Hardware guide (NS-RX231)

Contents

1 Preface

This board is an evaluation board of RX231 sensible a capacitive touch, and I made this board for the demonstration of RX231 MCU. With this board, I will guide how to design the circuitry, a printed circuit board, the application design that guides a capacitive touch part, the software designing method, and the system designer design hardware and program the software easily.

Additionally, using this board, you can learn how to design and program the infrared remote-control transmission department, infrared remote-control reception part, 6 axis acceleration / gyro sensors MPU-6050 module, the speaker output and UART and Bluetooth communication. In addition, you can connect to various boards and explain the method to utilize a capacitive touch sensor using these board and documents.

The purpose of this hardware guide is to support you to be able to design the circuitry about the capacitance-type touch sensor board and a printed circuit board efficiently.

2 Specifications of the board

MCU: RX231(R5F52315ADFM), 64 pin version of 32 bits, MAX 54MHz clock, 88.56 DMIPS,
USB 2.0 Full speed)

PLL circuit dedicated to USB: It supplies 48MHz that multiplied 12MHz from a crystal oscillation circuit 4 times as a clock for the USB

Subsystem clock circuit: Crystal 32.768kHz

A connector standard: A connector and compatibility of UNO of ARDUINO

Infrared remote-control transmission department (TSOP4138) ······························ 1pcs

Infrared remote-control reception department (ST-3311-H) ······························ 1pcs

Connector for the MPU-6050 6 axis acceleration / gyro sensor module.(J18) ················ 1pcs

Speaker (PS1240P02CT3 compatibility) ······························ 1pcs

Connector for the UART communication(female, the connector for the Bluetooth module)··········· 1pcs

Connector of the UART communication((male, the connector for the UART ······························ 1pcs

Power supply connector (the battery(J13), USB power supply(J12), adapter(J7) ······················ 1pcs

Jumper (power supply choice(J10), AVCC power supply to VCC directly(J11), USB boot mode choice(J3), (J17) selection of AD0 address of MPU-6050)······························each 1pcs

Reset switch (S1)······························ 1pcs

Boot mode selection switch (SW1)······························ 1pcs

USB connector (J12) ······························ 1pcs

CAP Touch pad······························ 5pcs

Proximity sensor pad ······························ 1pcs

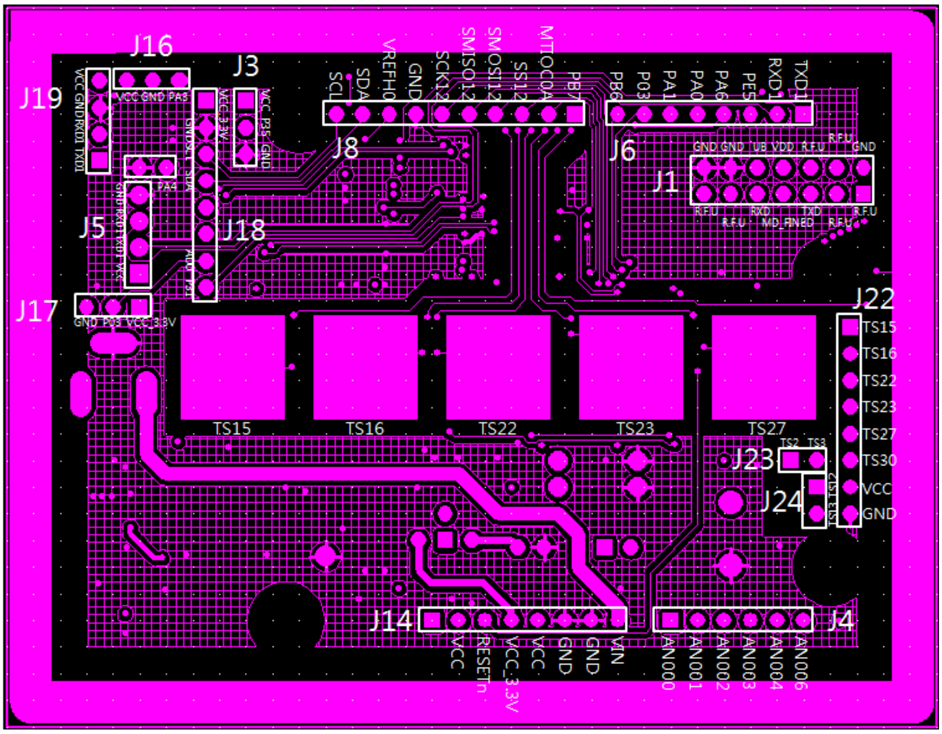
Additional external connector (6 channel Cap touch (J22), 2 channel(J23), 2 channel(J24) ·each 1pcs

3 Constitution of the board

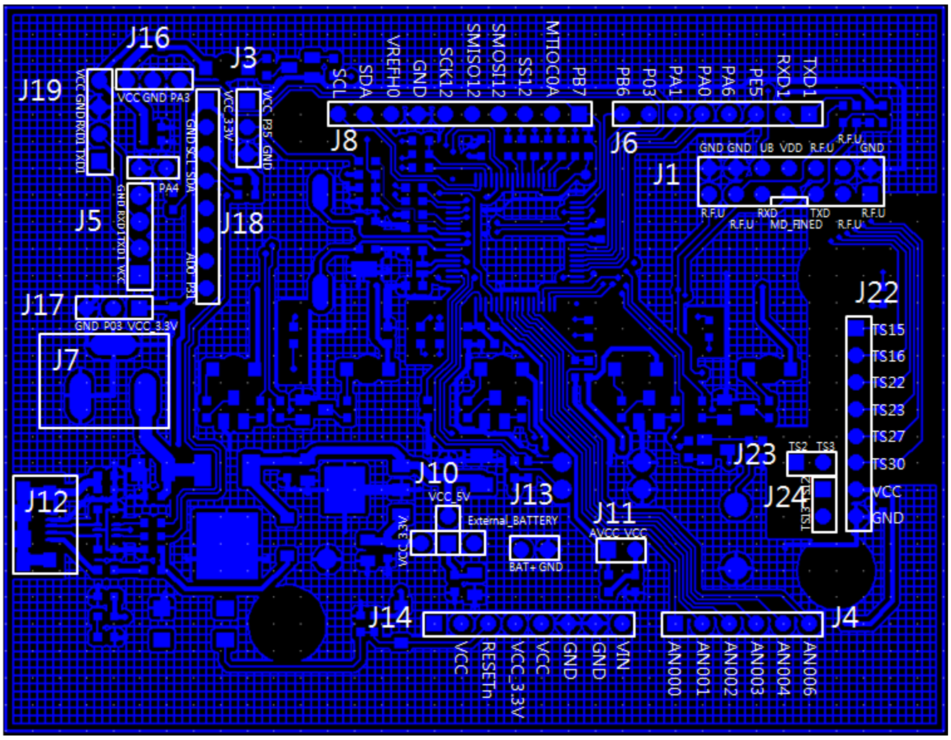
NS-RX231 capacitive touch sensor board ······· 1pcs
Acrylic board (90*68.5mm) ······· 1pcs
Simple guide of the board ······· 1pcs
Short Jumper ······· 4pcs

4 Explanation of the capacitive touch sensor board

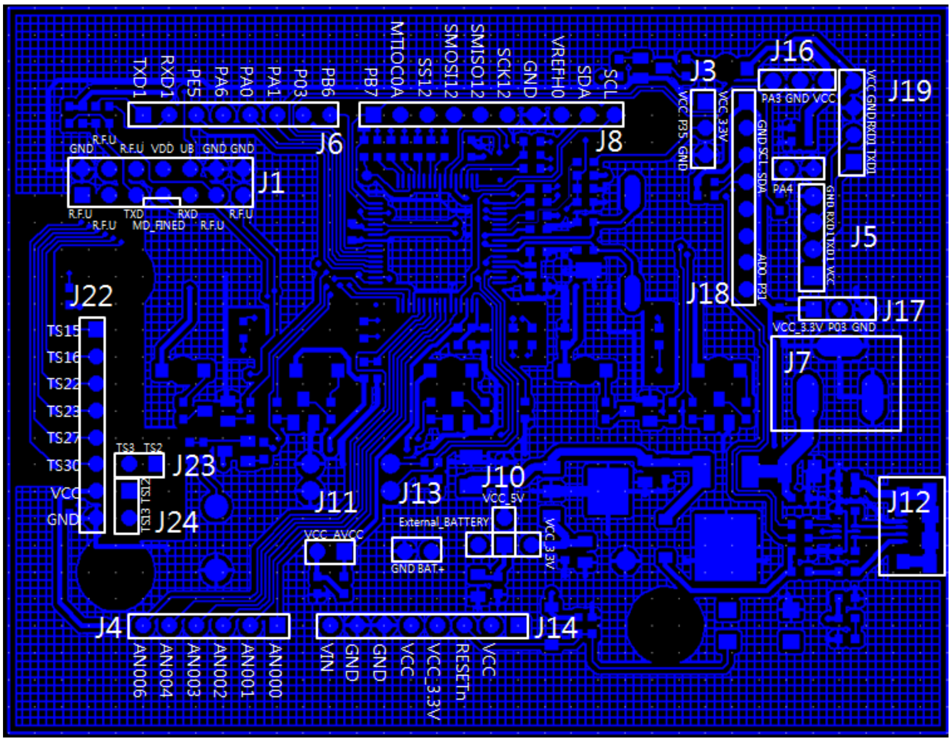
Here is a function name of the pin nearby connector so each function of the pin could be easily checked



