Ingenic® JZ4780

Board Design Guide

Revision: 1.1
Date: Sept. 2013

北京君正集成电路股份有限公司
Ingenic Semiconductor Co., Ltd.

Ingenic JZ4780

Board Design Guide

Release history

Date	Revision	Change
Jan 2013	1.0	First release
Sept.2013	1.1	Change the company address and etc

Disclaimer

This documentation is provided for use with Ingenic products. No license to Ingenic property rights is granted. Ingenic assumes no liability, provides no warranty either expressed or implied relating to the usage, or intellectual property right infringement except as provided for by Ingenic Terms and Conditions of Sale.

Ingenic products are not designed for and should not be used in any medical or life sustaining or supporting equipment.

All information in this document should be treated as preliminary. Ingenic may make changes to this document without notice. Anyone relying on this documentation should contact Ingenic for the current documentation and errata.

Ingenic Semiconductor Co., Ltd.

Junzheng Bld, Zhongguancun Software Park 2 Dongbeiwang West Road, Haidian District Beijing,China,100193
Tel: 86-10-56345000
Fax: 86-10-56345001
Http: //www.ingenic.cn

Content

1 Overview 1
1.1 Introduction 1
1.2 Reference Platform 2
2 Platform Stack-Up and Placement 3
2.1 General Design Considerations 3
2.2 Nominal 6-Layer Board Stack-Up 3
2.3 PCB Technology Considerations 4
2.4 8-Layer HDI Board Stack-Up 6
2.5 4-Layer Board Stack-Up 5
3 Static Memory Interface Design Guidelines 7
3.1 Overview 7
3.2 Boot Memory 7
3.3 NAND Flash Connection 8
4 DDR3 SDRAM 9
4.1 Overview 9
4.2 Connection to two 2Gb x 16 DDR3 SDRAM device 9
4.3 Connection to four 1Gb x 8 DDR3 SDRAM device 10
4.4 Layout Guideline 10
5 Audio Codec Design Guidelines 14
5.1 Overview 14
5.2 Audio Power 14
5.3 Headphone Out 14
5.4 Mic In 15
5.5 Speaker 16
5.6 Receiver 16
5.7 Line In 16
5.8 Ditigal Mic 17
5.9 Layout Guideline 17
6 USB and OTG Design Guidelines 19
6.1 USB Overview 19
6.1.1 USB Power 19
6.2 OTG Overview 19
6.2.1 OTG Power 19
6.3 Guidelines for the USB and OTG interface 20

7 LCD 22

8 LVDS 25

8.1 Overview 25

9 HDMI 26

9.1 HDMI Overview 26

9.1.1 HDMI Power 26

10 Camera 27

10.1 Overview 27

11 MSC 28

11.1 Overview 28

11.1.1 MSC Power 28

12 PS/2 and Keyboard 29

12.1 Overview 29

13 SAR A/D Controller 30

13.1 Overview 30

13.2 Touch Screen 30

13.3 Battery Voltage Measurement 31

14 OTP EFUSE 32

14.1 Overview 32

15 Ethernet Design Guidelines 33

15.1 Overview 33

15.2 JZ4780 Ethernet Controller Connection 33

16 RTC 34

16.1 Overview 34

16.2 RTC Clock 34

16.3 Power Control 34

17 Miscellaneous Peripheral Design Guidelines 36

17.1 SSI Design Guideline 36

17.2 UART 37

17.2.1 UART Implementation 37

17.3 I2C BUS 38

17.4 PWM 38

17.5 GPIO 38
17.6 JTAG/Debug Port 38

18 Platform Clock Guidelines 40

19 Platform Power Guidelines 41

19.1 Overview 41
19.2 Power Delivery and Decoupling 41

1 Overview

JZ4780 is a mobile application processor targeting for multimedia rich and mobile devices like tablet computer, smartphone, mobile digital TV, and GPS. This SOC introduces a kind of innovative architecture to fulfill both high performance mobile computing and high quality video decoding requirements addressed by mobile multimedia devices. JZ4780 provides high-speed CPU computing power, good 3D experience and fluent 1080p video replay.

The memory interface supports a variety of memory types that allow flexible design requirements, including glueless connection to SLC NAND flash memory or up to 64-bit ECC MLC/TLC NAND flash memory and toggle NAND flash for cost sensitive applications. It provides the interface to DDR2, DDR3, LPDDR and LPDDR2 memory chips with lower power consumption. JZ4780 also integrates DDR (including DDR2, DDR3, LPDDR and LPDDR2) memory controller, LCD controller (support regular RGB, LVDS and HDMI transmitter), Audio Codec, multi-channel SAR-ADC, AC97/I2S controller, Camera controller, PCM interface, TV encoder, TS interface, MMC/ SD/SDIO host controller, high speed SPI, I2C, PS2 interface, USB2.0 Host, USB OTG, UART, GPIO and so on.

1.1 Introduction

This design guide provides recommendations for system designs based on the JZ4780 processor. Design issues (e.g., thermal considerations) should be addressed using specific design guides or application notes for the processor.

The design guidelines in this document are used to ensure maximum flexibility for board designers while reducing the risk of board related issues. The design information provided in this document falls into two categories:

- **Design Recommendations:** It is based on INGENIC's simulations and lab experience to date are strongly recommended, if not necessary, to meet the timing and signal quality specifications.
- **Design Considerations:** Suggestions for platform design provide one way to meet the design recommendations. Design considerations are based on the reference platforms designed by INGENIC. They should be used as an example, but may not be applicable to particular designs.

Note: In this manual, processor means the JZ4780 processor if not specified.

The guidelines recommended in this manual are based on experience and simulation work completed by INGENIC while developing systems with JZ4780. This work is ongoing, and the recommendations and considerations are subject to change.

Platform schematics can be obtained and are intended as a reference for board designers. While the schematics may cover a specific design, the core schematics remain the same for most platforms. The schematic set provides a reference schematic for each platform component, and common system board options. Additional flexibility is possible through other permutations of these options and components.

The document can help customer span doorstep, design product using existent software and hardware resources. Your advice is the best encourage for us.

1.2 Reference Platform

Figure 1-1 shows the JZ4780 Development Board Architecture.

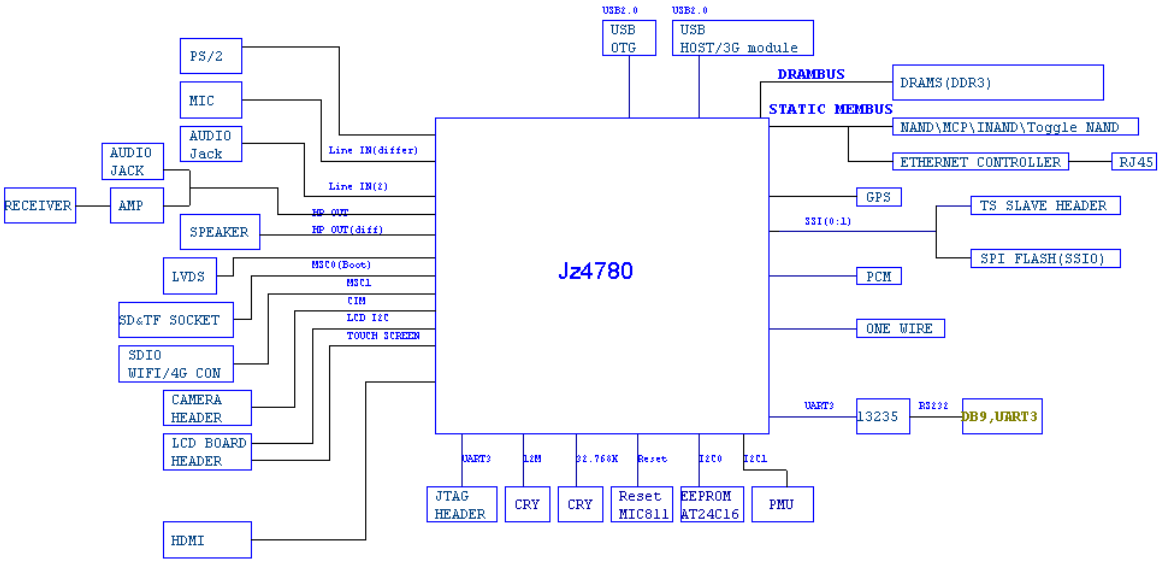


Figure 1-1 JZ4780 Development Board Architecture

2 Platform Stack-Up and Placement

In this section, an example of a JZ4780 platform component placement and stack-up is presented for a PMP product.

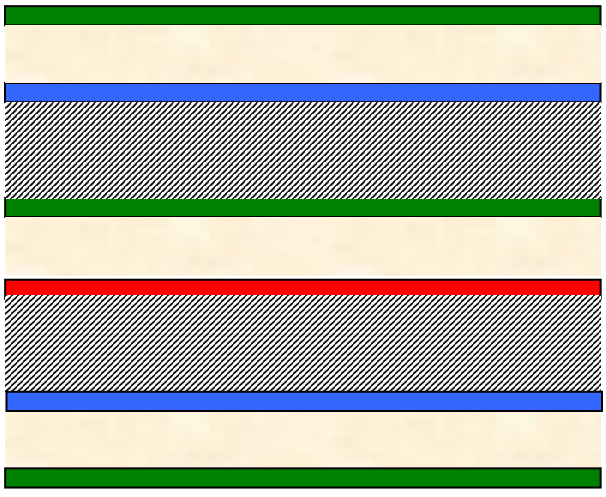
2.1 General Design Considerations

This section describes motherboard layout and routing guidelines for JZ4780 platforms. This section does not describe the function of any bus, or the layout guidelines for an add-in device. If the guidelines listed in this manual are not followed, it is very important that thorough signal integrity and timing simulations are completed for each design. Even when the guidelines are followed, critical signals are recommended to be simulated to ensure proper signal integrity and flight time. Any deviation from the guidelines should be simulated.

The trace impedance typically noted (i.e., 50Ω ± 10%) is the nominal trace impedance for a 4-mil wide trace. That is, the impedance of the trace when not subjected to the fields created by changing current in neighboring traces. When calculating flight times, it is important to consider the minimum and maximum impedance of a trace based on the switching of neighboring traces. Using wider spaces between the traces can minimize this trace-to-trace coupling. In addition, these wider spaces reduce settling time. Coupling between two traces is a function of the coupled length, the distance separating the traces, the signal edge rate, and the degree of mutual capacitance and inductance. To minimize the effects of trace-to-trace coupling, the routing guidelines documented in this section should be followed. Additionally, these routing guidelines are created using a PCB stack-up similar to that illustrated in Figure 2-1.

2.2 Nominal 6-Layer Board Stack-Up

The JZ4780 platform requires a board stack-up yielding a target board impedance of 50 Ω ± 10%. Recommendations in this design guide are based on the following a 6-layer board stack-up:



Signal	Layer 1
Prepreg	Layer 2
Ground	Layer 3
Core	Layer 4
Signal	Layer 5
Prepreg	Layer 6
Power	Layer 7
Core	Layer 8
Ground	Layer 9
Prepreg	Layer 10
Signal	Layer 11
Total Thickness	62 mils.

Figure 2-1 6-layer PCB Stack-Up

Table 2-1 PCB Parameter

Description	Nominal Value	Tolerance	Comments
Board Impedance Z0	50Ω	± 10%	With nominal 4 mil trace width
Dielectric Thickness	4.3 mils	± 0.5 mils	1 x 2116 Pre-Preg
Micro-stripline Er	4.1	± 0.4	@ 100 MHz
Trace Width	4.0 mils	± 0.5 mils	Standard trace
Trace Thickness	2.1 mils	± 0.5 mils	0.5 oz foil + 1.0 oz plate
Soldermask Er	4.0	± 0.5	@ 100 MHz
Soldermask Thickness	1.0 mils	± 0.5 mils	From top of trace

2.3 PCB Technology Considerations

The following recommendation aids in the design of a JZ4780 based platform. Simulations and reference platform are based on the following technology, and we recommend that designers adhere to these guidelines.

