FSC-BT9104DI

DATASHEET V1.2

Revision History

Version	Data	Notes	Author
V1.0	2024-10-25	Initial Version	Liu
V1.1	2025-01-06	1, Change the Supply Voltage 2, Update Tx Power	Liu
V1.2	2025-11-06	1, Update Tx POWER 2, Modify packaging information 3, Update the block diagram	Liu

Contact Us

Shenzhen Feasycom Co., LTD

Email: sales01@feasycom.com

Address: Rm 508, Building A, Fenghuang Zhigu, No.50, Tiezai Road, Xixiang, Baoan District, Shenzhen, 518100, China.
Tel: 86-755-27924639

1 INTRODUCTION

Overview

The FSC-BT9104DI module supports Bluetooth dual-mode V5.3 (BR/EDR/BLE) and comes equipped with Feasycom's powerful, easy-to-use firmware. It's easy to use and completely encapsulated, allowing access to Bluetooth functionality through simple ASCII commands over a serial interface, much like a Bluetooth modem.

The FSC-BT9104DI module is an ideal solution for developers who want to integrate Bluetooth wireless technology into their designs.

Features

- **Supporting feature**
  - Standard configurable RF transmission power -20dBm to 2dBm
  - Low Power
  - Supports 1Mbps/2Mbps/3Mbps BR/EDR
  - Supports 1Mbps/2Mbps LE
  - Supports long-distance transmission of LE 125Kbps/500Kbps
  - RSSI is supported
  - Dynamic TX power control
  - Bluetooth host protocol stack implemented by another MCU or host AP
- **RF Specification**
  - RX Receiving sensitivity -88dBm (BR 1Mbps)
  - RX Receiving sensitivity -90dBm (EDR 2Mbps)
  - RX Receive sensitivity -83dBm (EDR 3Mbps)
  - RX Receive sensitivity -90dBm (LE 1Mbps)
  - RX Receiving sensitivity -88dBm (LE 2Mbps)
  - RX Receiving sensitivity -93dBm (LE LR2 1Mbps)
  - RX Receiving sensitivity -96dBm (LE LR8 1Mbps)
  - TX Transmit power -20 dBm to 2dBm

Application

- Bluetooth KEY
- OBU

- Smart home
- data transmission module
- Dual-mode Bluetooth HCI controller
- High security and reliability of Bluetooth MCU applications

2 General Specification

Table 2-1： General Specifications

Categories	Features	Implementation
Bluetooth		
	Bluetooth Standard	Bluetooth V5.3
	Frequency Band	2402MHz ~ 2480MHz
	Interface	UART/PIO
	Transmit Power	Up to 2dBm
	Receiver	-90dBm（EDR 2Mbps）
Size		10.8mm × 13.5mm × 2.3mm
Operating temperature		-40°C ~+85°C
Storage temperature		-40°C ~+85°C
Supply Voltage		3.3V~3.6V
Miscellaneous	Lead Free Warranty	Lead-free and RoHS compliant One Year
Humidity		10% ~ 90% non-condensing
MSL grade		MSL 3
ESD grade		Human Body Model: Pass ±2000 V Charge device model: Pass ±500 V
Dimension		10.8mm(W) x 13.5mm(L) x 1.7mm(H) (without shielding cover) 10.8mm(W) x 13.5mm(L) x 2.3mm(H) (with shielding cover)