Function name of each pin on TOP layer



Function name of each pin on perspective view of the BOTTOM layer



Function name of each pin on BOTTOM layer

4.1 Explanation about the jumper

J10 : Power selection (#1 : VCC, #2 : VCC_3.3V, #3: External_battery, #4 : VCC_5V)
J11 : For direct connection of AVCC and VCC (#1: AVCC, #2: VCC)
J3 : USB boot mode selection (#1: VCC, #2: P35, #3: GND)
J17 : Address selection of AD0 on MPU-605 module (#1: VCC, #2: AD0, #3: GND)

4.2 Explanation about the connector

J1 : JTAG connector
(#1: R.F.U, #2: GND, #3: R.F.U, #4: R.F.U, #5: TXD, #6: R.F.U, #7: MD_FINED, #8: VDD, #9: R.F.U, #10: UB, #11: RXD, #12: GND, #13: RESn, #14: GND)
J4 : UNO compatible connector
(#1: AN000, #2: AN001, #3: AN002, #4: AN003, #5: AN004, #6: AN006)
J5 : UART(SCI1) : (#1: VCC, #2: TXD1, #3: RXD1, #4: GND)
J6 : UNO compatible connector
(#1: TXD1, #2: RXD1, #3: PE5, #4: PA6, #5: PA0, #6: PA1, #7: P03, #8: PB6)
J7 : Power adaptor (#1: VIN, #2: GND, #3: GND)
J8 : UNO compatible connector (#1: PB7, #2: MTIOC0A, #3: SS12, #4: SMOSI12, #5: SMISO12, #6: SCK12, #7: GND, #8: VREFH0, #9: SDA, #10: SCL)
J12 : USB connector : (#1: VBUS, #2: D-, #3: D+, #4: GND, #5: SHIELD)
J13 : External battery connector : (#1: BAT+, #2: GND)
J14 : UNO compatible connector
(#1:N.C, #2 : VCC, #3 :RESETn, #4 : VCC_3.3V, #5 : VCC, #6 : GND, #7 : GND, #8 : VIN)
J16 : InfraRed receiver module (#1 : PA3, #2 : GND, #3 : VCC)
J18 : MPU-6050 module connector
(#1 : VCC_3.3V, #2 : GND, #3 : SCL, #4 : SDA, #5 : N.C., #6 : N.C., #7 : AD0, #8 : P31)
J19 : UART(SCI1) : (#1 : TXD1, #2 : RXD1, #3 : GND, #4 : VCC)

J22 : external Cap touch connector (#1 : TS15_e, #2 : TS16_e, #3 : TS22_e, #4 : TS23_e,
#5 : TS27_e, #6 : TS30_e, #7 : VCC, #8 : GND)

J23 : external Cap touch connector (#1 : TS2_e, #2 : TS3_e)

J24 : external Cap touch connector (#1 : TS12_e, #2 : TS13_e)

J2 : This pin is directly connected with Speaker (+).

4.3 The name of function used in the multi-function pin

#2 pin(TXD1) of J5, #3 pin(RXD1) of the UART function of SCI1 works as TXD1, RXD1 function.

An applicable pin of J18 (terminal for the MPU-6050 module connection)

- #3 pin: It is connected to #10 pin of J8 in SCL of I2C(RIIC0)
- #4 pin: It is connected to #9 pin of J8 in SDA of I2C(RIIC0)

#1 pin (TXD1) of J19, #2 pin (RXD1) of the UART function of SCI1 works as TXD1,RXD1 function.

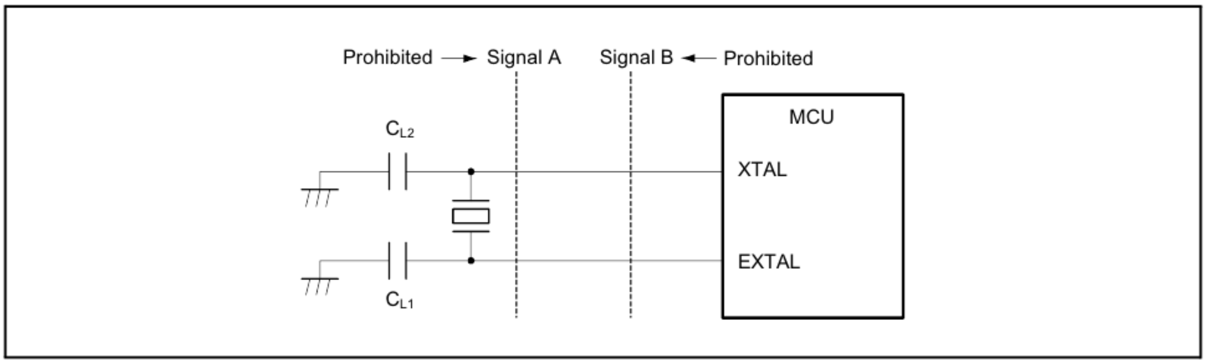
J22 and R58,R59,R60,R61,R62,R63 are in N.C (unconnected) state at the time of the factory shipment. The reason why not connected is to maximize the sensitivity of the proximity sensor (TS30) and five touch sensor (TS15,TS16,TS22,TS23,TS27). When you want to use a touch sensor to outside , please connect 0 Ω resistor to R58,R59,R60,R61,R62,R63. After connection, please use TS15,TS16,TS22,TS23,TS27 function as a capacitive touch sensor with that connector.

J23 is used to test TS2 and TS3 cap touch from outside of the board.
J24 is used to test TS12 and TS13 cap touch from outside of the board.
With J2, ON/OFF controls GPIO using P05 and outputs a sound from a speaker.

In addition, by a multi-function pin controller function, you can drive a speaker in DA1 of the Digital to Analog converter output.

5 Circuit design of capacitive touch sensor board using RX231

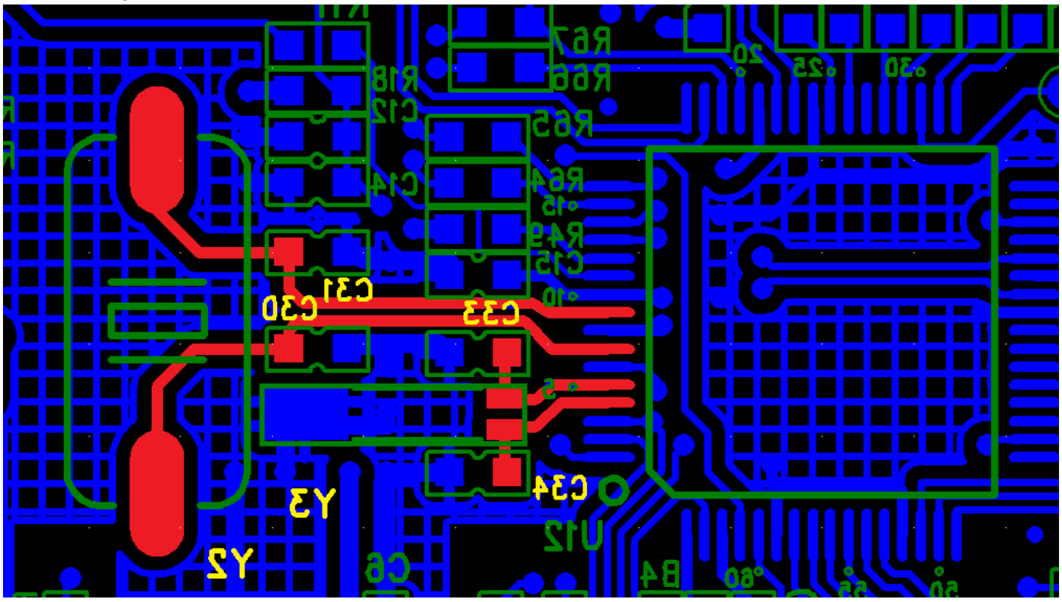
5.1 Design of the clock part



[1]

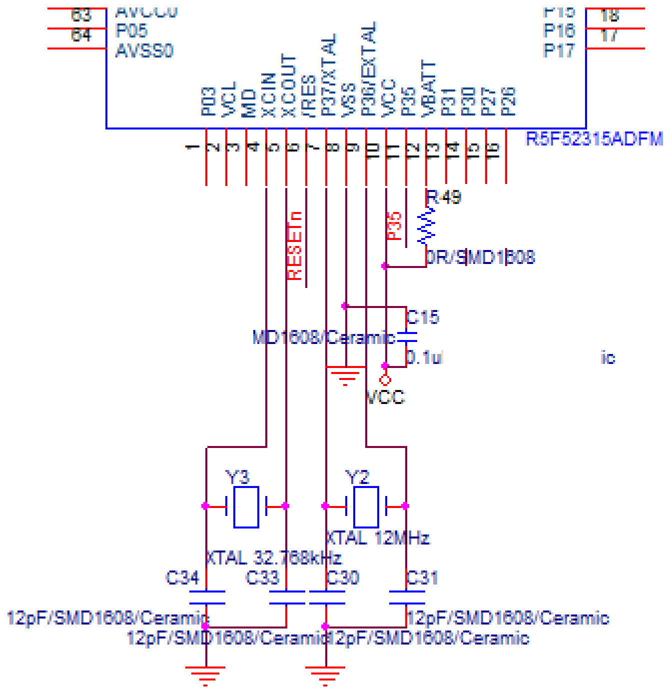
When using a crystal, a crystal and the load capacitor must be located near to XTAL and the EXTAL pin as close as possible. As shown above, if other signal lines are placed on the clock line, it may affect the clock by electromagnetic induction, so other signal lines should not be placed on the clock line.