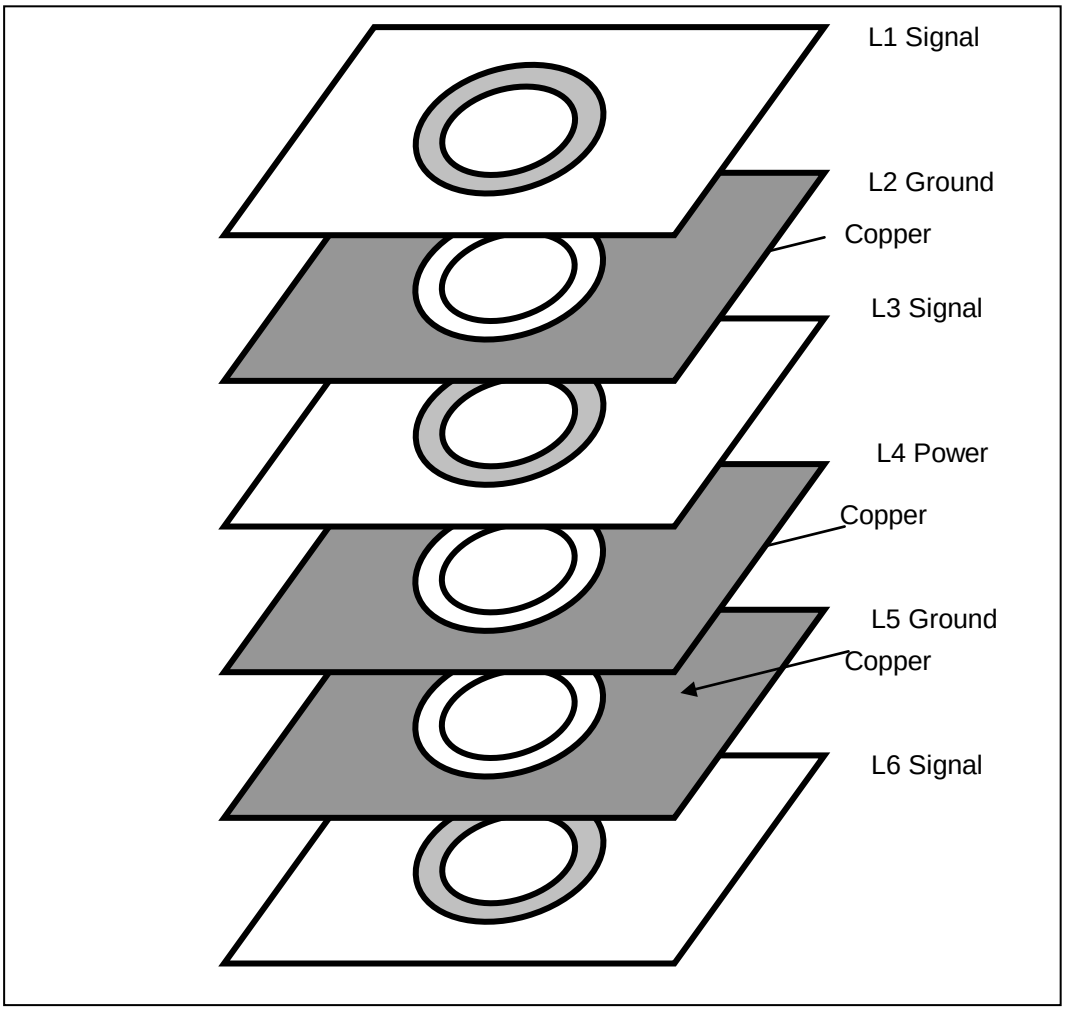


Figure 2-2 PCB Technologies – Stack-Up

Table 2-2 PCB Parameter for Vias

Number of Layers	
Stack Up	6 Layer
Cu Thickness[1]	0.5 oz Outer (before plating); 1 oz inner
Final Board Thickness	62 mils (- 5mils / +8mils)
Material	Fiberglass made of FR4
Signal and Power Via Stack	
Via Pad	16 mils
Via Anti-Pad	20 mils
Via Finished Hole	8 mils

1. The Cu Thickness is just a reference value. It is calculated by the PCB board producers for impedance matching.

2.4 4-Layer Board Stack-Up

The JZ4780 platform requires a board stack-up yielding a target board impedance of 50 Ω ± 10%. If a 4-layer board is used, the stack-up should be:

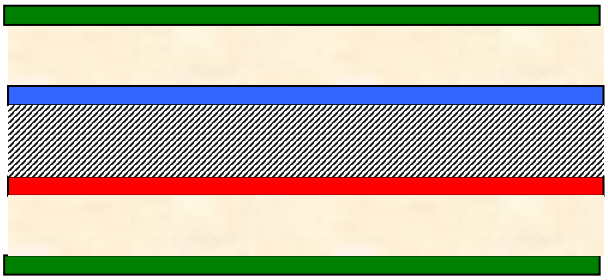


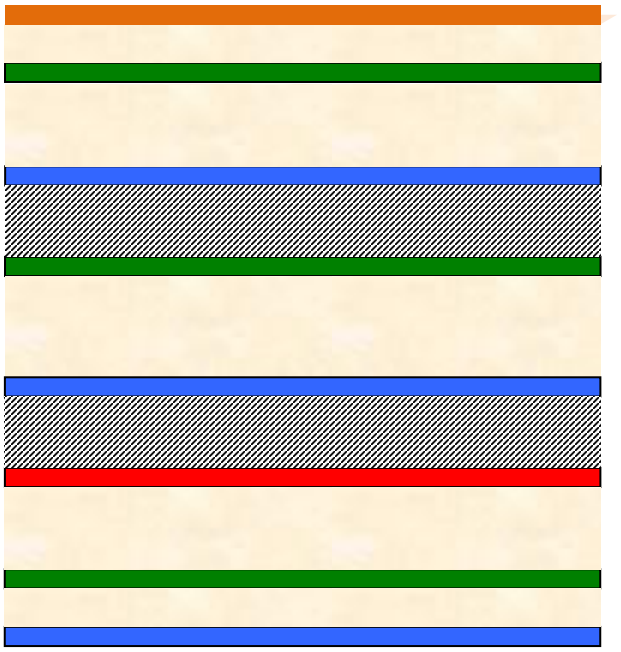
Figure 2-3 4-layer PCB Stack-Up

Table 2-3 PCB Parameter

Description	Nominal Value	Tolerance	Comments
Board Impedance Z0	50Ω	± 10%	With nominal 4 mil trace width
Micro-stripline Er	4.1	± 0.4	@ 100 MHz
Trace Width	4.0 mils	± 0.5 mils	Standard trace
Trace Thickness	2.1 mils	± 0.5 mils	0.5 oz foil + 1.0 oz plate
Soldermask Er	4.0	± 0.5	@ 100 MHz
Soldermask Thickness	1.0 mils	± 0.5 mils	From top of trace

2.5 8-Layer HDI Board Stack-Up

The JZ4780 platform requires a board stack-up yielding a target board impedance of 50 Ω ± 10%. If a 8-layer HDI board is used, the stack-up should be:



Placement	Layer 1
Prepreg	Layer 2
Signal	Layer 3
Prepreg	Layer 4
Ground	Layer 5
Core	Layer 6
Signal	Layer 7
Prepreg	Layer 8
Ground	Layer 9
Core	Layer 10
Power	Layer 11
Prepreg	Layer 12
Signal	Layer 13
Prepreg	Layer 14
Ground	Layer 15
Total Thickness	40 mils.

Figure 2-4 8-layer PCB Stack-Up

Table 2-4 PCB Parameter

Description	Nominal Value	Tolerance	Comments
Board Impedance Z0	50Ω	± 10%	With nominal 4 mil trace width
Micro-stripline Er	4.1	± 0.4	@ 100 MHz
Trace Width	4.0 mils	± 0.5 mils	Standard trace
Trace Thickness	2.1 mils	± 0.5 mils	0.5 oz foil + 1.0 oz plate
Soldermask Er	4.0	± 0.5	@ 100 MHz
Soldermask Thickness	1.0 mils	± 0.5 mils	From top of trace

3 Static Memory Interface Design Guidelines

3.1 Overview

The External NAND Memory Controller (NEMC) divides the off-chip memory space and outputs control signals complying with specifications of various types of static memory and bus interfaces. It enables the connection of static memory such as conventional NAND flash memory, Toggle NAND flash memory, etc. to this processor.

This section is the design guidelines for the external memory interface.

The static memory controller provides a glueless interface to NOR Flash ,NAND Flash and Toggle NAND flash. It supports 6 chips selection CS6~CS1 and each bank can be configured separately. JZ4780 supports most types of NAND flashes, including SLC and MLC/TLC, 8-bit data access, 512B/2K/4K/8KB page size. It also support boot from NAND flash.

3.2 Boot Memory

BOOT_SEL[2:0] pins define the boot time configurations as listed in the following table.

Table 3-1 Boot Configuration

BOOT_SEL2	BOOT_SEL1	BOOT_SEL0	Boot From
1	1	1	usb boot
1	0	0	msc1 boot
1	0	1	msc0 boot
0	1	1	emmc boot
1	1	0	nand boot
0	0	0	spi boot
0	0	1	Reserves
0	1	0	nor boot (CS2)

The boot procedure is showed in the following flow chart:

- In case of NAND/SDcard/iNAND/SPI boot, if it fails, enter MSC1 and USB boot.
- In case of USB boot, if it cannot connect to USB host within 10 seconds, restart the boot procedure.
- In case of NOR boot, if it fails, restart the boot procedure.
- If the boot procedure has been repeated more than 3 times, enter hibernating mode.

3.3 NAND Flash Connection

It supports on CS[6:1], sharing with static memory bank6~bank1.

The following Figure 3-1 is an example of 8-bit NAND Flash Interconnection, Figure3-2 is an example of 8-bit Toggle NAND Flash Interconnection.

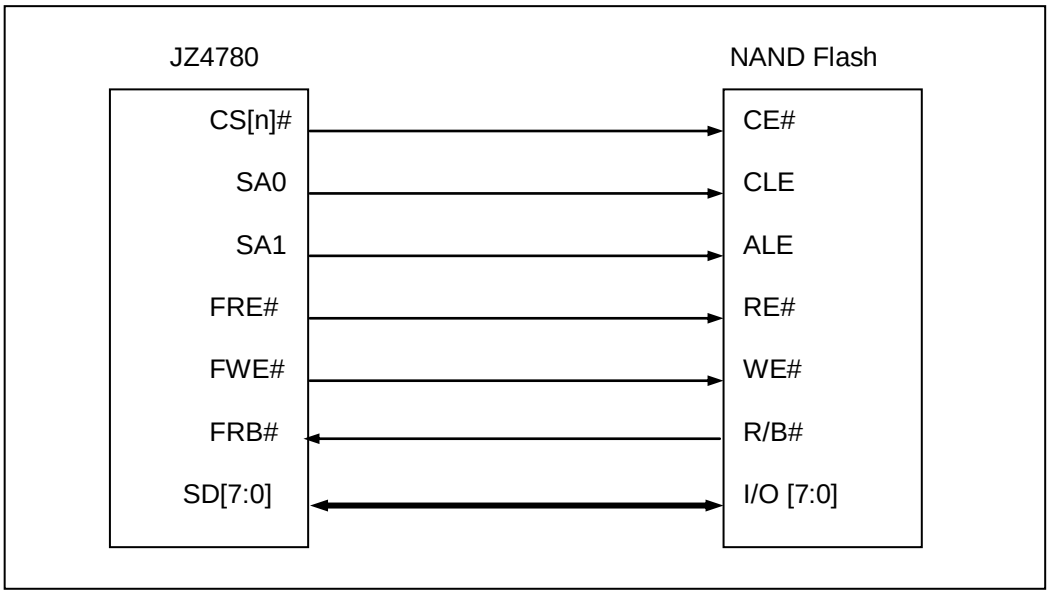


Figure 3-1 8-bit NAND Flash Interconnection Example

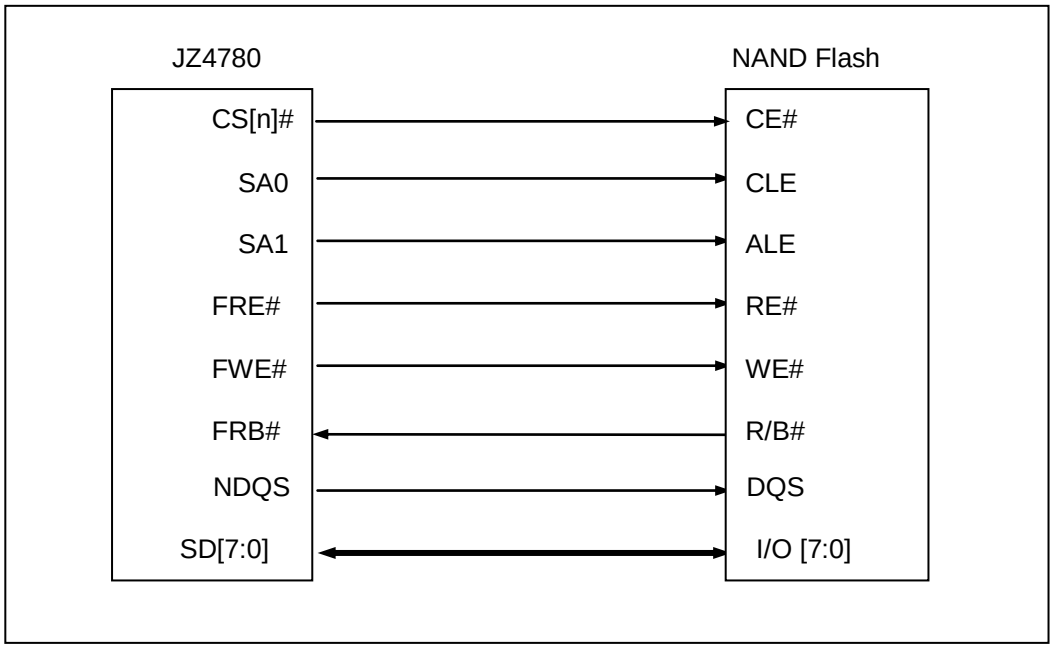


Figure 3-2 8-bit Toggle NAND Flash Interconnection Example

4 DDR3 SDRAM

4.1 Overview

JZ4780 contain a DDR Controller which is a general IP that provide an interface to DDR2, DDR3, LPDDR and LPDDR2 memory. The following figures give examples on the connection to external DDR3 SDRAM devices.

