-on and -off sequences?	3	3.3	
39	Power supply	Did you create a time chart of the power-on and -off sequences for the entire power tree, and confirm that there were no problems?	3	3.3	
40	Power supply	Did you create a time chart of special conditions, such as the time of the first power-on, for the power-on and -off sequences of the entire power tree, and confirm that there were no problems?	3	3.3	
41	Power supply	Is there any circuit that may cause damage when the power of targeted devices is being switched on or off, such as pulling up at the power supplies of peripheral devices?	3	3.3	
42	Power supply	Are there enough discharge circuits for power-off to conform to the power supply sequence?	3	3.3	
43	Power line filter	Have you selected and installed filters as recommended in the guidelines for power supply filter circuits to take account of the noise-sensitivity of dedicated power supplies?	3	3.4	
44	Power line filter	Is the noise filter for the CPG PLL power supply one provided in the guidelines?	3	3.4	
45	Power line filter	Have the noise filters for individual power supplies not been merged?	3	3.4	
46	Power line filter	As well as targeted devices, have you selected and installed filter circuits as recommended in the datasheet or technical documentation for peripheral devices?	3	3.4	
47	Power circuit	Did you analyze the impedances of the VDD power supplies by applying the method stated in the guidelines and confirm that there were no problems?	3	3.5	

#	Item	Point to Check	Section in the Application Note for Reference		Check
			Section	Sub section	
48	Power circuit	Did you analyze the impedance of the DRAM IO power supply by applying the method stated in the guidelines and confirm that there were no problems?	3	3.5	
49	Power circuit	Have multiple DRAM IO power supply systems not been merged to act as a single power supply?	3	3.5	
50	Power circuit	Did you analyze the inductance of the dedicated power supply for high-speed serial interfaces and IO power supply by applying the method stated in the guidelines and confirm that there were no problems?	3	3.5	
51	Power circuit	As well as targeted devices, did you analyze the power supply impedances of peripheral devices that came with recommendations on this point, and confirm that there were no problems?	3	3.5	
52	Bypass capacitor	Did you install the appropriate number and capacity of bypass capacitors for the IO power supply?	3	3.6	
53	Bypass capacitor	Did you install bypass capacitors in consideration of simultaneous-switching output noise SSO, such as placing capacitors with large capacitances of about 10 μ F at the end of the IO power supply pattern?	3	3.6	
54	Bypass capacitor	Did you include instructions regarding the PCB pattern in the circuit diagram, such as the placement of capacitors with small capacitances near the device?	3	3.6	
55	Bypass capacitor	Did you consider bypass capacitors for peripheral devices as well as for targeted devices?	3	3.6	
56	Bypass capacitor	Do the bypass capacitors have sufficient capacity in consideration of the decrease in the effective capacitance due to variations in DC bias and temperature?	3	3.7	
57	Bypass capacitor	Is the value of the bypass capacitor for the power supply circuit of the high-speed serial interface as recommended in the guide?	4	4.3	
58	Bypass capacitor	If aluminum electrolytic capacitors are to be used as bypass capacitors, have you taken measures in the silk-screened markings or pattern design to prevent insertion with reverse polarity?	5	5.8	
59	Power-IC circuit	Does the design conform to the recommended circuit provided in the datasheet or technical documentation of the power supply IC in use?	3	3.8	
60	Power-IC circuit	Did you perform PI simulations using the provided S-parameter models?	3	3.8	
61	Power-IC circuit	Do the components for the selected power supply circuit, such as coils, FETs, and diodes, correspond to the power supply specifications?	3	3.8	
62	Power-IC circuit	Did you include typical characteristics regarding the selection of peripheral components for the power supply IC in the circuit diagram?	3	3.8	
63	Power-IC circuit	Does the design take the allowable losses and heat dissipation into account?	3	3.8	
64	Power-IC circuit	Have you confirmed that the phase margin and gain margin of the power supply sufficient?	3	3.8	
65	Power-IC circuit	Did you place the remote sensing pin of the power supply IC in the vicinity of the load? Additionally, did you include instructions on PCB pattern design in the circuit diagram to avoid degradation of the power quality due to noise?	3	3.9	
66	Power pin on CN	Do the numbers of power pins and FGND pins allow margins for the maximum current when power is being supplied via a connector?	5	5.10	
67	Power pin on CN	Is a capacitor with a relatively large capacitance installed near the connector when power is being supplied via a connector?	5	5.10	
68	SD power circuit	Did you take care to prevent the SD card power supply adversely affecting other power supplies when it is turned on or off?	3	3.11	
69	SD power circuit	When the SD card power supply is controlled by the pin of targeted devices, does the configuration take into account the fact that the pin becomes Hi-Z when the power pin on the targeted devices side is initially turned on?	3	3.11	