The LGP(Local Ground Plane) is positioned at the circumference of the clock and connected directly to the PCB's main ground plane via at least two grounding Via holes because the clock is over 1MHz, it should be multi-grounded.



Example of design of clock part using LGP on actual PCB of NS-RX231 board

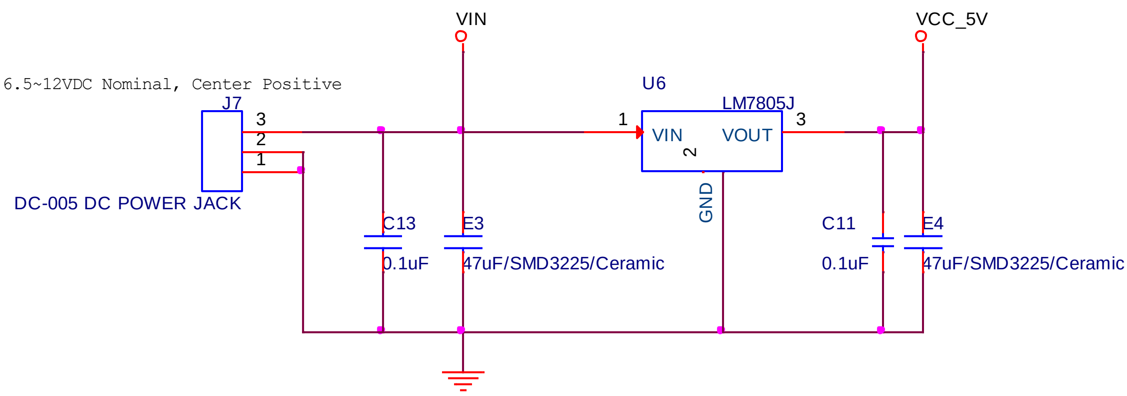
[1] HardwareUsersManual of RX231, Page246 Figure9.11



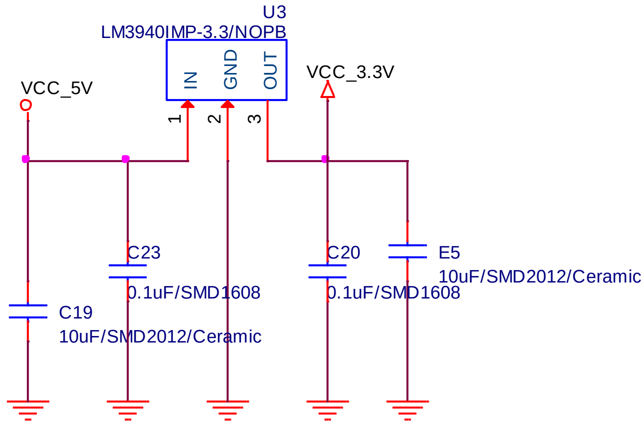
Example of circuit design of clock part using LGP on actual PCB of NS-RX231 board

5.2 Design of the power supply part

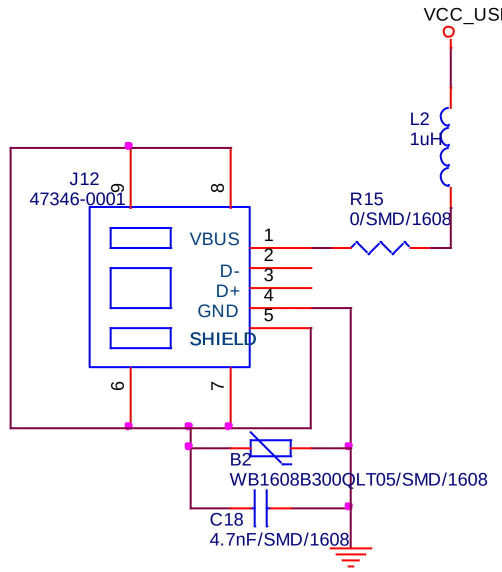
VCC_5V power supply 5V by using the voltage of 6.5V - 12V from the adapter of J7 through 7805, the regulation of 3 termination. In this case, 0.1uF and 47uF ceramic capacitor were used to prevent the high and low frequency noise in VIN. Consequently, the possibility of a short circuit due to a breakdown which occurs when use a tantalum capacitor is solved by using a ceramic capacitor. The high and low frequency characteristics are improved. A diode is used to prevent flowing current from VCC_5V to VCC_USB when the 5V from the adapter side is higher than 5V from the USB power side which can be occurred in connecting the adapter and the USB power at the same time. To minimize the voltage drop, I also used a schottky key barrier diode, so the V_F(Forward voltage) has just 0.2V, and limit to 500mA at the maximum value of the I_F so that it does not exceed the USB2.0 standard up to 500mA.



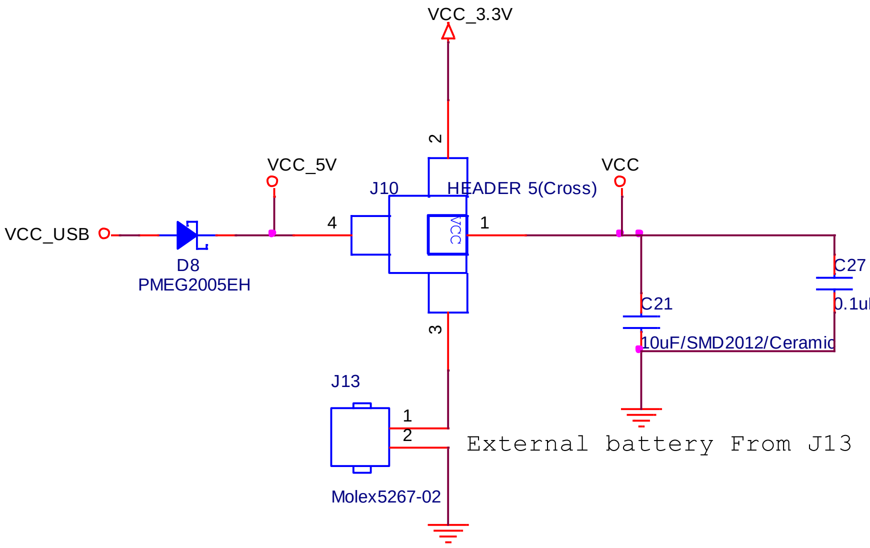
Example of a circuit design that converts from VIN(an adapter input) to VCC 5V



Example of circuit design that converts VCC_5V to VCC_3.3V



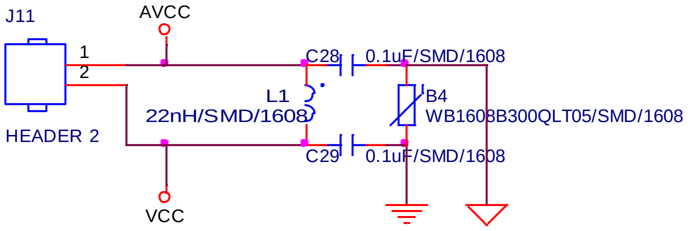
An example of a circuit design that creates VCC_USB from a USB powered source



An example of the jumper circuitry choosing VCC from VCC_5V, VCC_3.3V, outside battery

Jumper connection of J10	Power supply to supply
Connecttion of 1 and 2	DC 3.3V (VCC_3 .3V)
Connecttion of 1 and 3	Outside battery from J13
Connecttion of 1 and 4	5V (VCC_5V) from an adapter or USB