4.2 Connection to two 2Gb x 16 DDR3 SDRAM device

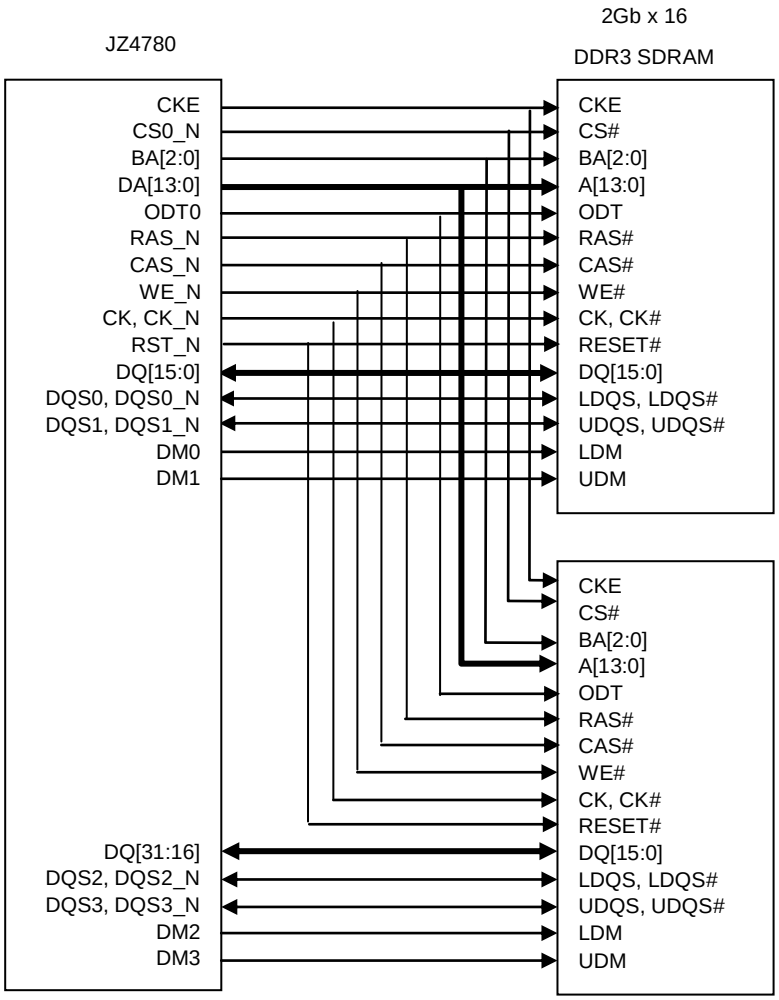


Figure 4-1 Two 16-bit DDR3 Interconnection Example

4.3 Connection to four 1Gb x 8 DDR3 SDRAM device

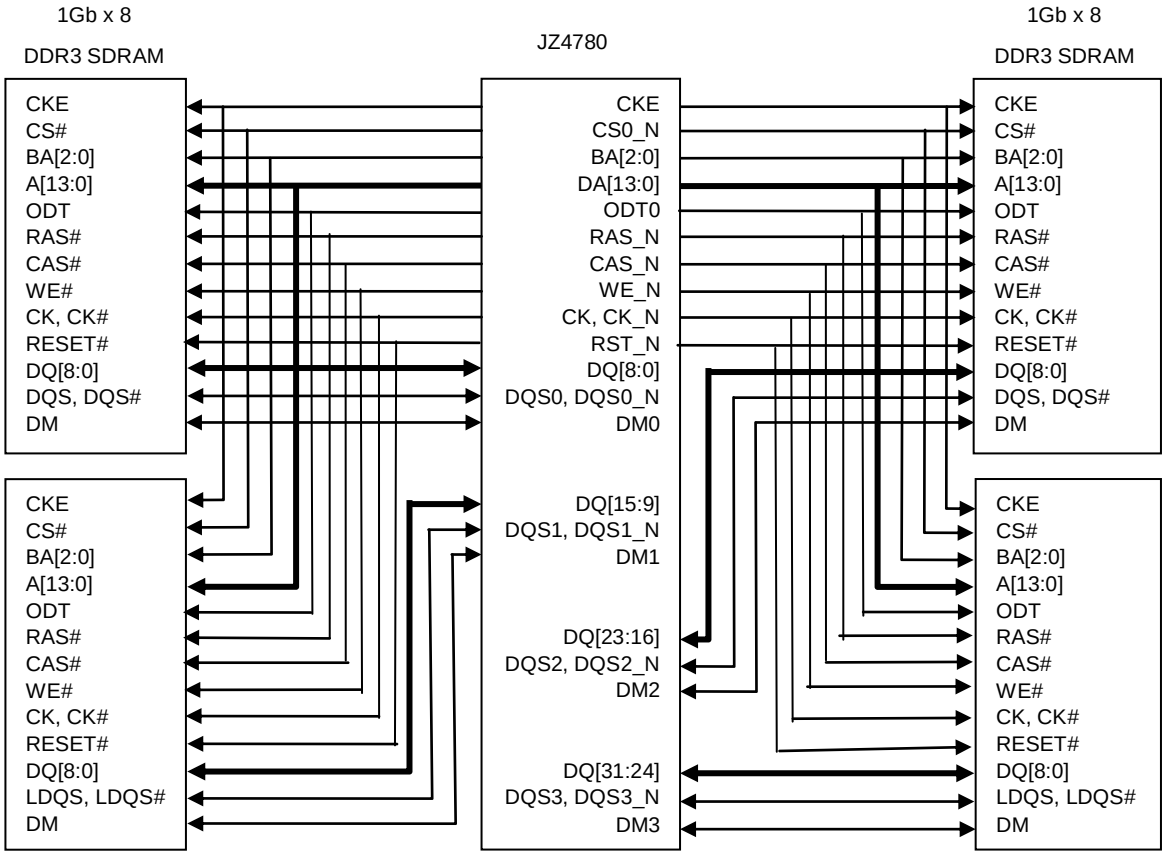


Figure 4-2 Four 8-bit DDR3 Interconnection Example

4.4 Layout Guideline

In the classical high-speed flow, to ensure the maximum performance of the DDR3, we should observe the following guidelines. The questions we should be noticed are: Flight time delay and skew, Signal integrity and impedance matching, Crosstalk, Power supply bypassing.

The basic recommendations are as follows:

- The minimum Stack-up required four layer stack. There must have a ground layer to separated two signal layers. Just as describes in Figure 2-3.
- Signals should be routed based on the relative tightness of the skew budgets. In order of priority:
  1) The double data rate signals, DQ, DM, and DQS/DQS# should be routed first since these have the strictest budgets, ¼ of a clock period available for set up or hold relative to the differential strobe.
  2) Differential clock, CK/CK# and single data rate signals, Address/Command/Control. These have looser budgets with ½ of a clock period for set up and hold.
  3) If read and write leveling techniques are not used, make sure that the rising edges of all differential DQS/DQS# signals are within ¼ of a clock period of the rising edge of the differential clock, CK/CK#.
  4) Route all Vref and support signals (JTAG etc. if implemented)

- The fundamental high-speed PCB issues are flight time delay and skew. Controlling the maximum placement of components. All of the shorter nets in a clock domain must be match the longest one. Therefore, flight time delay and skew are controlled by the matching of the trace.

  Assumptions are 151 ps./inch for top layer microstrip (air in cross section) and 179 ps./inch for stripline or embedded microstrip (no air in cross section). The below table is the recommended budgets.

Table 4-1 The Recommended Budgets

Skew Control Recommendations for DDR Interfaces.				
	Bit Rate	@ 400 Mbps	@ 800 Mbps	@ 1600 Mbps
DQ to DQS Domain	Skew in ps.	50	25	10
	Skew in Inches of Microstrip	0.33	0.17	0.07
	Skew in Inches of Stripline	0.28	0.14	0.06
Addr/Cmd to CK/CK# Domain	Skew in ps.	100	50	25
	Skew in Inches of Microstrip	0.67	0.33	0.17
	Skew in Inches of Stripline	0.56	0.28	0.14
DQS to CK	Skew in ps.	375	188	94
	Skew in Inches of Microstrip	2.50	1.25	0.63
	Skew in Inches of Stripline	2.08	1.04	0.52

- Signal integrity refers to controlling overshoot, ring back, and transition edges. These issues are caused by the mismatch of impedance. Trace impedance is governed by the trace width as well as the thickness and dielectric constant of the PCB insulating materials (usually FR-4). So you should keep the impedance average in a trace, be sure the bending and via as little as possible.

  When the signal has a via in it's trace, there must be a GND via beside the signal via.
- Crosstalk is fundamentally controlled by the PCB stack-up and minimum trace spacing. The best approach to avoiding a crosstalk problem is to ensure all the signals have high-quality signal return paths and to spread the signal out.
  - Each signal layer should have a nearby full ground plane to provide the shortest return current path. In order to maintain consistent characteristic impedance, it is important that the traces be routed over solid ground planes(Figure 4-3) not separate (Figure 4-4). A high speed signal trace should never be routed across a plane split. This will interrupt the return currents that flow beneath the conductor and can lead to crosstalk with neighboring traces. This will also increase emissions from the board.

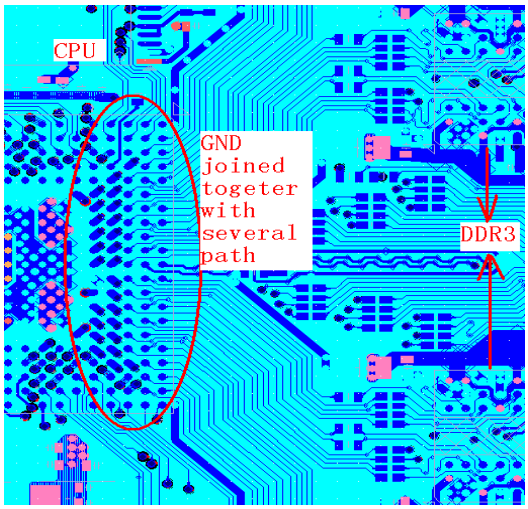


Figure 4-3 GND Joined Together

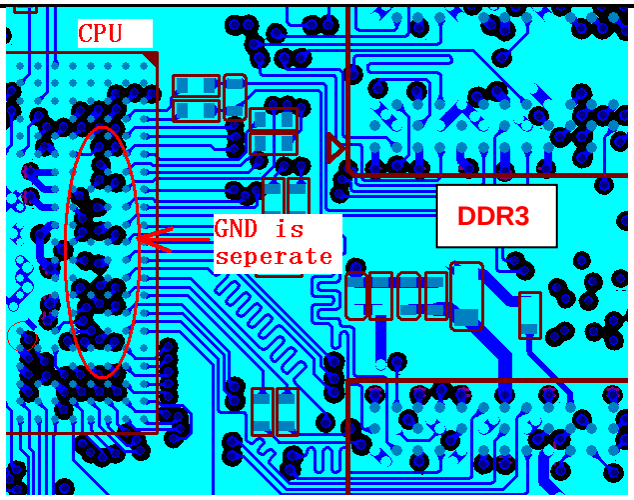


Figure 4-4 GND is seperate

- Each bus (D0~D8, DQS0/DQS0_N and DQM0 compose one bus, and etc.) should not cross other buses.If routing is carried out on two layers, byte lanes should be alternated between layers in order to reduce congestion, and crosstalk at the DRAMs; e.g. Byte lanes 0 and 2 should be routed on one layer, and 1 and 3 on the other.
- The crosstalk and characteristic impedance of an array of traces are interrelated. In order to minimize crosstalk, the characteristic impedance of a trace should be determined predominantly by the distance to the reference plane and not the distance to the neighboring traces. To achieve this, the space between traces should be twice the height of the trace above the ground plane. Figure4-5 show the recommended spacing (“H” means the height from reference plane.).

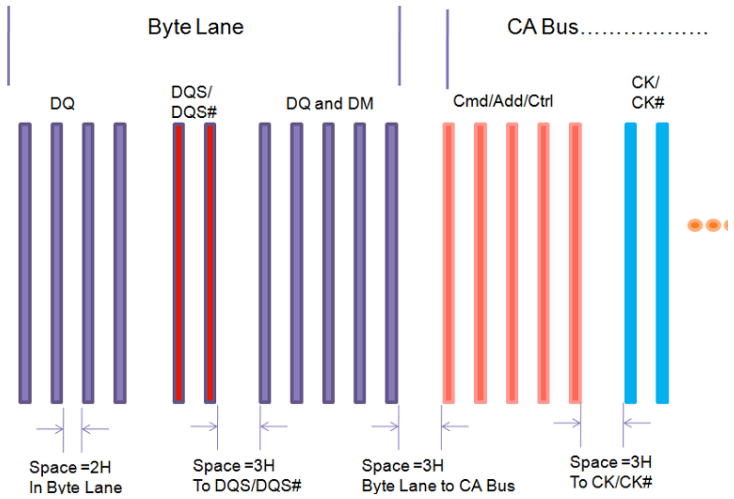


Figure 4-5 Spacing between Different Signal Groups

- Precise power supply bypassing is important for high-speed PCB. Control the power supply high-frequency impedance means controlling power supply inductance. Power supply high-frequency impedance is beaten down by many small capacitors connected between the power and ground plane. Using many capacitors, rather than a large one, will reduce the inductance. The inductance of a capacitor is dependent on its size. The capacitor

need to be placed very close to the device they are bypassing.

- VREF is used as a reference by the input buffers of the DDR3 memories. It is recommended to be 1/2 of the DDR3 power supply voltage and should be created using a resistive divider as shown in the schematic. Other methods are not recommended. Figure 4-5 shows the layout guidelines for VREF.

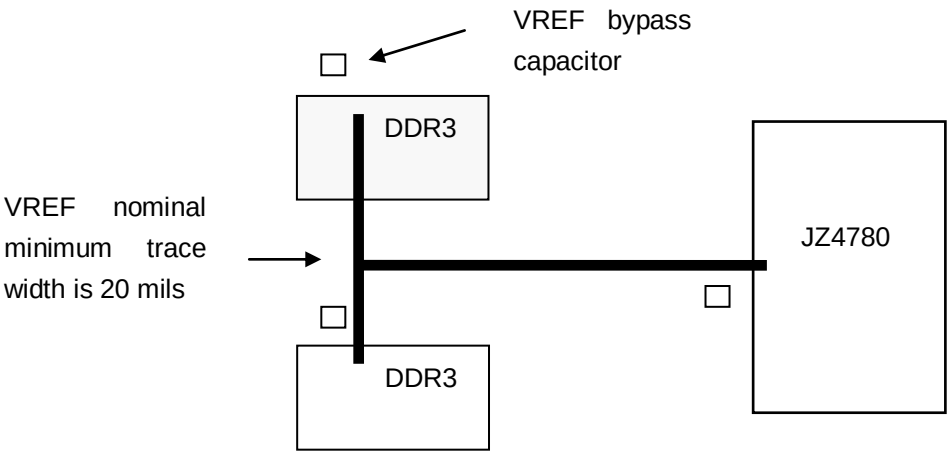


Figure 4-6 VREF Routing and Topology

- The region of the PCB used for DDR3 circuitry must be isolated from other signals. Region should be encompass all DDR3 circuitry and varies signals depending on placement. Non-DDR3 signals should not be routed on the DDR3 signal layer with in the DDR3 keep out region. No breaks should be allowed in the reference ground layers in the region. In addition, the +1.5V power plane should cover the entire keep out region.
- Bypassing capacitors should be close to the devices, or positioned for the shortest connections to pins, with wide traces to reduce impedance.