#	Item	Point to Check	Section in the Application Note for Reference		Check
			Section	Sub section	
70	SD power circuit	Does the bypass capacitor for the SD card interface have a proven capacitance?	3	3.11	
71	SD power circuit	Did you check that the SD card power supply being turned on while the power of targeted devices is off and IO pins are being pulled up does not violate the ratings?	3	3.11	
72	SD power circuit	Is a discharge circuit prepared for the SD card power supply?	3	3.11	
73	SD power circuit	Have you confirmed that no excessive voltage occurs when the voltage is changed in the SDHI IO power supply switching circuit?	3	3.11.2	
74	USB	Is a load switch or similar component placed between the VBUS and the USB20_VUBUSIN so that 5 V is supplied from the VBUS to the USB20_VUBUSIN after the USB power supplies such as the USB2_VDD33 and the USB20_VDD18 are turned on?	3	3.12	
75	LPDDR4/4X	Is the connected DRAM supported by targeted devices?	4	4.2.1	
76	LPDDR4/4X	Did you perform SI verification, timing verification, and PI verification for the connection between targeted devices and DRAM according to the guidelines?	4	4.2.1	
77	LPDDR4/4X	Is the DRAM to be used a product type (model) for which SI verification, timing verification, and PI verification have been completed?	4	4.2.1	
78	LPDDR4/4X	Do the circuits for signal termination and ZQ pin handling follow the recommendations in the guidelines?	4	4.2.1	
79	LPDDR4/4X	Have you confirmed the selection of an appropriate DRAM connection topology?	4	4.2.1	
80	LPDDR4/4X	Have the restrictions described in this guideline been applied to pin swapping of the CA and DQ pins when targeted devices are connected to LPDDR4/4X SDRAM?	4	4.2.2	
81	LPDDR4/4X	Has the power supply sequence of the DRAM been applied when the MPU is connected to the DRAM?	4	4.2.3 4.2.4	
82	High-speed serial IF	Does the design of the high-speed serial interface follow the restrictions and recommendations described in the guidelines?	4	4.1	
83	High-speed serial IF	Did you check the accuracy and the temperature coefficients of the components for connection to the high-speed serial interface?	4	4.1	
84	High-speed serial IF	Did you perform SI simulations using the provided IBIS / S-parameter models?	4	4.1	
85	High-speed serial IF	If you add pads to the signal traces to observe signals, is the circuit designed so that it does not degrade the signal quality?	4	4.6	
86	High-speed serial IF	If the signal observation points are different from the end of the trace, have you run simulations to verify the signal quality and see the difference?	4	4.6	
87	High-speed serial IF	Did you take appropriate measures against signal ESD and noise? Additionally, did you include instructions for the placement of these countermeasure components on the PCB in the circuit diagram?	4	4.7	
88	Others	Is the handling of unused pins of targeted devices in accord with the statements on the handling of unused pins in the hardware manual?	5	5.1	
89	Others	As well as targeted devices, is the handling of unused pins of peripheral devices in accord with the statements on the handling of unused pins in the hardware manual?	5	5.1	
90	Others	Did you provide circuits, such as damping and parallel termination, as countermeasures on circuits for interfaces where signals may undershoot or overshoot?	5	5.2	
91	Others	Did you consider the risk of interfaces where signals may undershoot or overshoot by checking the component specifications and simulating transmission?	5	5.2	