Method of the J10 jumper setting

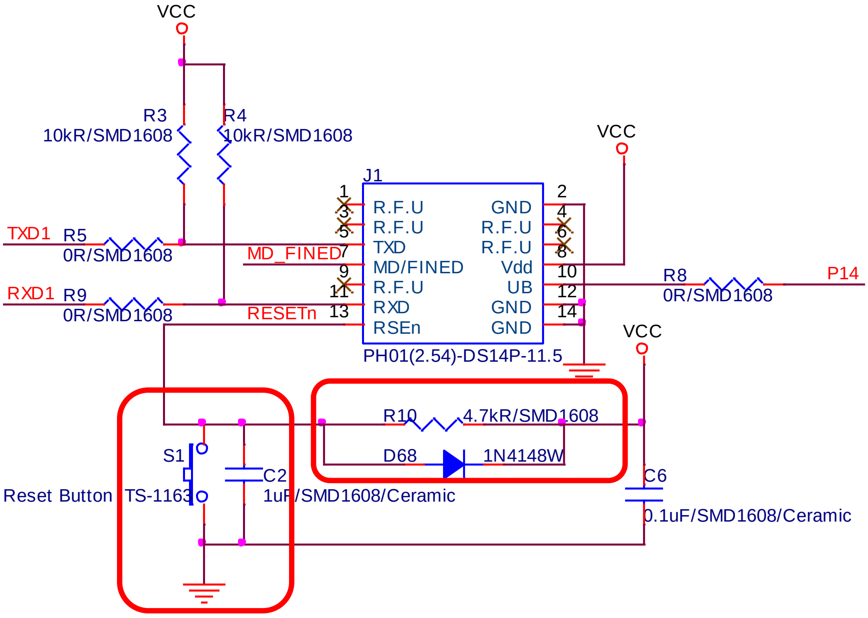


An example of Circuit Design to reduce Digital Noise from VCC to AVCC

5.3 Design of the power-on reset circuit / manual operation reset circuit

On page 1909 of HardwareUsersManual on RENESAS RX231, 50.3.2 Reset Timing is explained. It is stated here that the reset timing should be at least minimum 3 [ms] when the power is turned on. Therefore, we have implemented a generous 4.7 [ms].

An automatic reset circuit with reset timing of ┑ = R * C = 4.7 [ms] is designed using 1uF and 4.7kR. In addition, de-bounce, which eliminates high-frequency and low-frequency noise with C2 and C6 and eliminates noise, called chattering or bounce effect.



Examples of Automatic Reset Circuit and Manual Reset Circuit Design

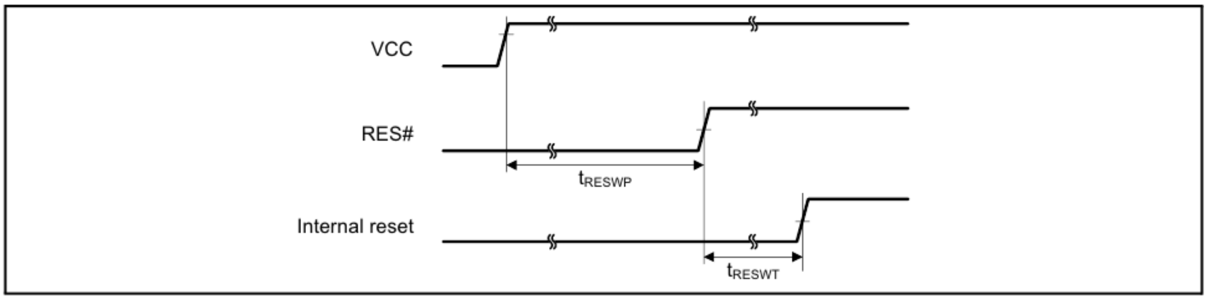
Reset Timing

Conditions: 1.8 V ≤ VCC = VCC_USB = AVCC0 ≤ 5.5 V, VSS = AVSS0 = VREFL0 = VSS_USB = 0 V, T_a = –40 to +105°C

	Item	Symbol	Min.	Typ.	Max.	Unit	Test Conditions
RES# pulse width	At power-on	t_{RESWP}	3	—	—	ms	Figure 50.31
	Other than above	t_{RESW}	30	—	—	μs	Figure 50.32
Wait time after RES# cancellation (at power-on)	At normal startup*1	t_{RESWT}	—	8.5	—	ms	Figure 50.31
	During fast startup time*2	t_{RESWT}	—	560	—	μs	
Wait time after RES# cancellation (during powered-on state)		t_{RESWT}	—	120	—	μs	Figure 50.32
Independent watchdog timer reset period		t_{RESWIW}	—	1	—	IWDT clock cycle	Figure 50.33
Watchdog timer reset period		t_{RESWWW}	—	4	—	PCLKB cycle	
Software reset period		t_{RESWSW}	—	1	—	ICLK cycle	
Wait time after independent watchdog timer reset cancellation*3		t_{RESWT2}	—	300	—	μs	
Wait time after watchdog timer reset cancellation*4		t_{RESWT2}	—	300	—	μs	
Wait time after software reset cancellation		t_{RESWT2}	—	170	—	μs	

[2]

Reset input timing at power-on



[3]

Reset input timing diagram at power-on

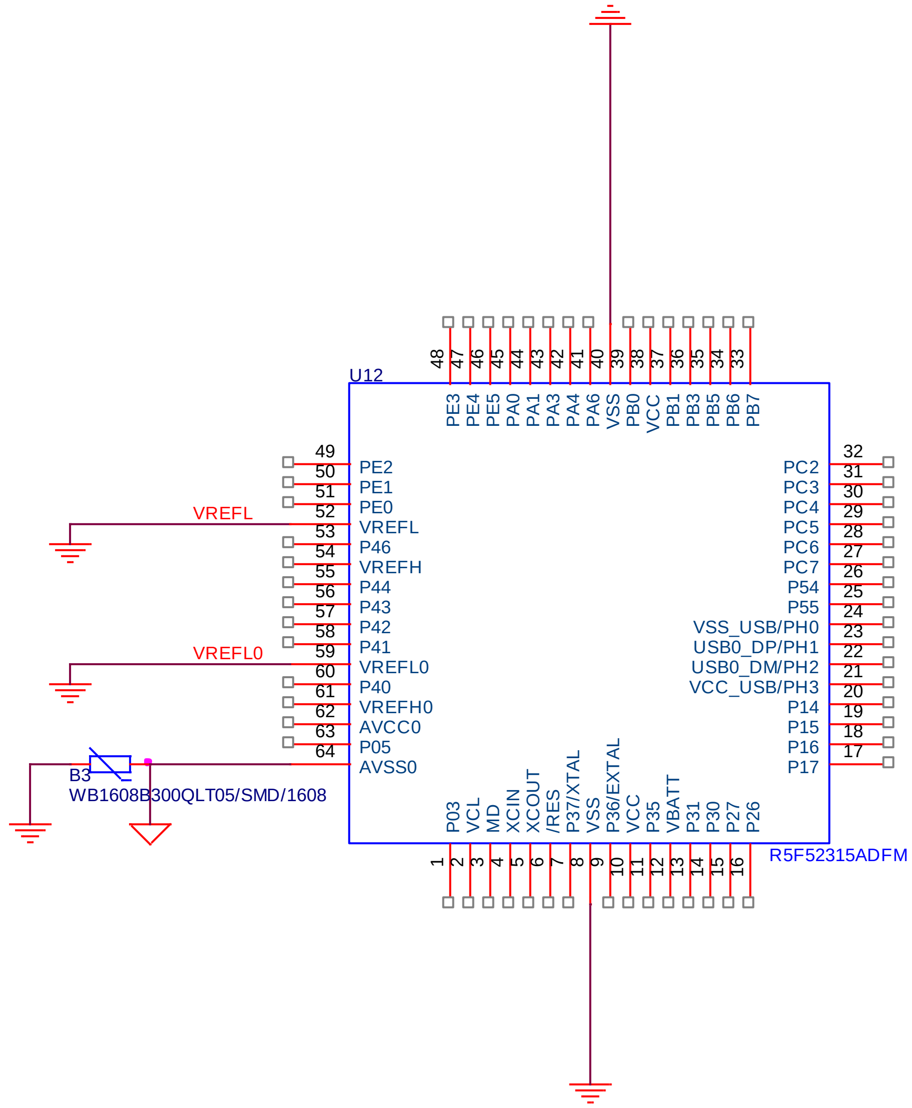
This upper picture shows the minimum value of the reset timing that t_{RESWP} set.

2 HardwareUsersManual of RX231, Page1909 Table 50.27

3 HardwareUsersManual of RX231, Page1909 Figure 50.31

5.4 Design of the grand wiring

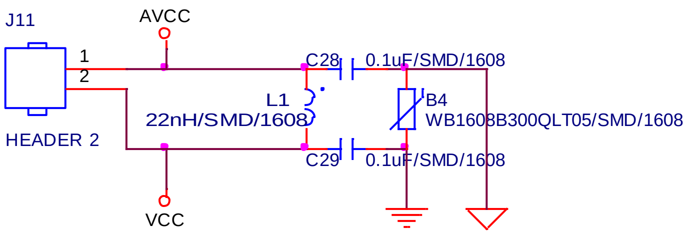
In order to prevent the noise of the digital ground VSS from being transmitted to the analog ground of AVSS0, it was connected using B3 ferrite bead. In addition, all VSSs on the chip are grounded to allow sufficient current to flow through all parts of the chip. The analog reference grounds VREFL and VREFL0 are both grounded. In this case, it is connected to the digital ground so that digital ground and common mode noise are not generated with the standard voltage. However, in some cases, if digital noise has a significant impact on the ADC, it would be suggested to connect VREFL and VREFL0 to analog ground to remove digital noise from the ADC pin. Since it is circuit of 10MHz or more, it is multi-grounded.