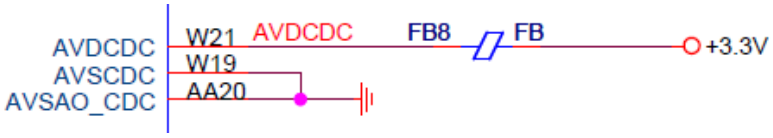
5 Audio Codec Design Guidelines

5.1 Overview

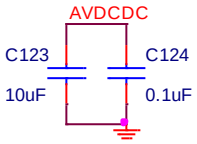
This chapter describes the embedded audio CODEC in the processor. This embedded CODEC is an I2S audio CODEC. AIC(AC'97 and I2S Controller) module is an interface to this CODEC for audio data replaying and recording.

5.2 Audio Power

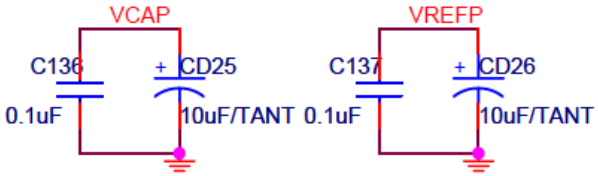
AVDCDC should be connected to a cleaned +3.3V power.



For a correct working, it is required to connect decoupling capacitors (10µF and 100nF ceramic) between the pins AVDCDC and AVSCDC.



An 10µF ceramic or tantalum capacitor in parallel with a 0.1µF ceramic capacitor should be attached from VCAP to VREFP to eliminates the effects of high frequency noise. This 0.1uF ceramic capacitor has to be kept as close as possible to IC package (closer than 0.2 inch)



5.3 Headphone Out

The AOHPL and AOHPR pins are connected to the headphone through an external bypass capacitor which is a DC blocking capacitors.

This capacitor is called CL. When the headphone resistance RL is 16 Ohm, The tantalum blocking capacitor CL is 220 uF.

The DC value of the signal AOHPL or AOHPR equals to AVDCDC/2.

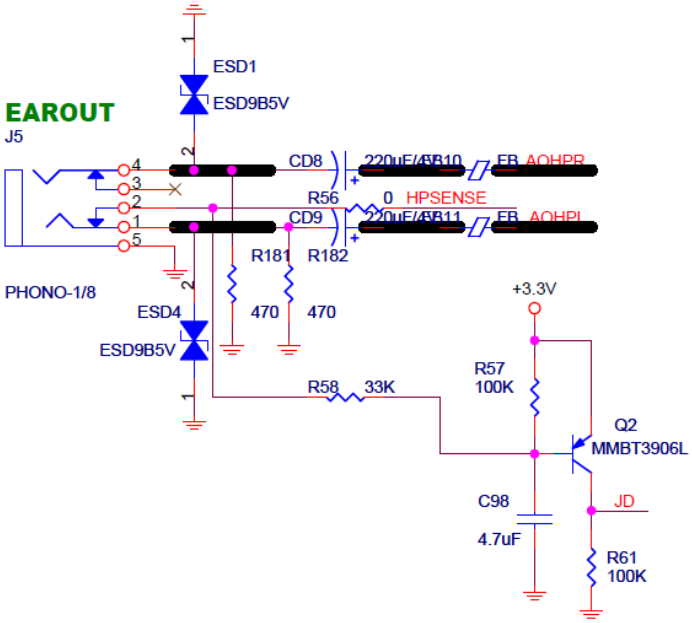
The ground of the headphone is connected to GND, which is the PCB analog single point reference (star connection) ground.

The R181 and R182 value are 470 Ohm, it use to prevent pop-up noise.

The ESD1 and ESD4 is an ESD transient voltage suppression component which provides a very high level of protection for sensitive electronic components that may be subjected to electrostatic discharge. The device provides protection for contact discharges to greater than +/-15KV.

JD are used to implement the headphone insert test. When the jack is inserted, the value of JD will be high: Otherwise, JD will be low.

HPSENSE is an alternative to implement the headphone insert test. It can be used just when CPU CODEC module work.



5.4 Mic In

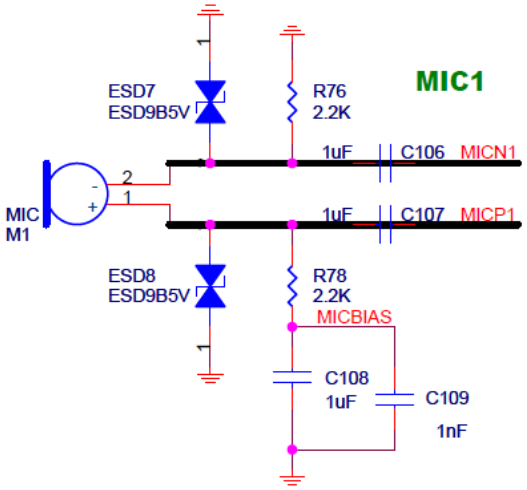
Specific value of resistor (R76,R78, commonly from 2.2k Ohm to 4.7k Ohm) and Vmicbias (if generated on board, usually from 1 to 2V or more) depends on the selected EC (Electret Condenser) microphone.

The 1nf decoupling capacitance used in MICBIAS pin removes high frequency noise of the chip.

Setting SB_MIC1/SB_MIC2 to 1 will close microphone input path for saving power, also setting SB_MICBIAS to 1 will close MICBIAS stage and the MICBIAS output voltage will be zero.

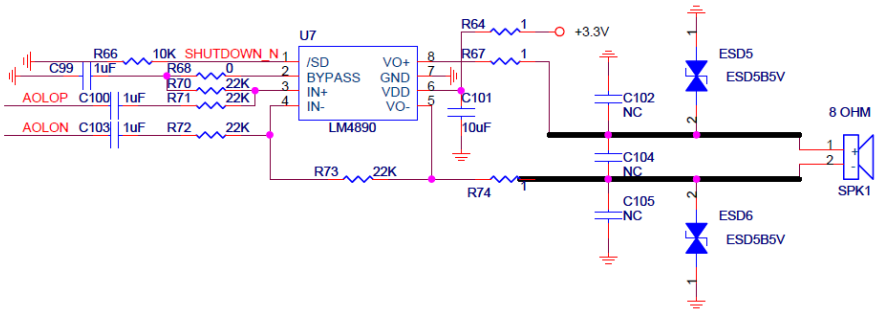
MICBIAS output voltage scales with AVDCDC, equals to 5/6*AVDCDC (typical 2.08V).

MICBIAS output current is 4mA max.



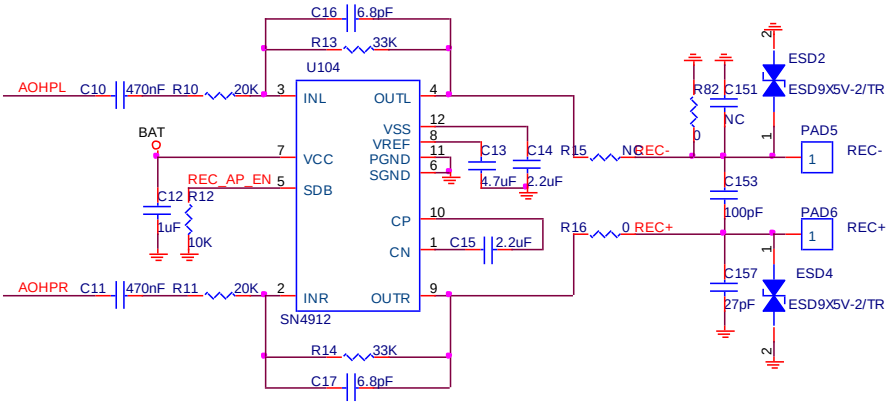
5.5 Speaker

The ESD5 and ESD6 is an ESD transient voltage suppression component which provides a very high level of protection for sensitive electronic components that may be subjected to electrostatic discharge. The device provides protection for contact discharges to greater than +/-15KV.



5.6 Receiver

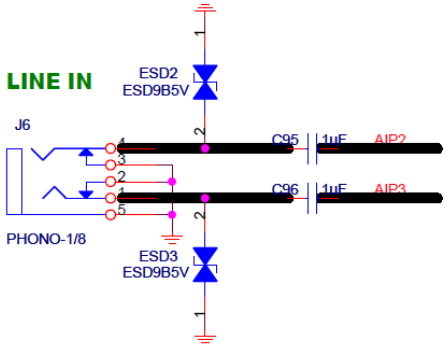
The ESD2 and ESD4 is an ESD transient voltage suppression component which provides a very high level of protection for sensitive electronic components that may be subjected to electrostatic discharge. The device provides protection for contact discharges to greater than +/-15KV.



5.7 Line In

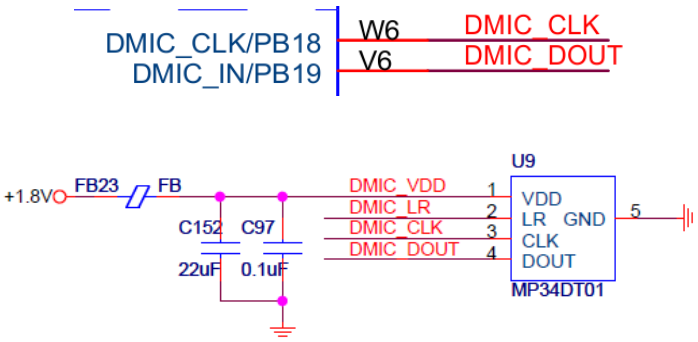
The ESD2 and ESD4 is an ESD transient voltage suppression component which provides a very high level of protection for sensitive electronic components that may be subjected to electrostatic discharge. The device provides protection for contact discharges to greater than +/-15KV.

Note: The single input port AIP1/AIN1 and single input port AIP2/AIP3 can be configure to microphone input or line input by software.



5.8 Digital Mic

For a correct working, it is required to connect decoupling capacitor (22μF and 100nF ceramic) between the pins DMIC_VCC and GND. The following figure is a typical design.



5.9 Layout Guideline

To ensure the maximum performance of the Audio, proper component placement and routing techniques are required. These techniques include properly isolating associated audio circuitry, analog power supplies, and analog ground planes, from the rest of the motherboard. This includes plane splits and proper routing of signals not associated with the audio section.

The basic recommendations are as follows:

- Special consideration must be given for the ground return paths for the analog signals.
- Digital signals routed in the vicinity of the analog audio signals must not cross the power plane split lines. Analog and digital signals should be located as far as possible from each other.
- Partition the board with all analog components grouped together in one area and all digital components in another.
- Separate analog and digital ground planes should be provided, with the digital components over the digital ground plane, and the analog components, including the analog power regulators, over the analog ground plane. The split between planes must be a minimum of 0.05 inch wide.
- Keep digital signal traces, especially the clock, as far as possible from the analog input and voltage reference pins.
- Do not completely isolate the analog/audio ground plane from the rest of the board ground plane. There should be a single point (0.25 inch to 0.5 inch wide) where the analog/isolated ground plane connects to the main ground plane. The split between planes must be a minimum of 0.05 inch wide.
- Any signals entering or leaving the analog area must not cross the ground split in the area where the analog ground is attached to the main motherboard ground. That is, no signal should cross the split/gap between the ground planes, which would cause a ground loop, thereby greatly increasing EMI emissions and degrading the analog and digital signal quality.
- Analog power and signal traces should be routed over the analog ground plane.

- Digital power and signal traces should be routed over the digital ground plane.
- Bypassing and decoupling capacitors should be close to the IC pins, or positioned for the shortest connections to pins, with wide traces to reduce impedance. It is especially for VCAP.
- All resistors in the signal path or on the voltage reference should be metal film. Carbon resistors can be used for DC voltages and the power supply path, where the voltage coefficient, temperature coefficient, and noise are not factors.
- Regions between analog signal traces should be filled with copper, which should be electrically attached to the analog ground plane. Regions between digital signal traces should be filled with copper, which should be electrically attached to the digital ground plane.

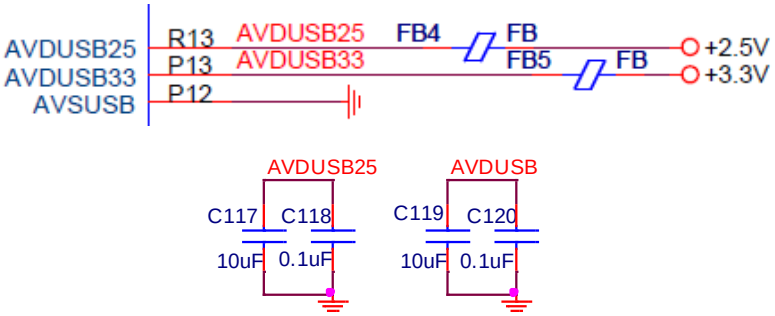
6 USB and OTG Design Guidelines

6.1 USB Overview

JZ4780 integrates USB Host Controller (UHC) which is Open Host Controller Interface (OHCI/EHCI)-compatible and USB Revision 1.1/2.0-compatible. It Embedded USB 2.0 PHY.
Familiarity with the Universal Serial Bus Specification, Revision 2.0 and the OHCI specification are necessary to fully understand the material.
It supports High speed, Full speed and low speed USB devices.

6.1.1 USB Power

For a correct working, it is required to connect decoupling capacitor (10μF and 100nF ceramic) between the pins AVDUSB25,AVDUSB33 and AVSUSB.
AVDUSB25 should be connected to a cleaned +2.5V power .
AUDUSB33 should be connected to a cleaned +3.3V power.