3 HARDWARE SPECIFICATION

3.1 Block Diagram and PIN Diagram

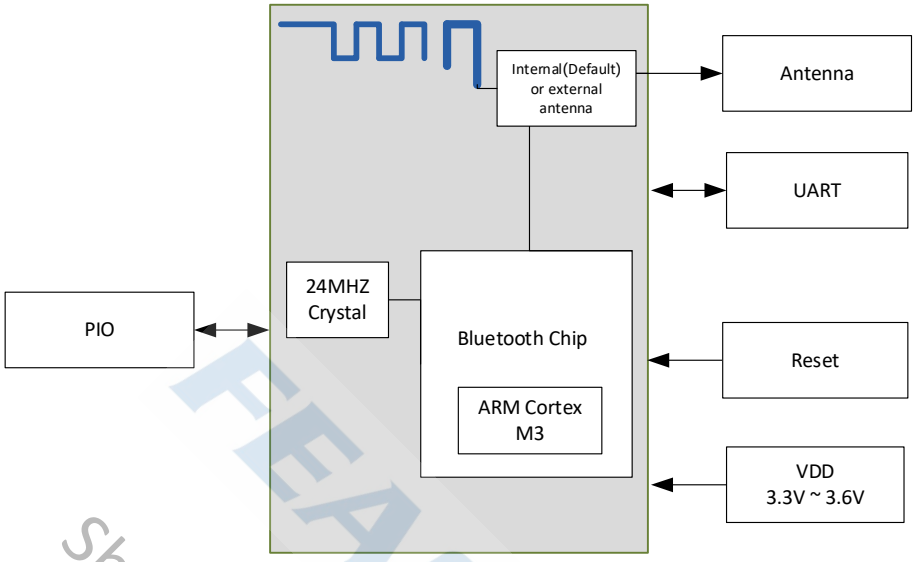

Figure 3-1-1: FSC-BT9104DI Block Diagram

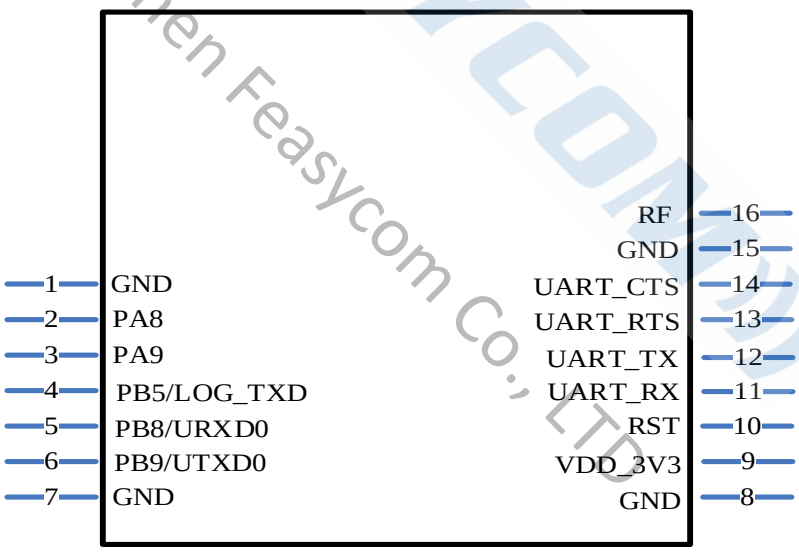

Figure 3-1-2: FSC-BT9104DI PIN Diagram (Top View)

3.2 PIN Definition Descriptions

LGA package, as shown in the figure below.

Figure 3-2-1: FSC-BT9104DI Package Module Appearance

Table 3-2： Pin definition

Pin	Pin Name	Type	Pin Descriptions	Notes
1	GND	VSS	Power Ground	
2	PA8	I/O	Programmable I/O	
3	PA9	I/O	Programmable I/O **Alternative Function 1: BT Status**	
4	PB5/LOG_TXD	I/O	LOG output	
5	PB8/URXD0	I	UART_RXD0(FW default)	
6	PB9/UTXD0	O	UART_TXD0(FW default)	
7	GND	VSS	Power Ground	
8	GND	VSS	Power Ground	
9	VDD_3V3	VDD	Power supply voltage 3.3V	
10	RST	I	External reset input：Active LOW	
11	UART_RX	CMOS input	UART data input	
12	UART_TX	CMOS output	UART data output	
13	UART_RTS	CMOS output	UART request to send active low	
14	UART_CTS	CMOS input	UART clear to send active low	
15	GND	VSS	Power Ground	
16	RF	RF_IN/ OUT	Bluetooth 50ohm transmitter output/receiver input NC if using internal antenna(default) Connect to external antenna(optional)	

4 PHYSICAL INTERFACE

4.1 UART Interface

The FSC-BT9104DI module features a standard 4-wire UART interface with RX and TX, supporting both the H4 HCI interface and raw UART. The default baud rate is 921.6k baud. To accommodate various speed requirements, the FSC-BT9104DI offers multiple UART clock options, enabling flexibility in both high and low-speed baud rates.

Table 4-1： Possible UART Settings

Parameter	Possible Values
Baudrate	Minimum - default 921600bps
Flow control	-
Parity	None, Odd or Even
Number of stop bits	1
Bits per channel	8

5 MSL & ESD

Table 5-1: MSL and ESD

Parameter	Value
MSL grade	MSL 3
ESD grade	Electrostatic discharge
ESD – Human-body model (HBM) rating, JESD22-A114-F (Total samples from one wafer lot)	Pass ±2000 V, all pins
ESD – Charge-device model (CDM) rating, JESD22-C101-D (Total samples from one wafer lot)	Pass ±500 V, all pins

6 RECOMMENDED TEMPERATURE REFLOW PROFILE

Prior to any reflow, it is important to ensure the modules were packaged to prevent moisture absorption. New packages contain desiccate (to absorb moisture) and a humidity indicator card to display the level maintained during storage and shipment. If directed to bake units on the card, please check the below and follow instructions specified by IPC/JEDEC J-STD-033.

Note: The shipping tray cannot be heated above 65°C. If baking is required at the higher temperatures displayed in the below, the modules must be removed from the shipping tray.

Any modules not manufactured before exceeding their floor life should be re-packaged with fresh desiccate and a new humidity indicator card. Floor life for MSL (Moisture Sensitivity Level) 3 devices is 168 hours in ambient environment 30°C/60%RH.