Example of Grounding Circuit Design in MCU

5.5 Design of the analog power supply (AVCC) and digital power supply (VCC)

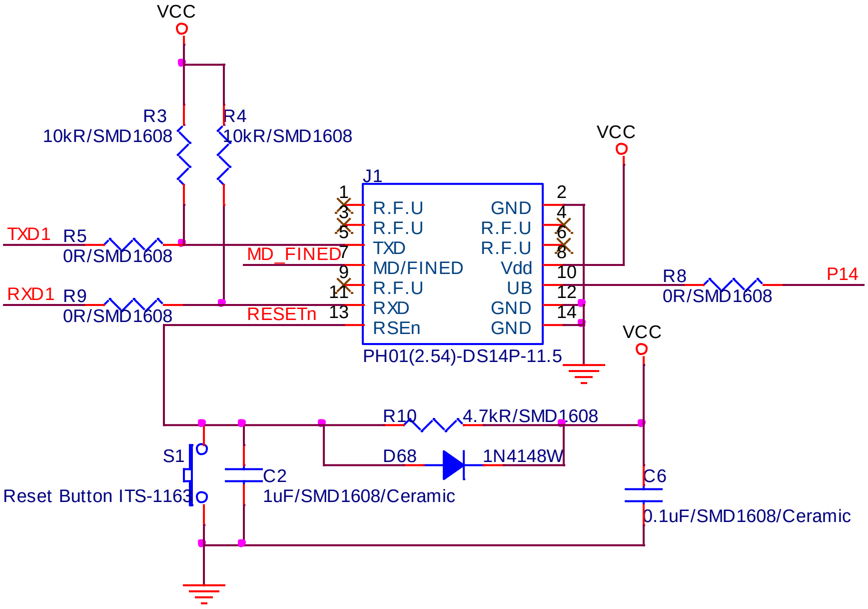
An inductor is connected between AVCC and VCC to prevent AC noise from entering, and DC voltage can be transmitted without attenuation. In addition, 0.1uF is placed between AVCC and analog ground to eliminate high frequency noise of AVCC, and 0.1uF between VCC and digital ground to eliminate high frequency noise of VCC. In addition, beads are placed between the analog ground and the digital ground so that they are connected together with noise removed.
In addition, J11 jumper is provided so that AVCC and VCC can be connected directly in a case. By default, the jumper is not connected at the time of a shipment from a factory, and the spare jumper is packed separately at the time of a board purchase.



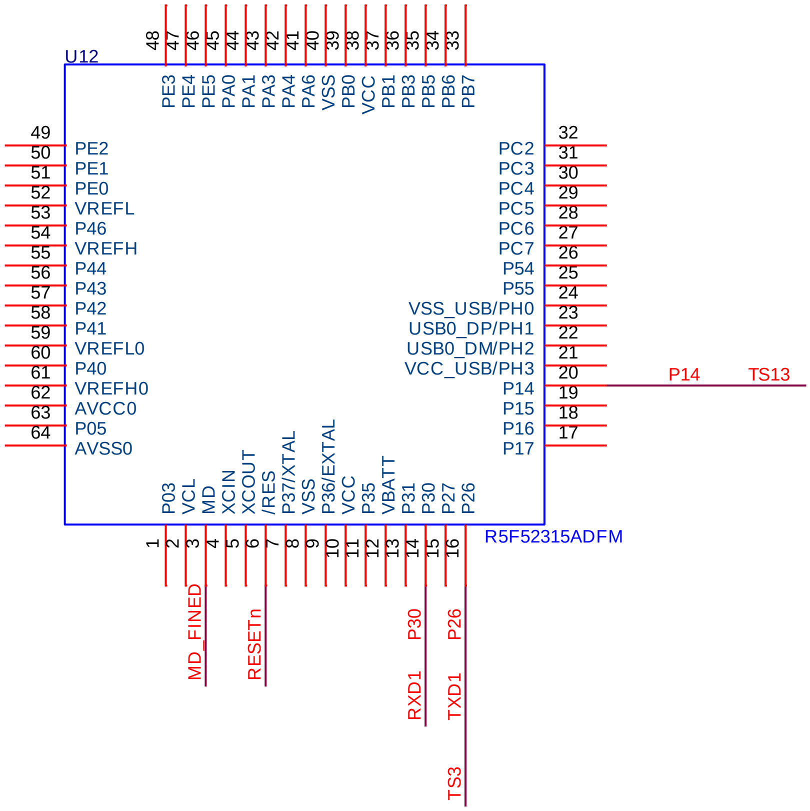
Example of Circuit Design to Prevent Digital Noise from VCC into AVCC

5.6 Connector diagram of the E1 JTAG debugger

E1 JTAG Debugger is connected as shown below.



Example of circuit design of E1 JTAG Debugger side



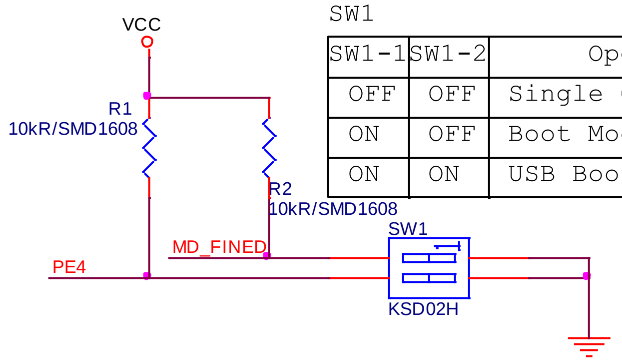
Example of E1 connection pin circuit design with RX231

5.7 A USB boot mode and power supply

The operation mode of default is all OFF, so it is shipped from a factory.in Single Chip/ DEBUG mode

SW1

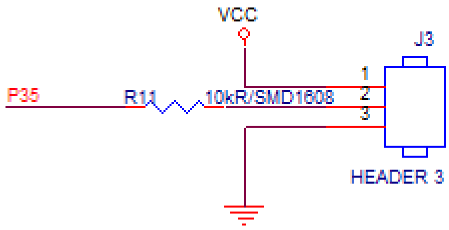
SW1-1	SW1-2	Operating Mode
OFF	OFF	Single Chip/DEBUG Mode
ON	OFF	Boot Mode (SCI)
ON	ON	USB Boot Mode



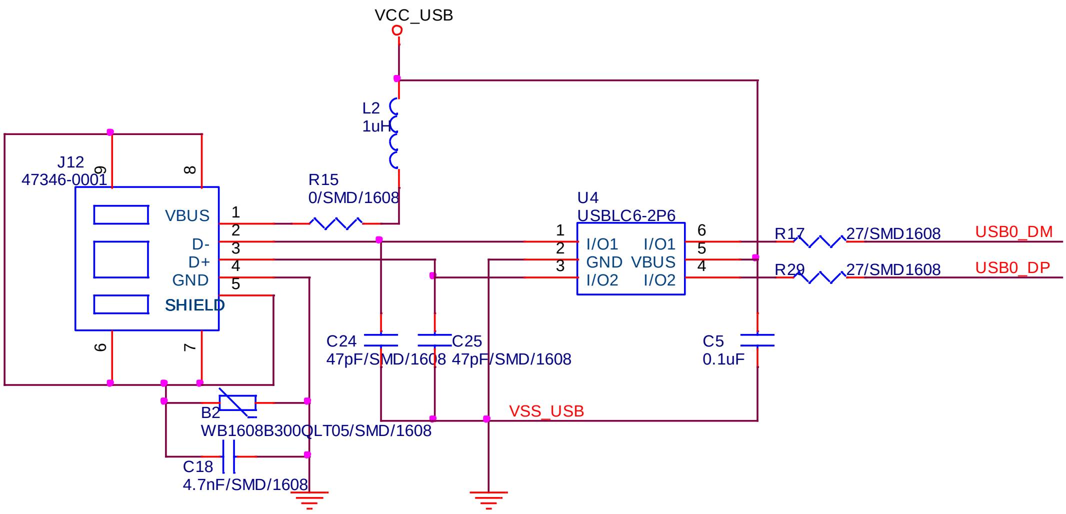
The default USB boot mode Power condition is jumper connected between 1-2.