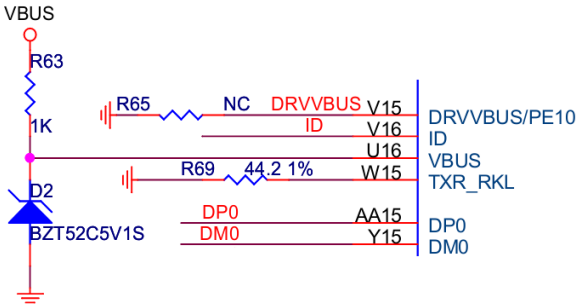


6.2 OTG Overview

The Universal Serial Bus (USB) supports serial data exchanges between a host computer and a variety of simultaneously accessible portable peripherals. Many of these portable devices would benefit a lot from being able to communicate to each other over the USB interface. And OTG make this possible. An OTG device can plays the role of both host and device.
JZ4780 also Complies with the USB 2.0 standard for high-speed (480 Mbps) functions and with the On-The-Go supplement to the USB 2.0 specification. Operates either as the function controller of a high-/full-speed USB peripheral or as the host/peripheral in point-to-point or multi-point communications with other USB functions

6.2.1 OTG Power

We should have a pin of VBUS supply power OTG, it have to connect to an external charge. The DRVVBUS is used to control whether to supply power for OTG.



To enable the OTG, the circuit should monitor the VBUS pin and can supply voltage for this pin and ID pin need connect CPU's ID pin and one GPIO pin for insert detection. Figure 7-1 shows the classic design for OTG function.

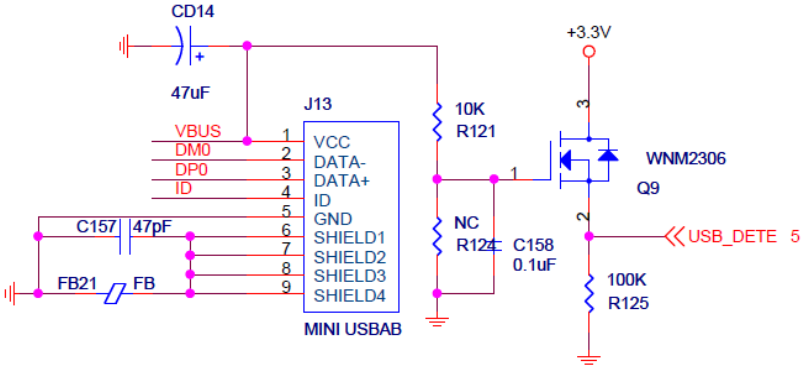


Figure 7-1 Classic Design for USB 2.0 OTG

To achieve this function, DRVBUS control 5V boost circuit whether supply voltage for VBUS or not. DRVVBUS controlled by the processor. Via the state of USB_DETE and ID pins, the processor can complete this task.

6.3 Guidelines for the USB and OTG interface

- The trace impedance for the DP and DM signals should be 45 Ω (to ground) for each USB signal DP or DM. This may be achieved with 9-mil-wide traces on the motherboard based on the stack-up recommended in Figure 7-3. The impedance is 90 Ω between the differential signal pairs DP and DM, to match the 90 Ω USB twisted-pair cable impedance. Note that the twisted-pair characteristic impedance of 90 Ω is the series impedance of both wires, which results in an individual wire presenting 45 Ω impedance. The trace impedance can be controlled by carefully selecting the trace width, trace distance from power or ground planes, and physical proximity of nearby traces.
- USB data lines should be routed as 'critical signals'. (i.e., hand-routing preferred). The DP/DM signal pair should be routed together and not parallel to other signal traces, to minimize cross-talk. Doubling the space from the DP/DM signal pair to adjacent signal traces will help to prevent cross-talk. The DP/DM signal traces should also be the same length, which will minimize the effect of common mode current on EMI.

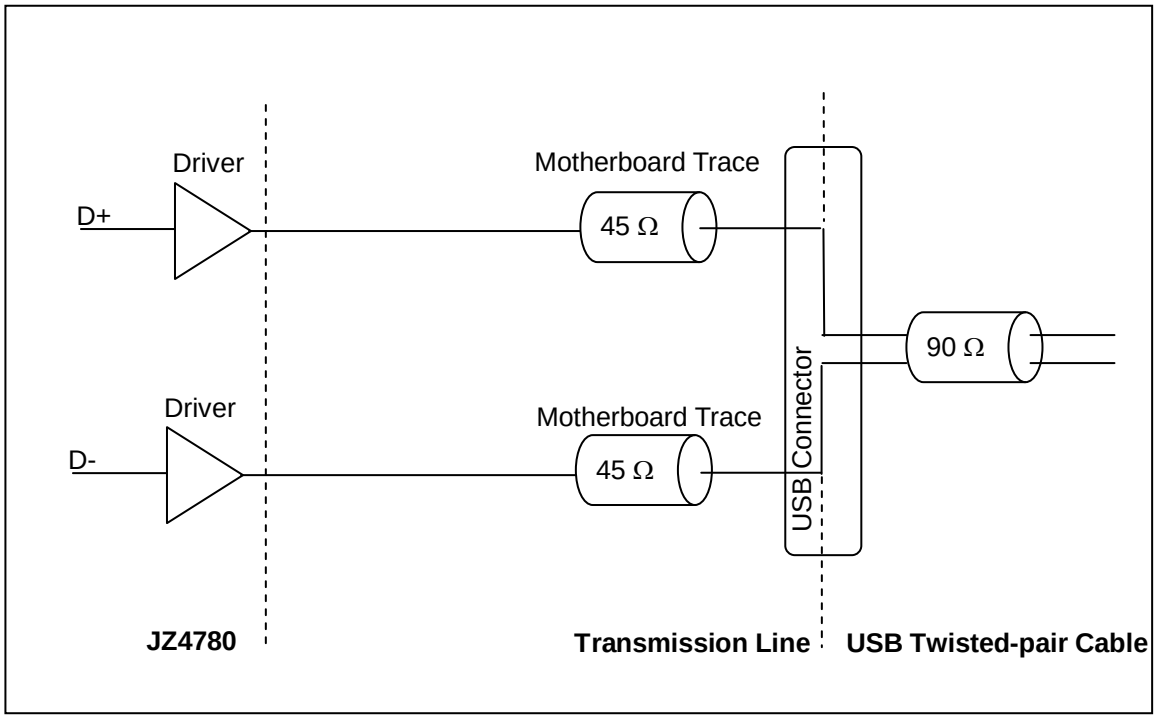


Figure 6-1 Recommend USB Schematic

7 LCD

The JZ4780 integrated LCD controller has the capabilities to driving the latest TFT LCD panels. It also supports some special TFT panels used in consuming electronic products. The controller performs the basic memory based frame buffer and palette buffer to LCD panel data transfer through use of a dedicated DMA controller. And OSD is also supported for LCD controller

Table 7-1 TFT and CCIR Pin Mapping

Pin Name	Funtion	Generic 8-bit Serial TFT	Generic 18/16-bit Parallel TFT		Special TFT 1 18/16-bit Parallel		Special TFT 2 18/16-bit Parallel		Special TFT 3 18/16-bit Parallel		CCIR656 8-bit	CCIR601 16-bit
LCD_PCLK PC8	Lcd_pclk/ Slcd_clk	CLK	CLK		DCLK		CLK		HCLK		CLK	CLK
LCD_VSYN PC19	Lcd_vsync/ Slcd_cs	VSYNC	VSYNC		SPS		GSRT		STV		VSYNC	VSYNC
LCD_HSYN PC18	Lcd_hsync/ Slcd_rs	HSYNC	HSYNC		LP		GPCK		STH		HSYNC	HSYNC
LCD_DE PC9	Lcd_de	DE	DE		-		-		-		-	-
LCD_B1 LCD_PS PC1	Lcd_ps	-	-		Pulse		Toggle		Toggle		-	-
LCD_R0 LCD_CLS UART4_RxD PC20	Lcd_cls	-	-		Pulse		Pulse		Pulse		-	-
LCD_B0 LCD_REV PC0	Lcd_rev	-	-		Toggle		Toggle		Toggle		-	-
LCD_G0 LCD_SPL UART4_TxD PC10	Lcd_spl	-	-		Pulse		Pulse		Toggle		-	-
LCD_R7 PC27	Lcd_dat17	-	R5	-	R5	-	R5	-	R5	-	-	-
LCD_R6 PC26	Lcd_dat16	-	R4	-	R4	-	R4	-	R4	-	-	-
LCD_R5 PC25	Lcd_dat15	-	R3	R5	R3	R5	R3	R5	R3	R5	-	D15
LCD_R4 PC24	Lcd_dat14	-	R2	R4	R2	R4	R2	R4	R2	R4	-	D14
LCD_R3 PC23	Lcd_dat13	-	R1	R3	R1	R3	R1	R3	R1	R3	-	D13
LCD_R2 PC22	Lcd_dat12	-	R0	R2	R0	R2	R0	R2	R0	R2	-	D12
LCD_G7 PC17	Lcd_dat11	-	G5	R1	G5	R1	G5	R1	G5	R1	-	D11
LCD_G6 PC16	Lcd_dat10	-	G4	G5	G4	G5	G4	G5	G4	G5	-	D10
LCD_G5 PC15	Lcd_dat9	-	G3	G4	G3	G4	G3	G4	G3	G4	-	D9

LCD_G4 PC14	Lcd_dat8	-	G2	G3	G2	G3	G2	G3	G2	G3	-	D8
LCD_G3 PC13	Lcd_dat7	R7/G7/B7	G1	G2	G1	G2	G1	G2	G1	G2	D7	D7
LCD_G2 PC12	Lcd_dat6	R6/G6/B6	G0	G1	G0	G1	G0	G1	G0	G1	D6	D6
LCD_B7 PC7	Lcd_dat5	R5/G5/B5	B5	G0	B5	G0	B5	G0	B5	G0	D5	D5
LCD_B6 PC6	Lcd_dat4	R4/G4/B4	B4	B5	B4	B5	B4	B5	B4	B5	D4	D4
LCD_B5 PC5	Lcd_dat3	R3/G3/B3	B3	B4	B3	B4	B3	B4	B3	B4	D3	D3
LCD_B4 PC4	Lcd_dat2	R2/G2/B2	B2	B3	B2	B3	B2	B3	B2	B3	D2	D2
LCD_B3 PC3	Lcd_dat1	R1/G1/B1	B1	B2	B1	B2	B1	B2	B1	B2	D1	D1
LCD_B2 PC2	Lcd_dat0	R0/G0/B0	B0	B1	B0	B1	B0	B1	B0	B1	D0	D0

Table 7-2 TFT 24 bit parallel mode/16 bit parallel mode2

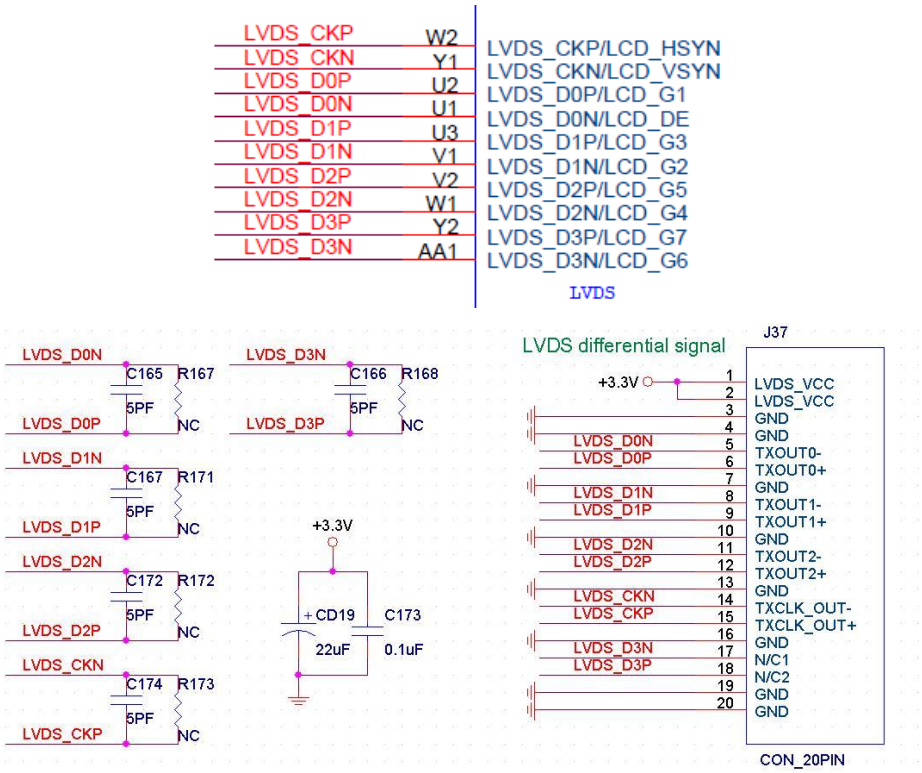
Pin Name	Function	16 bit Parallel mode2	24 bit Parallel
LCD_PCLK PC8	Lcd_pclk/ Slcd_clk	CLK	CLK
LCD_VSYN PC19	Lcd_vsync /Slcd_cs	VSYNC	VSYNC
LCD_HSYN PC18	Lcd_hsync /Slcd_rs	HSYNC	HSYNC
LCD_DE PC9	Lcd_de	DE	DE
LCD_B1 LCD_PS PC1	Lcd_ps	-	-
LCD_R0 LCD_CLS UART4_RxD PC20	Lcd_cls	-	-
LCD_B0 LCD_REV PC0	Lcd_rev	-	-
LCD_G0 LCD_SPL UART4_TxD PC10	Lcd_spl	-	-
LCD_R7 PC27	Lcd_dat17	R7	R7
LCD_R6 PC26	Lcd_dat16	R6	R6
LCD_R5 PC25	Lcd_dat15	R5	R5
LCD_R4 PC24	Lcd_dat14	R4	R4
LCD_R3 PC23	Lcd_dat13	R3	R3
LCD_R2 PC22	Lcd_dat12	G7	R2
LCD_G7 PC17	Lcd_dat11	G6	G7

LCD_G6 PC16	Lcd_dat10	G5	G6
LCD_G5 PC15	Lcd_dat9	0 (NC for panel)	G5
LCD_G4 PC14	Lcd_dat8	G4	G4
LCD_G3 PC13	Lcd_dat7	G3	G3
LCD_G2 PC12	Lcd_dat6	G2	G2
LCD_B7 PC7	Lcd_dat5	B7	B7
LCD_B6 PC6	Lcd_dat4	B6	B6
LCD_B5 PC5	Lcd_dat3	B5	B5
LCD_B4 PC4	Lcd_dat2	B4	B4
LCD_B3 PC3	Lcd_dat1	B3	B3
LCD_B2 PC2	Lcd_dat0	0 (NC for panel)	B2
LCD_R1 PC21	Lcd_lo6_o[5]	0	R1
LCD_R0 LCD_CLS UART4_RxD PC20	Lcd_lo6_o[4]	0	R0
LCD_G1 PC11	Lcd_lo6_o[3]	0	G1
LCD_G0 LCD_SPL UART4_TxD PC10	Lcd_lo6_o[2]	0	G0
LCD_B1 LCD_PS PC1	Lcd_lo6_o[1]	0	B1
LCD_B0 LCD_REV PC0	Lcd_lo6_o[0]	0	B0

8 LVDS

8.1 Overview

This product is a single-Link high speed LVDS (Low-Voltage Differential Signaling) transmitter used for digital flat panel display systems. It's compatible with ANSI/TIA/EIA-644-A (LVDS) Standard. The transmitter converts 28bits parallel TTL data into four LVDS data streams. An in-phase transmit clock is transmitted in parallel with the data streams over a fifth LVDS link. It support full HDTV display up to 1920x1080p @ 60 Hz.