#	Item	Point to Check	Section in the Application Note for Reference		Check
			Section	Sub section	
92	Others	Did you confirm that there is no problem with the voltage of all the signals connected to targeted devices?	5	5.3	
93	Others	Did you confirm sufficient AC and DC margins for the connections between targeted devices and peripheral devices?	5	5.4	
94	Others	Did you provide circuits as countermeasures on connections between targeted devices and peripheral devices for which AC and DC margins may be insufficient?	5	5.4	
95	Others	Did you check all internal pull-up and pull-down resistors of targeted devices and peripheral devices, and apply measures in the form of external circuits to prevent the generation of intermediate potentials due to contention between pulling up and down?	5	5.5	
96	Others	For relatively high-speed parallel IO interfaces such as xSPI and VIN/VOUT, did you include instructions on pattern design in the circuit diagram, such as instructions regarding equal lengths and delays, and stating the wiring layers?	5	5.6	
97	Others	For relatively high-speed parallel IO interfaces such as xSPI and VIN/VOUT, did you confirm that there is no problem with the signal quality by simulating transmission?	5	5.6	
98	Others	When connecting a relatively high-speed parallel IO interfaces such as xSPI and VIN/VOUT through a connector, did you prepare a sufficient number of ground pins to secure the return path?	5	5.6	
99	Others	When targeted devices and peripheral devices are connected, did you confirm that Hi-Z input does not occur even in specific situations such as resets, and that there will be no problems even if Hi-Z input does occur?	5	5.7	
100	Others	When targeted devices and peripheral devices are connected, did you confirm that output signals are not directly connected even in specific situations such as resets?	5	5.7	
101	Others	Is the polarity of the aluminum electrolytic capacitors used in the power supply circuits and peripheral circuits correct?	5	5.8	
102	Others	In a circuit configuration where power is partially turned on to operate the board, such as EEPROM programming, has application of the high level to devices that are not turned on, such as targeted devices, been avoided?	5	5.9	
103	Others	Are the connectors correctly arranged in terms of male and female matching?	5	5.10	
104	Others	Are the pin 1 positions of the connectors correct?	5	5.10	
105	Others	Did you include a pattern diagram in the circuit diagram to specify the positions and arrangements of pin 1 of each of the connectors?	6	5.10	
106	Others	Did you provide fail safes to prevent the reverse insertion of connectors?	5	5.10	
107	Others	Does the circuit that has been created match the revision of targeted devices to be installed?	5	5.11	
108	Others	If the circuit is an adaptation of a previous design, did you review the circuit configuration and the revision of the installed device, and confirm which parts of the circuit raise no problem in being used as is?	5	5.11	
109	Others	Did you understand the contents of this circuit design guideline, including restrictions and recommendations, before designing the PCB patterns?	6	6.1 to 6.8	

REVISION HISTORY		RZ/V2H, RZ/V2N PCB Design Checklist	
Rev.	Date	Description	
		Page	Summary
1.00	Sep. 30, 2025	—	First edition issued

General Precautions in the Handling of Microprocessing Unit and Microcontroller Unit Products

The following usage notes are applicable to all Microprocessing unit and Microcontroller unit products from Renesas. For detailed usage notes on the products covered by this document, refer to the relevant sections of the document as well as any technical updates that have been issued for the products.

1. Precaution against Electrostatic Discharge (ESD)

A strong electrical field, when exposed to a CMOS device, can cause destruction of the gate oxide and ultimately degrade the device operation. Steps must be taken to stop the generation of static electricity as much as possible, and quickly dissipate it when it occurs. Environmental control must be adequate. When it is dry, a humidifier should be used. This is recommended to avoid using insulators that can easily build up static electricity. Semiconductor devices must be stored and transported in an anti-static container, static shielding bag or conductive material. All test and measurement tools including work benches and floors must be grounded. The operator must also be grounded using a wrist strap. Semiconductor devices must not be touched with bare hands. Similar precautions must be taken for printed circuit boards with mounted semiconductor devices.

2. Processing at power-on

The state of the product is undefined at the time when power is supplied. The states of internal circuits in the LSI are indeterminate and the states of register settings and pins are undefined at the time when power is supplied. In a finished product where the reset signal is applied to the external reset pin, the states of pins are not guaranteed from the time when power is supplied until the reset process is completed. In a similar way, the states of pins in a product that is reset by an on-chip power-on reset function are not guaranteed from the time when power is supplied until the power reaches the level at which resetting is specified.

3. Input of signal during power-off state

Do not input signals or an I/O pull-up power supply while the device is powered off. The current injection that results from input of such a signal or I/O pull-up power supply may cause malfunction and the abnormal current that passes in the device at this time may cause degradation of internal elements. Follow the guideline for input signal during power-off state as described in your product documentation.

4. Handling of unused pins

Handle unused pins in accordance with the directions given under handling of unused pins in the manual. The input pins of CMOS products are generally in the high-impedance state. In operation with an unused pin in the open-circuit state, extra electromagnetic noise is induced in the vicinity of the LSI, an associated shoot-through current flows internally, and malfunctions occur due to the false recognition of the pin state as an input signal become possible.