USB Boot Mode Power Configuration (1)

J3	Power Configuration for USB Boot
1-2	Bus Powered
2-3	Self Powered



5.8 Designed to eliminate noise in electromagnet authentication of USB

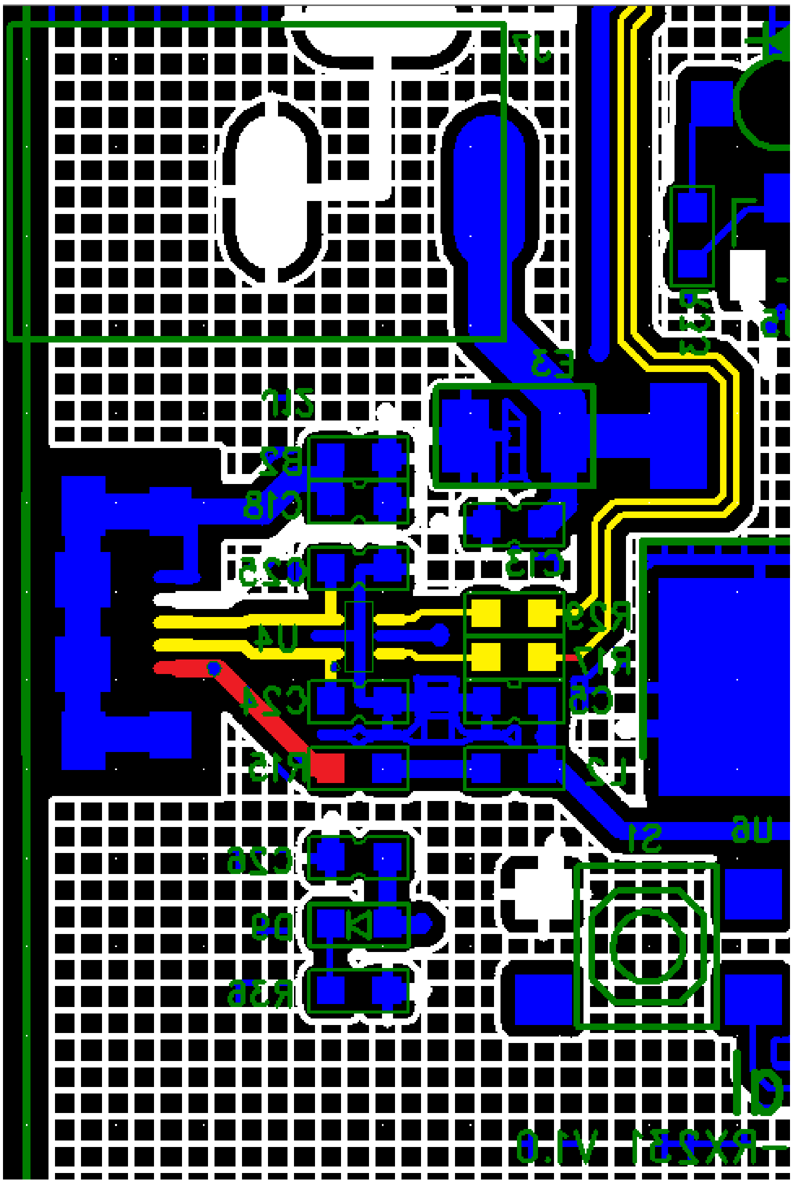


Example of circuit design considering elimination of noise in EMI/EMC of USB

VCC and GND are designed with inductors, beads and capacitors as shown above, and the data lines D + and D- use 47 pF to eliminate input noise and provide a high level of IEC61000-4-2 level4 compatibility at device level ESD protection circuitry to prevent ESD from entering the data. At the same time, for the case of the ESD comes directly into the USB connector, it is connected to be removed through the SHIELD. In addition, when it is lined together in parallel to MCU, even noise is induced, they will be induced together. If the values are subtracted from each other, noise is prevented from flowing by designing the same voltage as the starting point.

In addition, when designing the PCB circuit, the path is provided so that it can be flowed into the GND immediately when the noise is introduced, and the wide GND is laid out using the copper pour so that the noise is sufficiently absorbed into the GND and disappears. In addition, if there is a case, the external chassis of the USB and the conductive surface inside or outside are contacted through the conductive sponge, etc., the GND surface is wide and the ESD noise spreads widely.

.



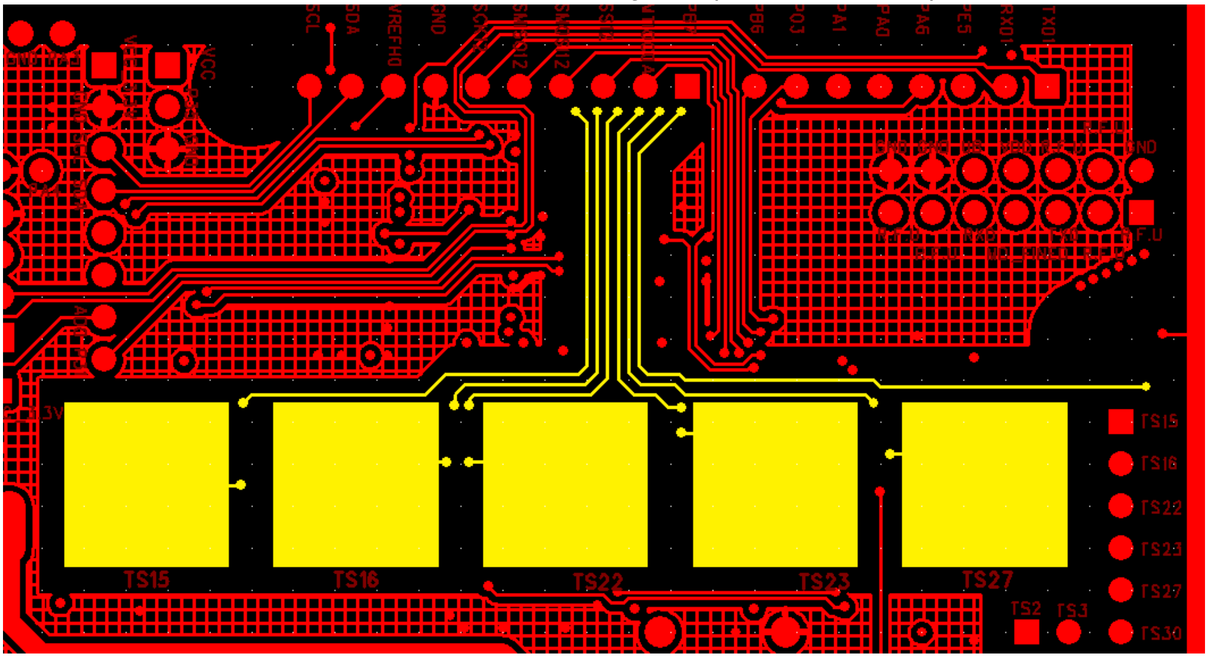
PCB design considering effective noise elimination for USB

5.9 Design of the capacitive touch sensor part

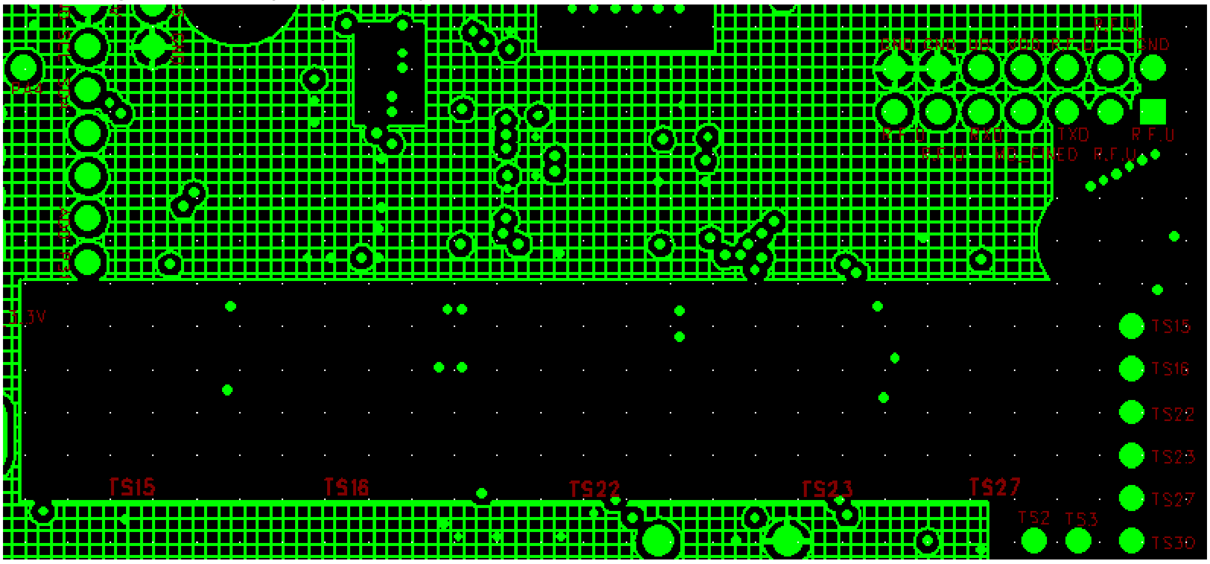
To improve the sensitivity of the cap touch, various techniques are used to minimize the capacitance.

1. Copper pouring of the GND pattern to the mesh GND has reduced the ground area so that the parasitic capacitance is low. The lower the parasitic capacitance, the more the capacitance at the time of touch.