The values of C165,C167,C172,C174 and R167,R171,R172,R173 are based on the datasheet of LCD.

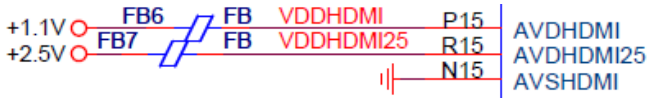
9 HDMI

9.1 HDMI Overview

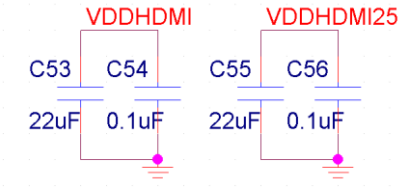
The JZ4780 support regular HDMI transmitter. The Interface conforms to the HDMI 1.4a standard。

9.1.1 HDMI Power

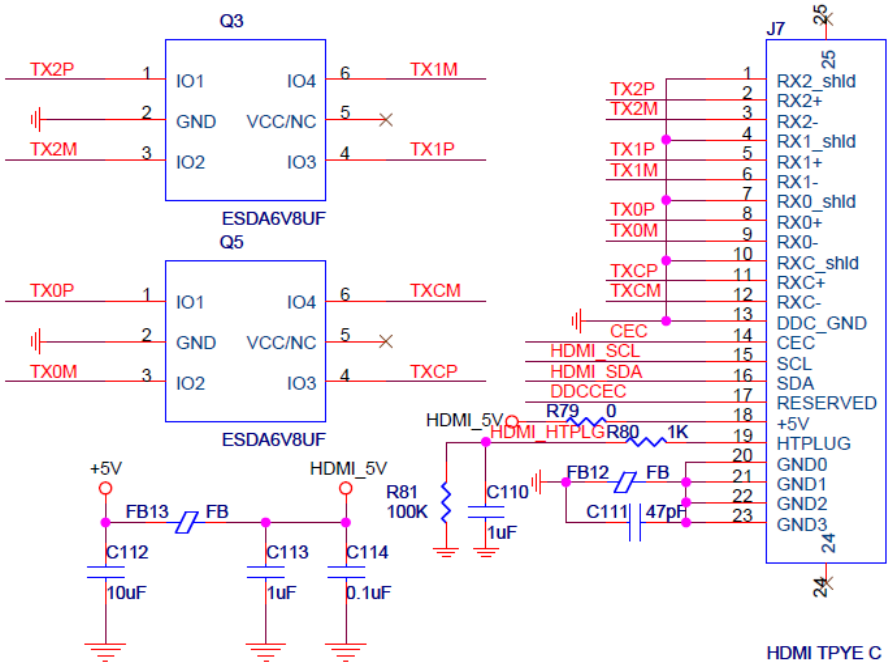
AVDHDMI should be connected to a cleaned +1.1V power.
AVDHDMI25 should be connected to a cleaned +2.5V power.



For a correct working, it is required to connect decoupling capacitors (22μF and 100nF ceramic) between the pins AVDHDMI，AVDHDMI25 and AVSHDMI.



The Q3 and Q5 is an ESD transient voltage suppression component which provides a very high level of protection for sensitive electronic components that may be subjected to electrostatic discharge. The device provides protection for contact discharges to greater than +/-15KV.



10 Camera

10.1 Overview

The CIM (Camera Interface Module) of JZ4780 connects to a CMOS or CCD type image sensor. The CIM source the digital image stream through a common 8-bit parallel common digital protocol. The CIM can directly connect to external CMOS image sensors and ITU656 standard video decoders.

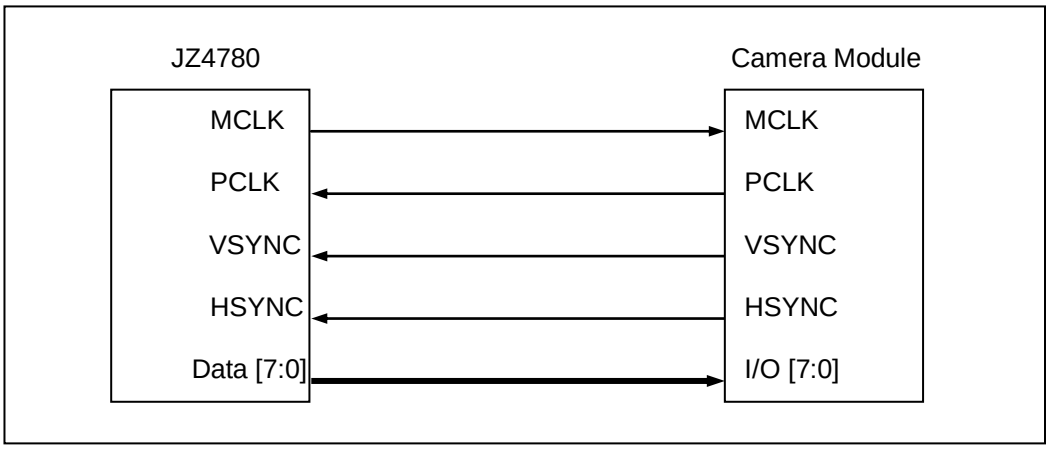


Figure 10-1 Example of Camera Module Interconnection

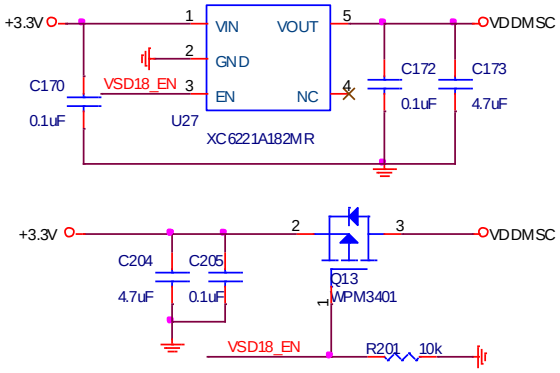
11 MSC

11.1 Overview

The Multi Media Card (MMC) is a universal low cost mass storage and communication media that is designed to cover a wide area of applications such as electronic toys, organizers, PDAs, smart phones, and so on. JZ4780 support SD Specification 3.0 and fully compatible with the MMC System Specification version 4.2.

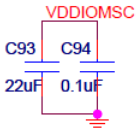
11.1.1 MSC Power

If used SD Specification 3.0, VDDMSC should be connected to a cleaned +3.3V and +1.8V power.The voltage change by software. The following figure is a typical design.



If not, VDDMSC could connected to +3.3V only.

For a correct working, it is required to connect decoupling capacitors (22µF and 100nF ceramic) between the pins VDDIOMSC and GND.

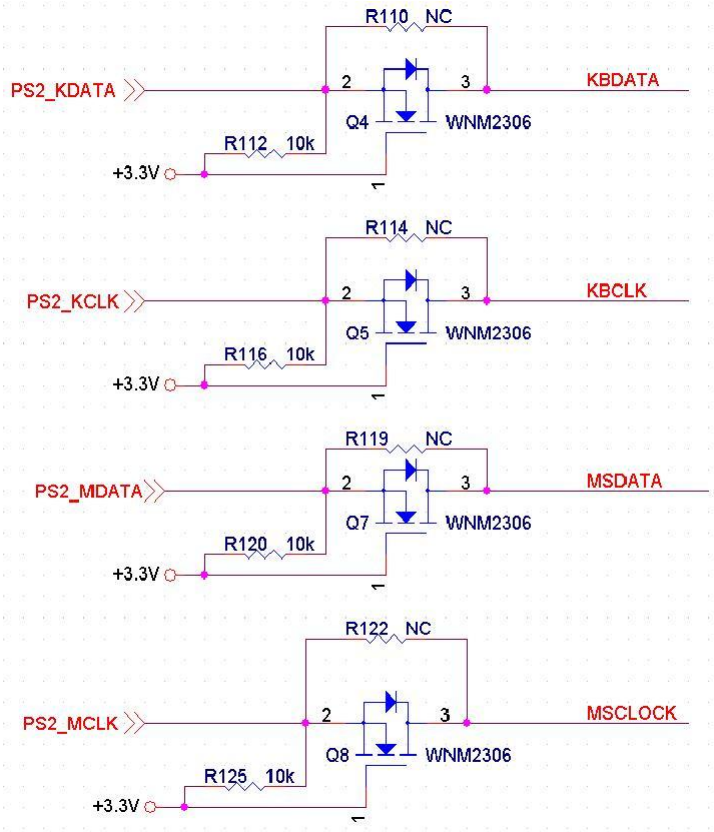


12 PS/2 and Keyboard

12.1 Overview

The JZ4780 processor integrate PS/2 keyboard controller (KBC) to provide the functions to a keyboard or to a PS/2 mouse. KBC receives serial data from the keyboard or mouse, checks the parity of the data, and presents the data to the system as a byte of data in its output buffer. The KBC is compatible with 8042.

The following figure is a typical design.



13 SAR A/D Controller

13.1 Overview

The A/D in falcon is CMOS low-power dissipation 12bit touch screen SAR analog to digital converter. It operates with 3.3/1.2V power supply. It is developed as an embedded high resolution ADC targeting to the 65nm CMOS process and has wide application in portable electronic devices, high-end home entertainment center, communication systems and so on.

The SAR A/D controller is dedicated to control A/D to work at three different modes: Touch Screen (measure pen position and pen down pressure), Battery (check the battery power), and two auxiliary input. Touch Screen can transfer the data to memory though the DMA or CPU. Battery and two auxiliary input can transfer the data to memory though CPU.

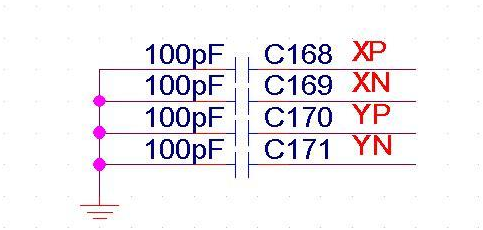
Table 13-1 SAR ADC Pins Description

NAME	I/O	Description
XN	AI	Touch screen analog differential X- position input
YN	AI	Touch screen analog differential Y- position input
XP	AI	Touch screen analog differential X+ position input
YP	AI	Touch screen analog differential Y+ position input
VBAT	AI	VBAT direct input
AUX1	AI	Auxiliary Input
AUX2	AI	Auxiliary Input

13.2 Touch Screen

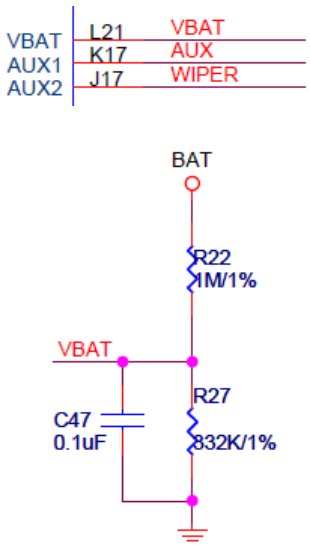
The JZ4780 can support 5-wire resistive touch screen.

There is needed a decouple capacitor for every channel to avoid the crosstalk from LCD. The value is decided by the touch screen and can be from 100pF to 1000pF.



13.3 Battery Voltage Measurement

Users who already deployed divider resistors on board level can use VBAT to direct measure the battery value. The following figure is the approach we recommend. Use the recommended resistance, you can control the power consumption easier.



14 OTP EFUSE

14.1 Overview

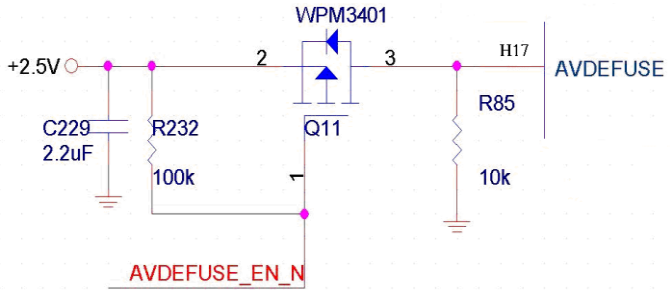
The EFUSE is provided 8K programmable bit, Total 8K bits are separated into seven segments, as below table show.