5. Clock signals

After applying a reset, only release the reset line after the operating clock signal becomes stable. When switching the clock signal during program execution, wait until the target clock signal is stabilized. When the clock signal is generated with an external resonator or from an external oscillator during a reset, ensure that the reset line is only released after full stabilization of the clock signal. Additionally, when switching to a clock signal produced with an external resonator or by an external oscillator while program execution is in progress, wait until the target clock signal is stable.

6. Voltage application waveform at input pin

Waveform distortion due to input noise or a reflected wave may cause malfunction. If the input of the CMOS device stays in the area between V_{IL} (Max.) and V_{IH} (Min.) due to noise, for example, the device may malfunction. Take care to prevent chattering noise from entering the device when the input level is fixed, and also in the transition period when the input level passes through the area between V_{IL} (Max.) and V_{IH} (Min.).

7. Prohibition of access to reserved addresses

Access to reserved addresses is prohibited. The reserved addresses are provided for possible future expansion of functions. Do not access these addresses as the correct operation of the LSI is not guaranteed.

8. Differences between products

Before changing from one product to another, for example to a product with a different part number, confirm that the change will not lead to problems. The characteristics of a microprocessing unit or microcontroller unit products in the same group but having a different part number might differ in terms of internal memory capacity, layout pattern, and other factors, which can affect the ranges of electrical characteristics, such as characteristic values, operating margins, immunity to noise, and amount of radiated noise. When changing to a product with a different part number, implement a system-evaluation test for the given product.

Notice

1. Descriptions of circuits, software and other related information in this document are provided only to illustrate the operation of semiconductor products and application examples. You are fully responsible for the incorporation or any other use of the circuits, software, and information in the design of your product or system. Renesas Electronics disclaims any and all liability for any losses and damages incurred by you or third parties arising from the use of these circuits, software, or information.
2. Renesas Electronics hereby expressly disclaims any warranties against and liability for infringement or any other claims involving patents, copyrights, or other intellectual property rights of third parties, by or arising from the use of Renesas Electronics products or technical information described in this document, including but not limited to, the product data, drawings, charts, programs, algorithms, and application examples.
3. No license, express, implied or otherwise, is granted hereby under any patents, copyrights or other intellectual property rights of Renesas Electronics or others.
4. You shall be responsible for determining what licenses are required from any third parties, and obtaining such licenses for the lawful import, export, manufacture, sales, utilization, distribution or other disposal of any products incorporating Renesas Electronics products, if required.
5. You shall not alter, modify, copy, or reverse engineer any Renesas Electronics product, whether in whole or in part. Renesas Electronics disclaims any and all liability for any losses or damages incurred by you or third parties arising from such alteration, modification, copying or reverse engineering.
6. Renesas Electronics products are classified according to the following two quality grades: "Standard" and "High Quality". The intended applications for each Renesas Electronics product depends on the product's quality grade, as indicated below.

"Standard": Computers; office equipment; communications equipment; test and measurement equipment; audio and visual equipment; home electronic appliances; machine tools; personal electronic equipment; industrial robots; etc.

"High Quality": Transportation equipment (automobiles, trains, ships, etc.); traffic control (traffic lights); large-scale communication equipment; key financial terminal systems; safety control equipment; etc.

Unless expressly designated as a high reliability product or a product for harsh environments in a Renesas Electronics data sheet or other Renesas Electronics document, Renesas Electronics products are not intended or authorized for use in products or systems that may pose a direct threat to human life or bodily injury (artificial life support devices or systems; surgical implantations; etc.), or may cause serious property damage (space system; undersea repeaters; nuclear power control systems; aircraft control systems; key plant systems; military equipment; etc.). Renesas Electronics disclaims any and all liability for any damages or losses incurred by you or any third parties arising from the use of any Renesas Electronics product that is inconsistent with any Renesas Electronics data sheet, user's manual or other Renesas Electronics document.