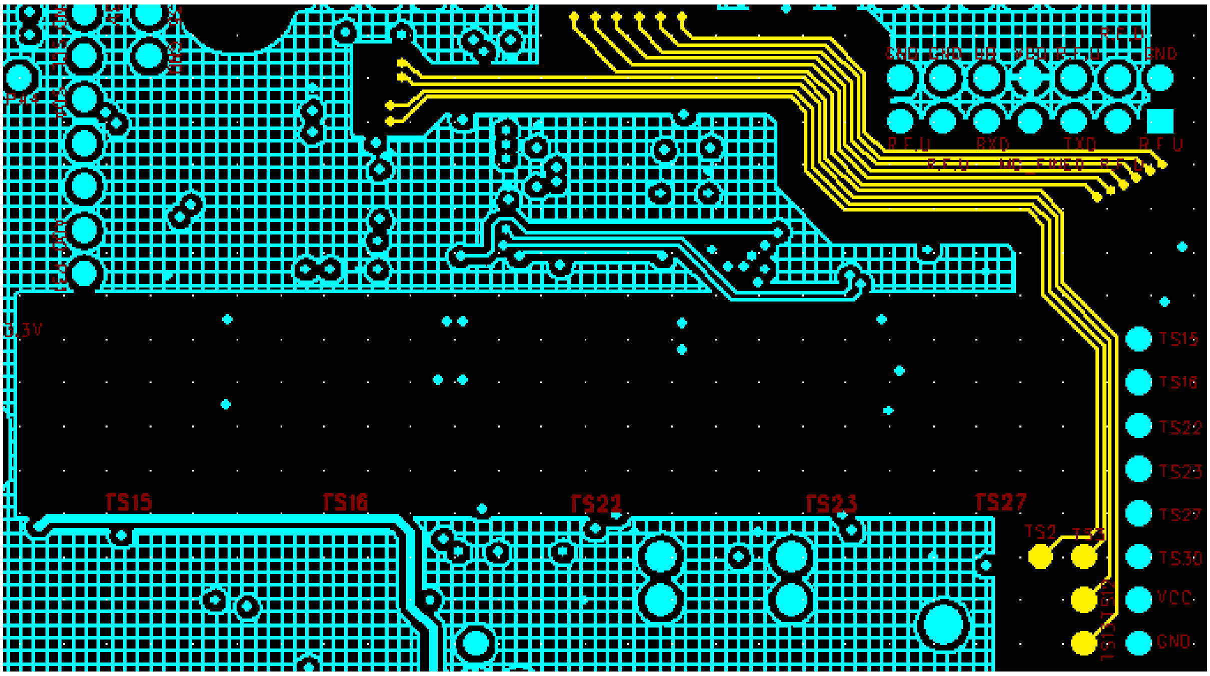
2. In Cap Touch design, all TS signal lines and mesh GND are set at least with 2mm clearance and at least 2mm between electrode and electrode.
3. In order to get the maximum Cap Touch sensitivity, PCB is designed to be 2.0 ~ 2.4mm thickness to make the Cap Touch button and the GND pattern on the bottom be as far as possible. This will reduce the parasitic capacitance because the distance between the cap touch of the TOP and the mesh GND of the BOTTOM is the same to the PCB thickness.
4. If possible, please make mesh GND on TS signal line, so it prevents a noise flowing from the outside.
5. Design to insert a series protection resistor (560Ω) to prevent ESD and protect the microcontroller.
6. R58, R59, R60, R61, R62, R63 are shipped as N.C (Not connected) because the connection to the connector for external connection is cut off to maximize the sensitivity of the currently designed Cap Touch button. Therefore, the reason for not connecting is not connected for the maximum sensitivity of five touch sensors (TS15, TS16, TS22, TS23, TS27) in the center of the board and proximity sensor (TS30) If you want to connect to an external device, connect the resistor to 0Ω. After connection, use the function of TS15, TS16, TS22, TS23, TS27 among multi pin functions to implement touch.



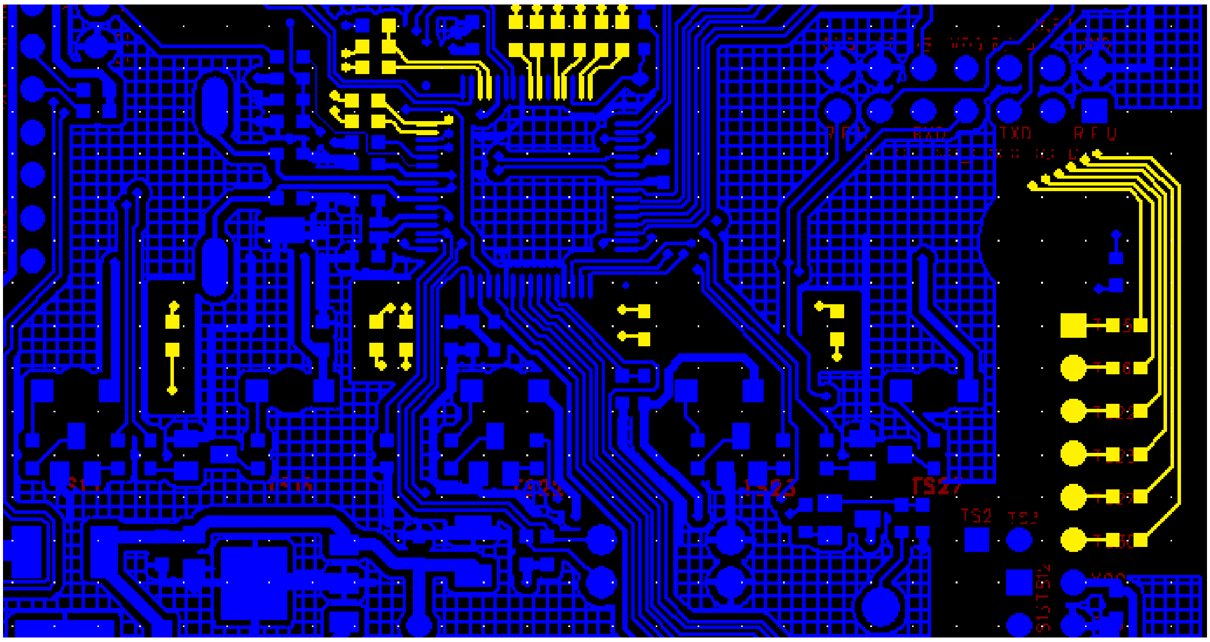
PCB design of TOP Layer (1st floor)



PCB design of Inner Layer 2 Layer



PCB design for Inner Layer 3 Layer(the third layer)