64bit	128bit	128bit	3520bit	8bit	2296bit	2048bit

Each segment can be programmed separately or together.
Each segment has a protect bit, which has higher priority than program segment selection.
Programming frequency is wide, and programming time can adjust use the EFUSE register.
Initial value of EFUSE is 0, when programmed to 1, it doesn't able to program back to 0 anymore.
Do not attempt to program any bit that already programmed to 1, such action will result unpredictable status to whole effuse block.

Programming voltage supply pin AVDEFUSE:

- In program mode, supply AVDEFUSE with 2.5V.
  Maximum accumulative time for AVDEFUSE pin exposed under 2.5V+/-10% should be less than 1sec.
- In read mode, leave AVDEFUSE to 0V or floating.



15 Ethernet Design Guidelines

15.1 Overview

As the JZ4780 processor not contains Ethernet media access controller (MAC), so it need external Ethernet MAC controller.

This section describes design guidelines for the LAN on board based JZ4780.

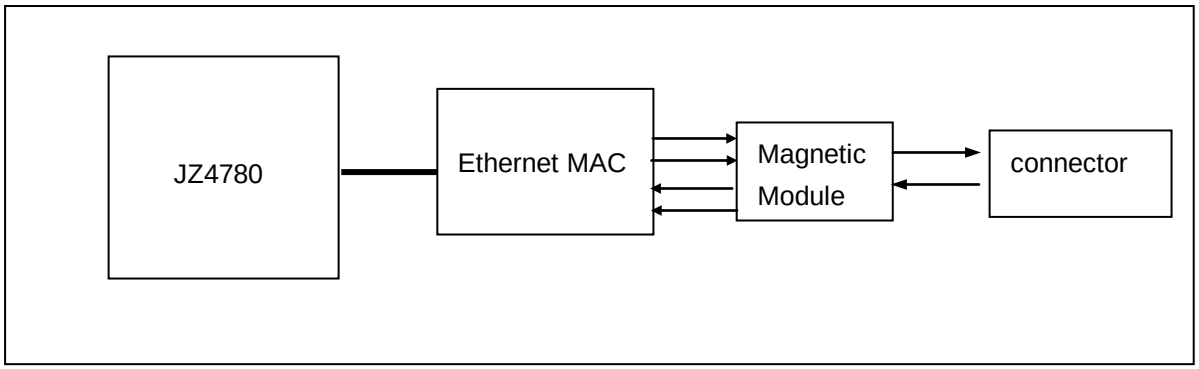


Figure 15-1 LAN On Board Implementation

15.2 JZ4780 Ethernet Controller Connection

JZ4780 Ethernet controller connection example is shown as figure 14-2, the Ethernet MAC chip maybe varied for specific OEM design targets.

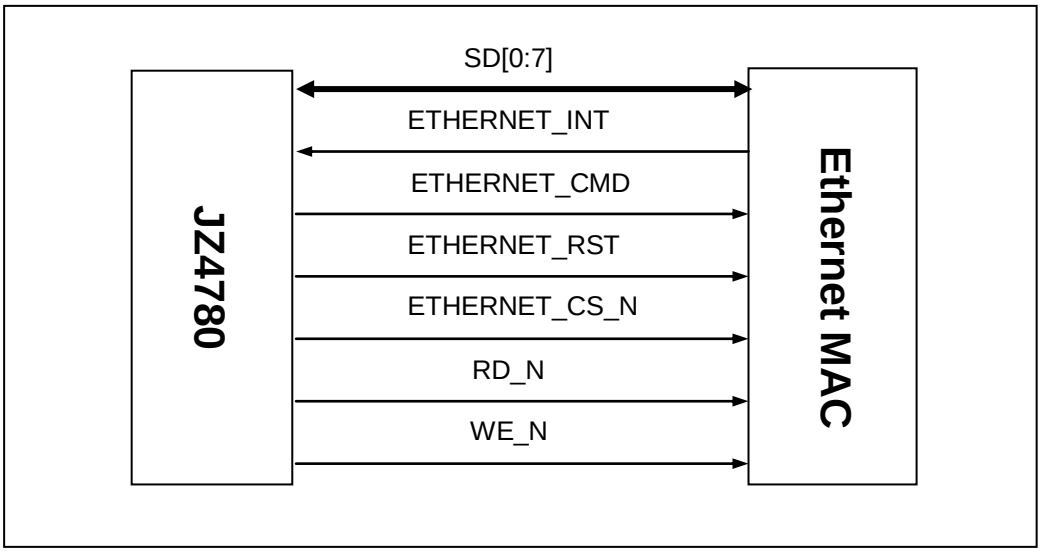


Figure 15-2 Controller Connection

16 RTC

16.1 Overview

The Real-Time Clock (RTC) unit can be operated in either chip main power is on or the main power is down but the RTC power is still on. In this case, the RTC power domain consumes only a few micro watts power.

The JZ4780 Need external 32768Hz oscillator for 32k clock generation

RTCLK selectable from the oscillator or from the divided clock of EXCLK, so that 32k crystal can be absent if the hibernating mode is not needed.

16.2 RTC Clock

The following figure is a typical design.

CLK32K PIN can output 32768KHz clock for other device.

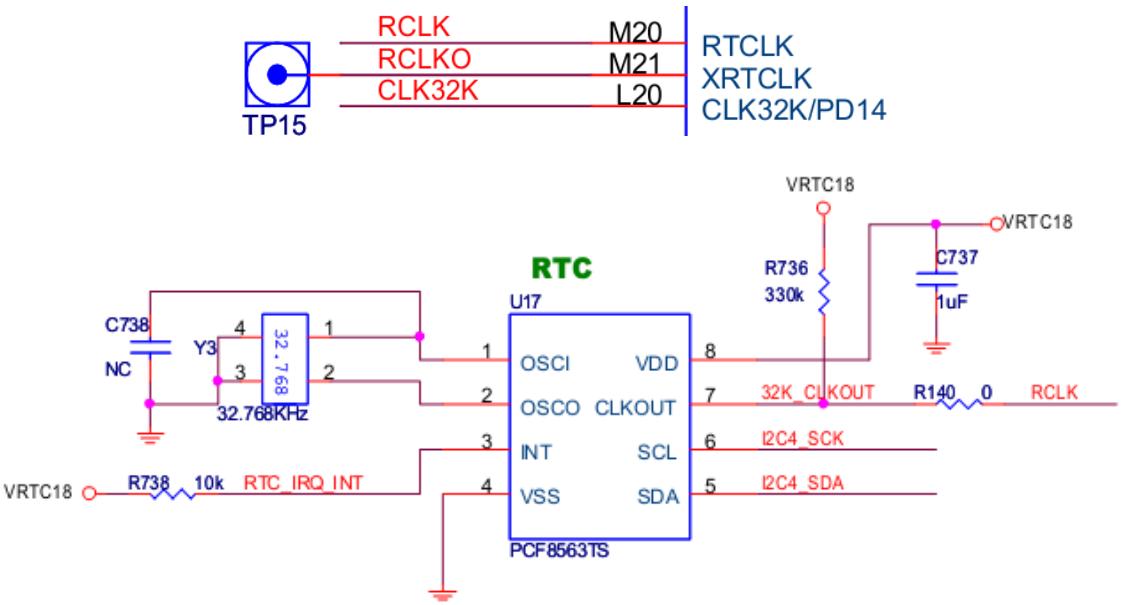


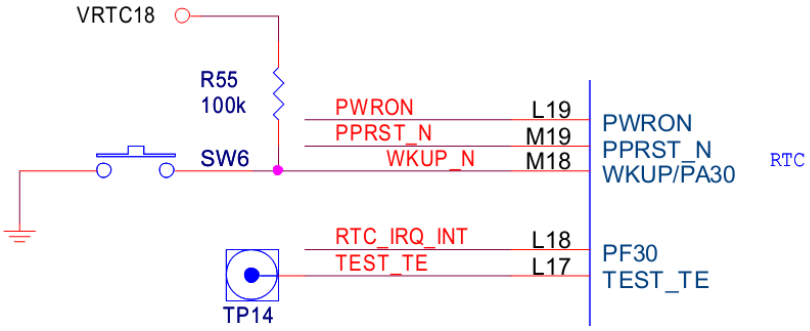
Table 16-1 RTC Clock Routing Summary

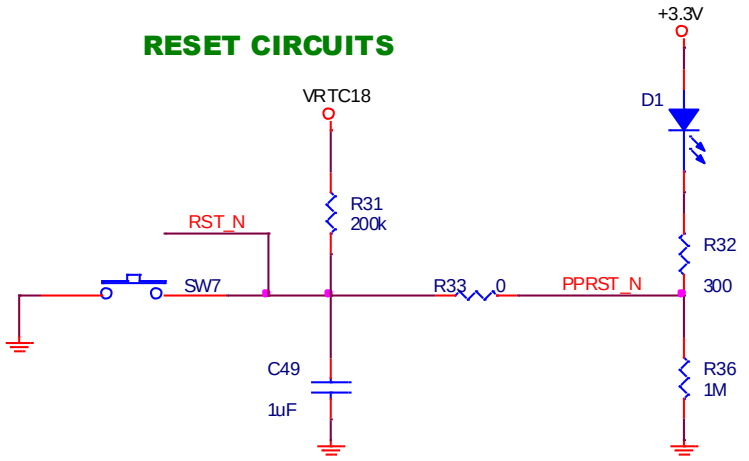
Trace Impedance	Routing Requirements	Maximum Trace Length To Crystal	Signal Length Matching	Signal Referencing
45 Ω to 69 Ω, 60 Ω Target	5 mil trace width (results in ~2pF per inch)	1 inch	NA	Ground

16.3 Power Control

The following is the recommended circuit for the system power control.

PWRON is an active high signal from CPU. If the power circuit enable signal is active high signal, you can use the PWRON directly. The resistance of R36 in the next figure is recommended to be 1M ohm, in this case ,you can consume less current when power down mode.





17 Miscellaneous Peripheral Design Guidelines

17.1 SSI Design Guideline

The SSI is a full-duplex synchronous serial interface and can connect to a variety of external analog-to-digital (A/D) converters, audio and telecom codecs, and other devices that use serial protocols for transferring data. The SSI supports National's Microwire, Texas Instruments Synchronous Serial Protocol (SSP), and Motorola's Serial Peripheral Interface (SPI) protocol.
The following figures show the connection example:

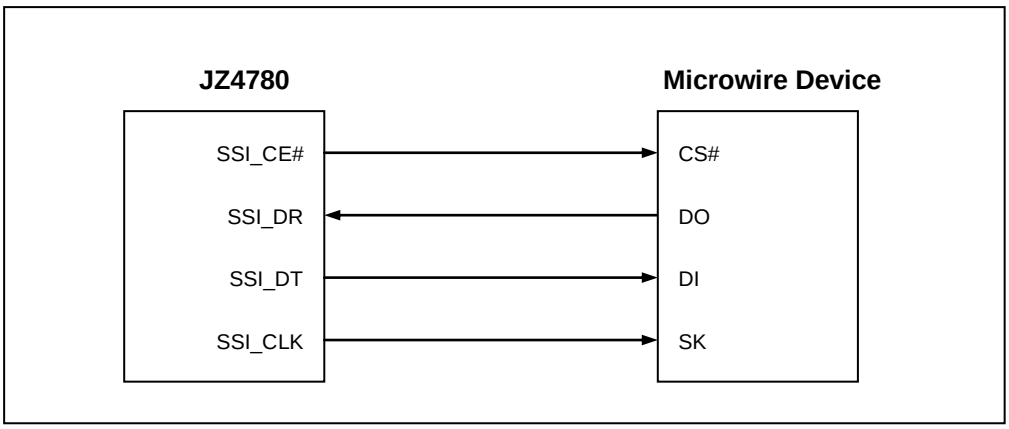


Figure 17-1 Microwire Interconnection

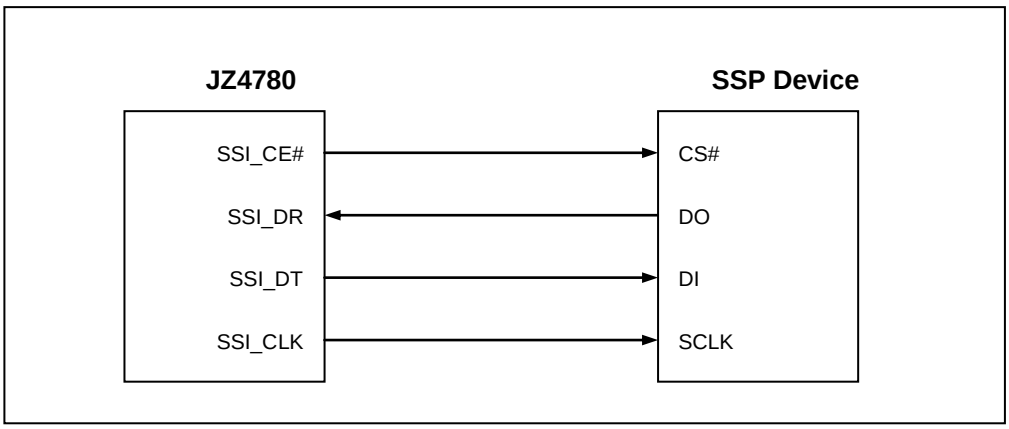


Figure 17-2 SSP Interconnection

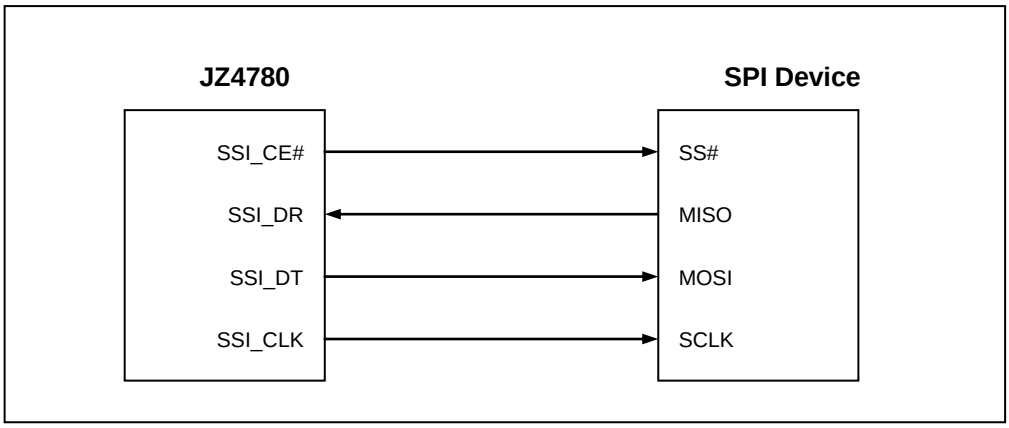


Figure 17-3 SPI Interconnection

17.2 UART

The JZ4780 processor has four UARTs: All UARTs use the same programming model. Each of the serial ports can operate in interrupt based mode or DMA-based mode.