7. No semiconductor product is absolutely secure. Notwithstanding any security measures or features that may be implemented in Renesas Electronics hardware or software products, Renesas Electronics shall have absolutely no liability arising out of any vulnerability or security breach, including but not limited to any unauthorized access to or use of a Renesas Electronics product or a system that uses a Renesas Electronics product. RENASAS ELECTRONICS DOES NOT WARRANT OR GUARANTEE THAT RENASAS ELECTRONICS PRODUCTS, OR ANY SYSTEMS CREATED USING RENASAS ELECTRONICS PRODUCTS WILL BE INVULNERABLE OR FREE FROM CORRUPTION, ATTACK, VIRUSES, INTERFERENCE, HACKING, DATA LOSS OR THEFT, OR OTHER SECURITY INTRUSION ("Vulnerability Issues"). RENASAS ELECTRONICS DISCLAIMS ANY AND ALL RESPONSIBILITY OR LIABILITY ARISING FROM OR RELATED TO ANY VULNERABILITY ISSUES. FURTHERMORE, TO THE EXTENT PERMITTED BY APPLICABLE LAW, RENASAS ELECTRONICS DISCLAIMS ANY AND ALL WARRANTIES, EXPRESS OR IMPLIED, WITH RESPECT TO THIS DOCUMENT AND ANY RELATED OR ACCOMPANYING SOFTWARE OR HARDWARE, INCLUDING BUT NOT LIMITED TO THE IMPLIED WARRANTIES OF MERCHANTABILITY, OR FITNESS FOR A PARTICULAR PURPOSE.
8. When using Renesas Electronics products, refer to the latest product information (data sheets, user's manuals, application notes, "General Notes for Handling and Using Semiconductor Devices" in the reliability handbook, etc.), and ensure that usage conditions are within the ranges specified by Renesas Electronics with respect to maximum ratings, operating power supply voltage range, heat dissipation characteristics, installation, etc. Renesas Electronics disclaims any and all liability for any malfunctions, failure or accident arising out of the use of Renesas Electronics products outside of such specified ranges.
9. Although Renesas Electronics endeavors to improve the quality and reliability of Renesas Electronics products, semiconductor products have specific characteristics, such as the occurrence of failure at a certain rate and malfunctions under certain use conditions. Unless designated as a high reliability product or a product for harsh environments in a Renesas Electronics data sheet or other Renesas Electronics document, Renesas Electronics products are not subject to radiation resistance design. You are responsible for implementing safety measures to guard against the possibility of bodily injury, injury or damage caused by fire, and/or danger to the public in the event of a failure or malfunction of Renesas Electronics products, such as safety design for hardware and software, including but not limited to redundancy, fire control and malfunction prevention, appropriate treatment for aging degradation or any other appropriate measures. Because the evaluation of microcomputer software alone is very difficult and impractical, you are responsible for evaluating the safety of the final products or systems manufactured by you.
10. Please contact a Renesas Electronics sales office for details as to environmental matters such as the environmental compatibility of each Renesas Electronics product. You are responsible for carefully and sufficiently investigating applicable laws and regulations that regulate the inclusion or use of controlled substances, including without limitation, the EU RoHS Directive, and using Renesas Electronics products in compliance with all these applicable laws and regulations. Renesas Electronics disclaims any and all liability for damages or losses occurring as a result of your noncompliance with applicable laws and regulations.
11. Renesas Electronics products and technologies shall not be used for or incorporated into any products or systems whose manufacture, use, or sale is prohibited under any applicable domestic or foreign laws or regulations. You shall comply with any applicable export control laws and regulations promulgated and administered by the governments of any countries asserting jurisdiction over the parties or transactions.
12. It is the responsibility of the buyer or distributor of Renesas Electronics products, or any other party who distributes, disposes of, or otherwise sells or transfers the product to a third party, to notify such third party in advance of the contents and conditions set forth in this document.
13. This document shall not be reprinted, reproduced or duplicated in any form, in whole or in part, without prior written consent of Renesas Electronics.
14. Please contact a Renesas Electronics sales office if you have any questions regarding the information contained in this document or Renesas Electronics products.

(Note1) "Renesas Electronics" as used in this document means Renesas Electronics Corporation and also includes its directly or indirectly controlled subsidiaries.

(Note2) "Renesas Electronics product(s)" means any product developed or manufactured by or for Renesas Electronics.

(Rev.5.0-1 October 2020)

Corporate Headquarters

TOYOSU FORESIA, 3-2-24 Toyosu,
Koto-ku, Tokyo 135-0061, Japan
www.renesas.com

Trademarks

Renesas and the Renesas logo are trademarks of Renesas Electronics Corporation. All trademarks and registered trademarks are the property of their respective owners.

Contact information

For further information on a product, technology, the most up-to-date version of a document, or your nearest sales office, please visit:
www.renesas.com/contact/