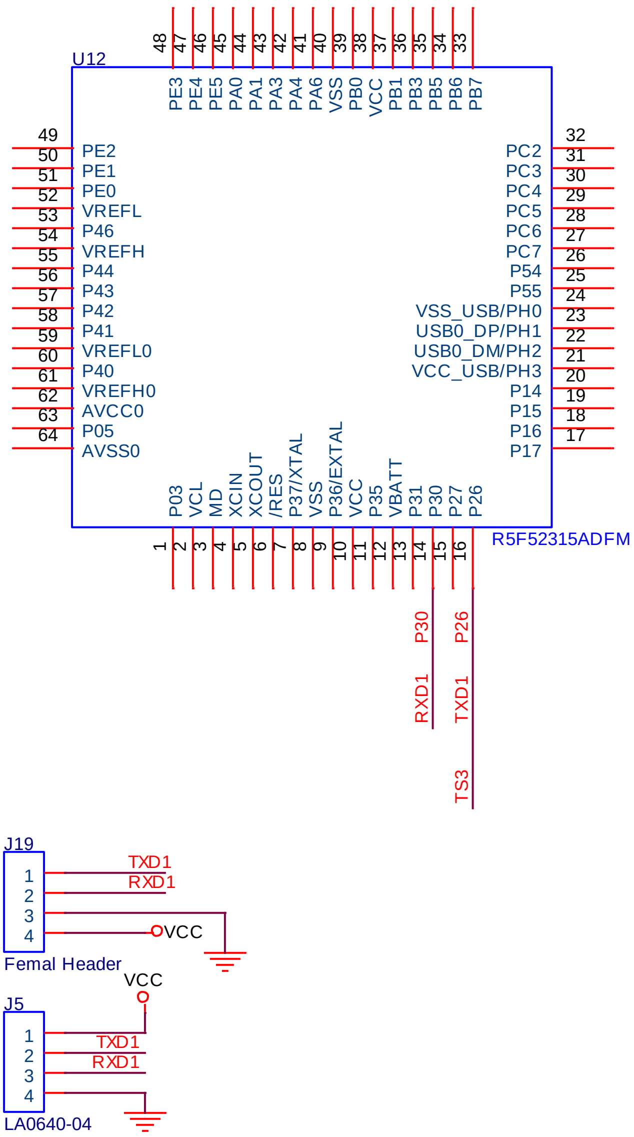


BOTTOM Layer (4th floor) PCB Design

SiAL

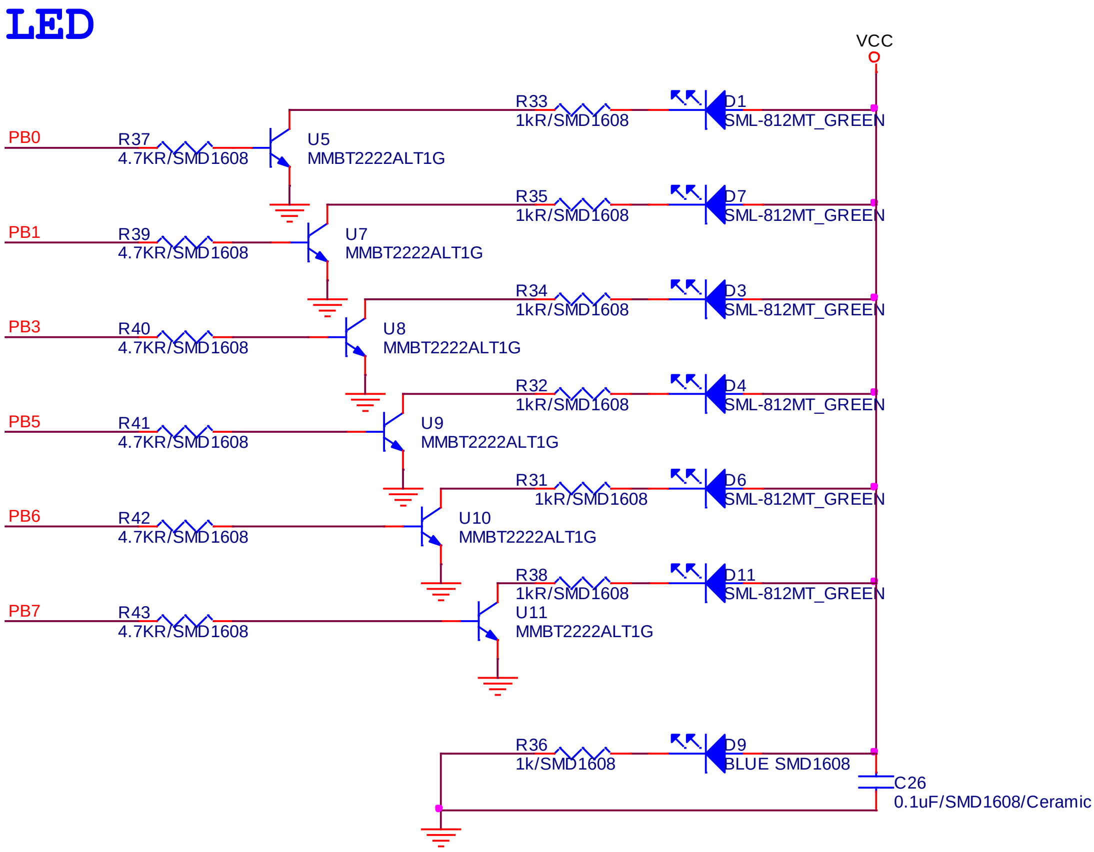
5.10 Design of the UART-related part

As shown below, UART is used as UART and J19 is used by connecting Bluetooth Module (HC-06).



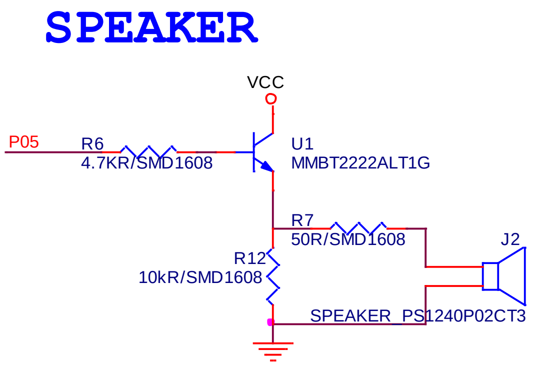
5.11 Design of the LED part for the test

Since the allowable current per pin of RX231 is relatively low, about 5mA, the transistors are used to control the LEDs to reduce the load on the microcontroller. On this board, we use NPN transistor to turn on the LED when pin output is 1, and turn off when it is 0.



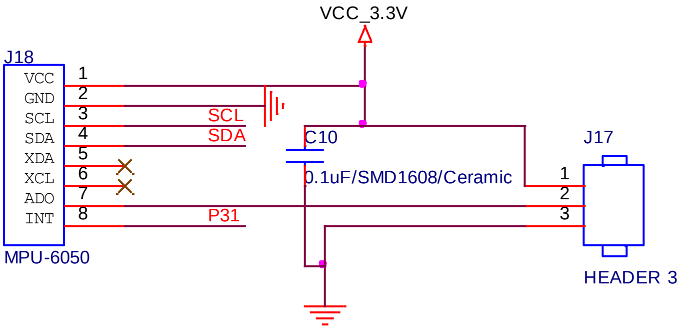
5.12 Design of driving part of speaker

P05 is to control port output, allowing the timer to drop using a constant frequency. P05 is a multifunction pin with DA1. In other words, you can output to the DA converter if necessary.



5.13 Design of MPU-6050 6-axis acceleration / gyro sensor module connector

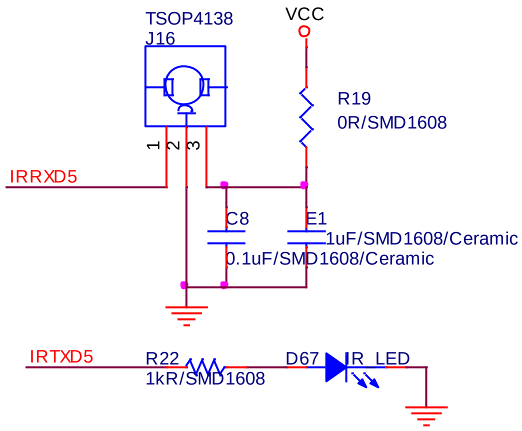
The MPU-6050 module is a 6-axis acceleration / gyro module controlled by I2C (RIIC). Also, since 3.3V is used as the power source, VCC_3.3V is connected.



5.14 Design of Infrared (IR) Remote Control Transmitter and Receiver

It is a remote-control transmission using infrared rays and reception department circuit diagram. Please refer to a software guide for the applicable sauce of a transmission department and the reception department.

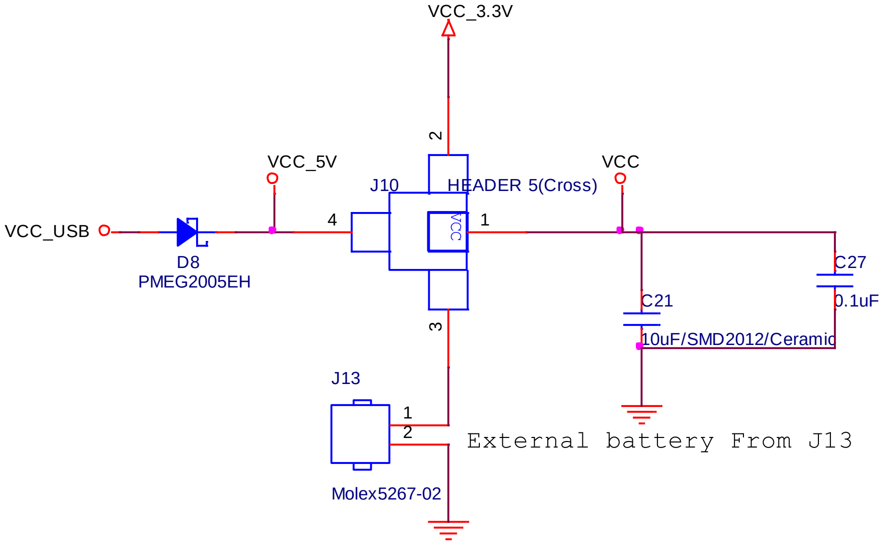
Infra Red RX / TX



5.15 Design of Power Selection Switch

The power supply comes to generate VCC_5V from VCC_USB or an adapter and, the power supply is designed to generate VCC_5V at VCC_USB or an adapter.

Respectively the possibility of a short circuit due to a breakdown occurs in using a tantalum capacitor, which is solved by using a ceramic capacitor. The high frequency characteristics are improved. A diode is used to prevent VCC_5V from going to VCC_USB when the 5V made at the adapter side is higher than 5V of the USB power in connecting the adapter and the USB power at the same time. And the V_F(Forward voltage) has just 0.2V, and using a diode that can be used up to 500mA at the maximum limit of the I_F of the diode so that it does not exceed the USB2.0 standard up to 500mA. Therefore, this circuit safely protects the PC when connected to a USB of PC.



Example of the jumper design to choose VCC among VCC_5V, VCC_3.3V, outside battery

Jumper connection of J10	Power supply to supply
I connect 1 and 2	DC 3.3V (VCC_3 .3V)
I connect 1 and 3	Outside battery from J13
I connect 1 and 4	5V (VCC_5V) from an adapter or USB

J10 jumper setting method