The Universal asynchronous receiver/transmitter (UART) is compatible with the 16550 industry standard and can be used as slow infrared asynchronous interface that conforms to the Infrared Data Association (IrDA) serial infrared specification 1.1.0

17.2.1 UART Implementation

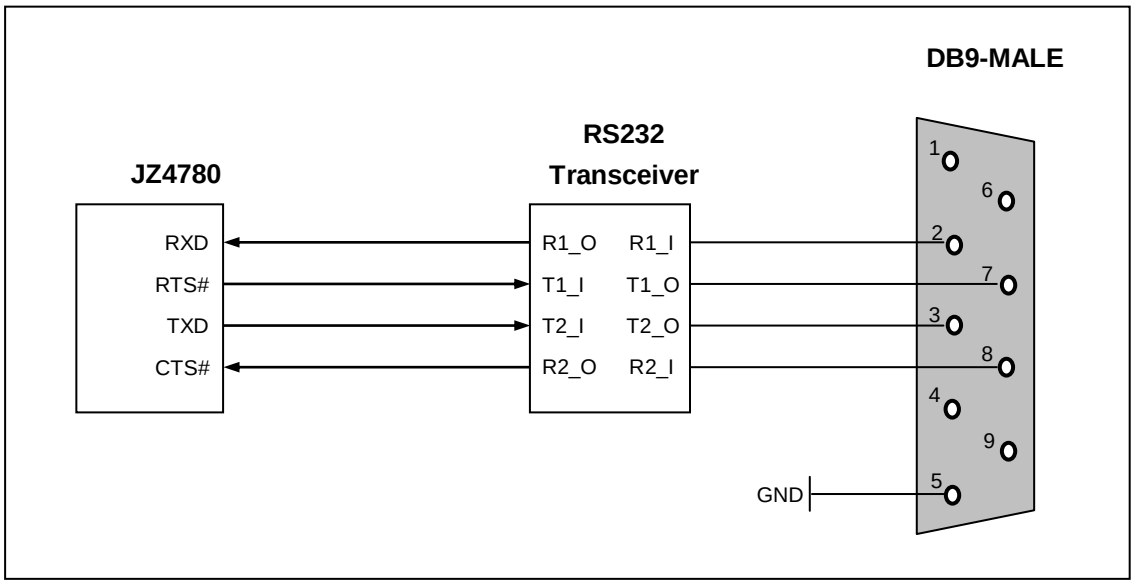


Figure 17-4 RS232 Serial Port Interconnection

17.3 I2C BUS

The I2C bus was created by the Phillips Corporation and is a serial bus with a two-pin interface. The SDA data pin is used for input and output functions and the SCL clock pin is used to control and reference the I2C bus. The I2C bus requires a minimum amount of hardware to relay status and reliability information concerning the processor subsystem to an external device.

The I2C module supports I²C standard-mode and fast-mode up to 400 kHz. The interface example is shown as following figure. The I2C bus serial operation uses an open-drain, wired-AND bus structure, so the pull-up (R1, R2=2.2K) is required on SCL and SDA. Refer to The I2C-Bus Specification for complete details on I2C bus operation.

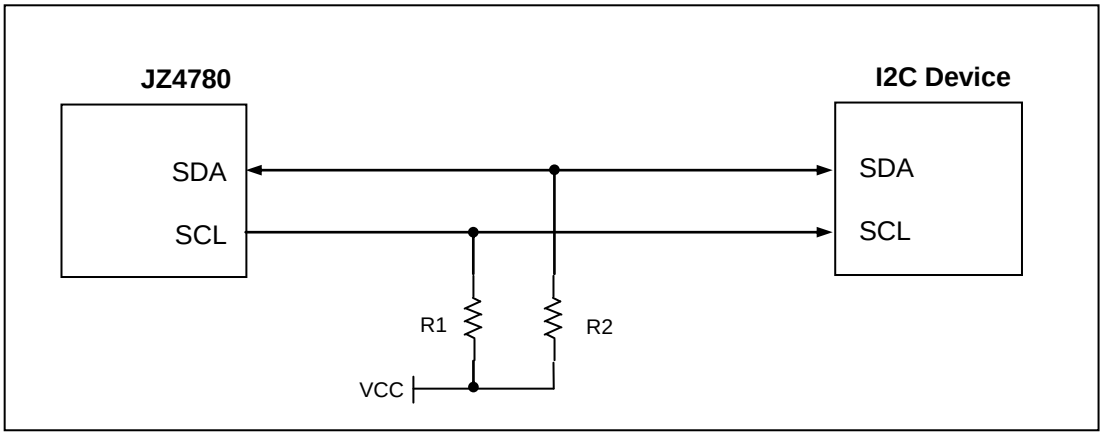


Figure 17-5 I2C Interconnection

17.4 PWM

The Pulse Width Modulator (PWM) is used to control the back light inverter or adjust bright or contrast of LCD panel and also can be used to generate tone. PWM consists of a simple free-running counter with two compared registers, each compare register performs a particular task when it matches the count value. The period comparator causes the output pin to be set and the free-running counter to reset when it matches the period value. The width comparator causes the output pin to reset when the counter value matches. JZ4780 contains eight pulse width modulators: PWM0 ~ PWM7.

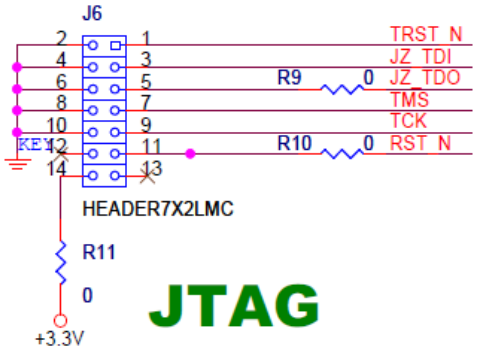
17.5 GPIO

The JZ4780 processor provides 180 multiplexed General Purpose I/O Ports (GPIO) for use in generating and capturing application-specific input and output signals. Each port can be programmed as an output, an input or function port that serves certain peripheral. As input, pull up/down can be enabled/disabled for the port and the port also can be configured as level or edge tripped interrupt source.

17.6 JTAG/Debug Port

JZ4780 has a built-in JTAG/Debug port. All JTAG pins are directly connected. The following figure shows the connection of the JTAG port. Pin 11 RST_N should be connected to system reset circuit.

Pin 12 is a KEY. The header should be a 7X2(2.54mm Pitch) male header with coat.



18 Platform Clock Guidelines

The JZ4780 processor contains one 48MHZ oscillator circuit

The Chip has four Phase Locked Loops (PLL):

1. APLL mainly is used to CPU and L2CACHE, also may use the AHB0 and AHB2, other peripherals
2. MPLL mainly is used to DDR, also uses AHB0 and AHB2 APB, other peripherals
3. VPLL mainly is used to VIDEO and other special clocks
4. EPLL mainly is used to AUDIO and other special clocks

The following is the recommended circuit for main clock. When layout the board, you should keep the distance between Y1 and JZ4780 as short as possible.

The 48MHz crystal should be +/-20ppm, CL≤12pF, ESR≤80ohm

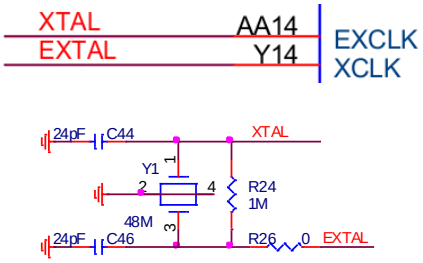


Table 17-1 Main Clock Routing Summary

Trace Impedance	Routing Requirements	Maximum Trace Length To Crystal	Signal Length Matching	R52, C34, and C35 Tolerances	Signal Referencing
45 Ω to 69 Ω, 60 Ω Target	5 mil trace width (results in ~2pF per inch)	1 inch	NA	R24 = 1M ± 5% C44=C46=24pF±10% (Typical) The value of C44, C46, R26 and R24 should be referred to the crystal's specification	Ground

19 Platform Power Guidelines

19.1 Overview

The JZ4780 processor needs four voltages: +3.3V, +1.5V for memory, +1.1V for core,+2.5V for USB OTG and HDMI. The following figure is a typical power circuit in the tablet and smart phone application.

The +1.1V need a power chip which can supply at least 1A, 1.2A is recommended.

The +1.5V current request is based on the external memory type and quantity.

And the +3.3V current request is based on the external device.

Normally, we use an 1.2A power chip for +1.5V and a 950mA power chip for +3.3V.

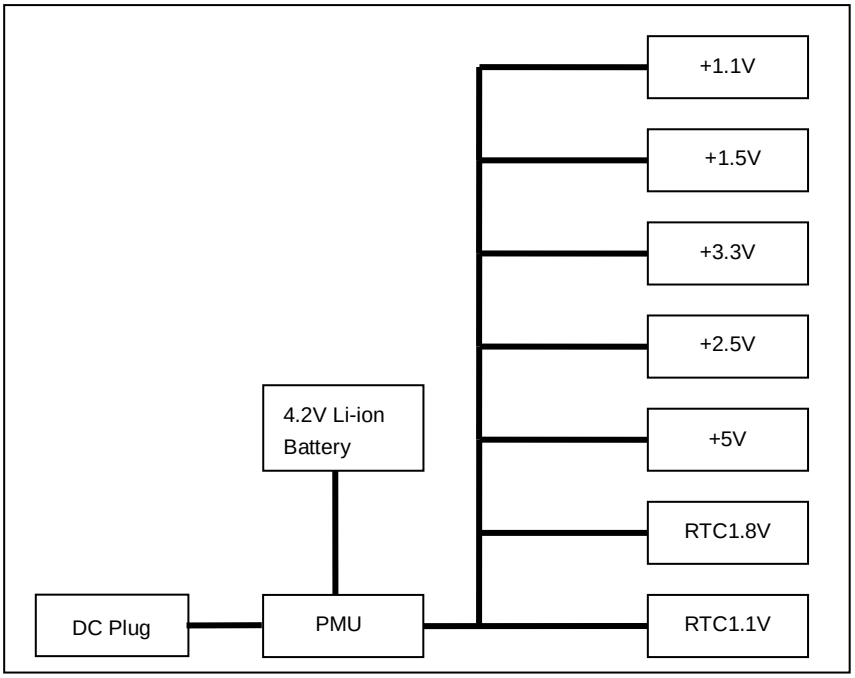
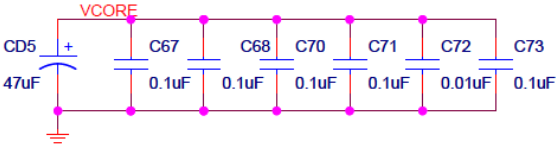


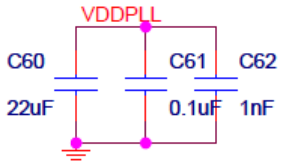
Figure 19-1 Power Architecture

19.2 Power Delivery and Decoupling

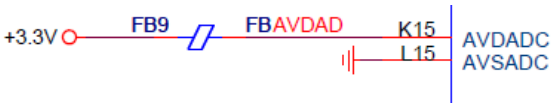
The VDDIO (+3.3V), VMEM(+1.5V) and VDDCORE (+1.1V) of JZ4780 should be decoupled with 100nF, 10nF and 1nF capacitor.

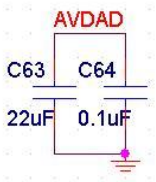


The Power of PLL should be as the following circuit.

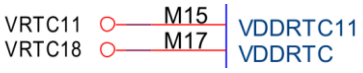


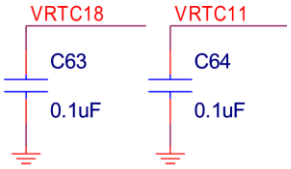
The power of ADC should be as the following circuit.



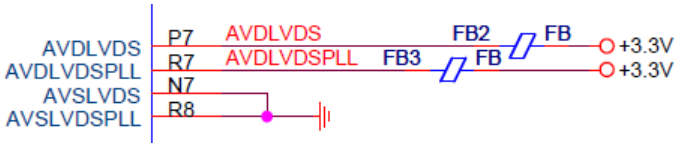


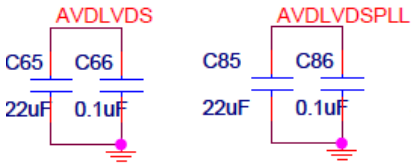
The power of RTC should be as the following circuit. The capacitors should be placed near the Pin of power. The traces from capacitor to the Pin should be short and width.



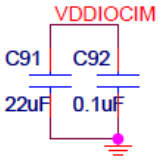


The power of LVDS should be as the following circuit.





The power of CIM should be as the following circuit.



The power of GPIO should be as the following circuit.