Notice (注意) :

Feasycom module must use Step-Stencil, suggestion using the stencil thickness about 0.16-0.2mm,it could be modify with the product.

Table 6-1: Recommended baking times and temperatures

MSL	125°C Baking Temp.		90°C/≤ 5%RH Baking Temp.		40°C/ ≤ 5%RH Baking Temp.	
	Saturated @ 30°C/85%	Floor Life Limit + 72 hours @ 30°C/60%	Saturated @ 30°C/85%	Floor Life Limit + 72 hours @ 30°C/60%	Saturated @ 30°C/85%	Floor Life Limit + 72 hours @ 30°C/60%
3	9 hours	7 hours	33 hours	23 hours	13 days	9 days

Feasycom surface mount modules are designed to be easily manufactured, including reflow soldering to a PCB. Ultimately it is the responsibility of the customer to choose the appropriate solder paste and to ensure oven temperatures during reflow meet the requirements of the solder paste. Feasycom surface mount modules conform to J-STD-020D1 standards for reflow temperatures.

The soldering profile depends on various parameters necessitating a set up for each application. The data here is given only for guidance on solder reflow.

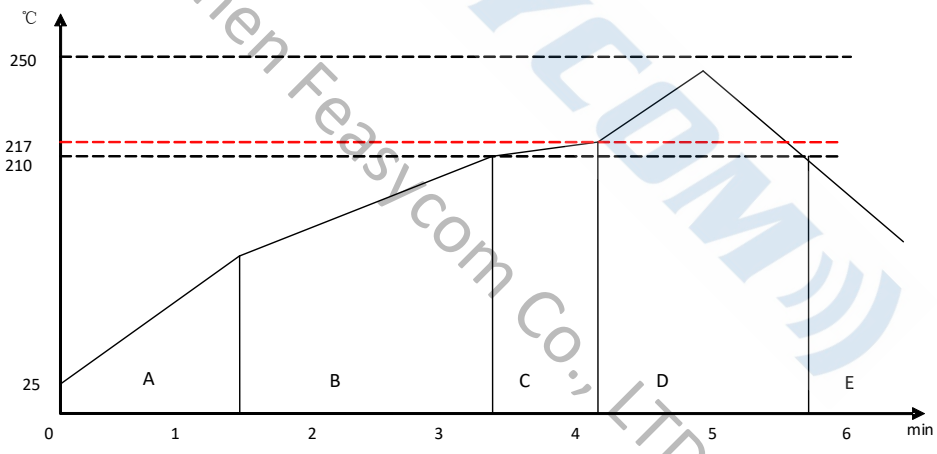

Figure 6-1： Typical Lead-free Re-flow

Pre-heat zone (A) — This zone raises the temperature at a controlled rate, **typically 0.5 – 2 °C/s**. The purpose of this zone is to preheat the PCB board and components to 120 ~ 150 °C. This stage is required to distribute the heat uniformly to the PCB board and completely remove solvent to reduce the heat shock to components.

Equilibrium Zone 1 (B) — In this stage the flux becomes soft and uniformly encapsulates solder particles and spread over PCB board, preventing them from being re-oxidized. Also with elevation of temperature and liquefaction of flux, each activator and rosin get activated and start eliminating oxide film formed on the surface of each solder particle and PCB board. **The temperature is recommended to be 150° to 210° for 60 to 120 second for this zone.**

Equilibrium Zone 2 (C) (optional) — In order to resolve the upright component issue, it is recommended to keep the temperature in 210 – 217 ° for about 20 to 30 second.

Reflow Zone (D) — The profile in the figure is designed for Sn/Ag3.0/Cu0.5. It can be a reference for other

lead-free solder. The peak temperature should be high enough to achieve good wetting but not so high as to cause component discoloration or damage. Excessive soldering time can lead to intermetallic growth which can result in a brittle joint. The recommended peak temperature (Tp) is 230 ~ 250 °C. The soldering time should be 30 to 90 second when the temperature is above 217 °C.

Cooling Zone (E) — The cooling ate should be fast, to keep the solder grains small which will give a longer-lasting joint. **Typical cooling rate should be 4 °C.**

7 MECHANICAL DETAILS

7.1 Mechanical Details

- Dimension: 10.8mm(W) x 13.5mm(L) x 1.7mm(H) Tolerance: ±0.2mm (without shielding cover)
  10.8mm(W) x 13.5mm(L) x 2.3mm(H) Tolerance: ±0.2mm (with shielding cover)
- Module size: 10.8mm X 13.5mm Tolerance: ±0.2mm
- Pad size: 1mmX0.6mm Tolerance: ±0.1mm
- Pad pitch: 1.0mm Tolerance: ±0.1mm

(分板后边角残留板边误差：≤ 0.5mm) (Residual plate edge error: ≤ 0.5mm)

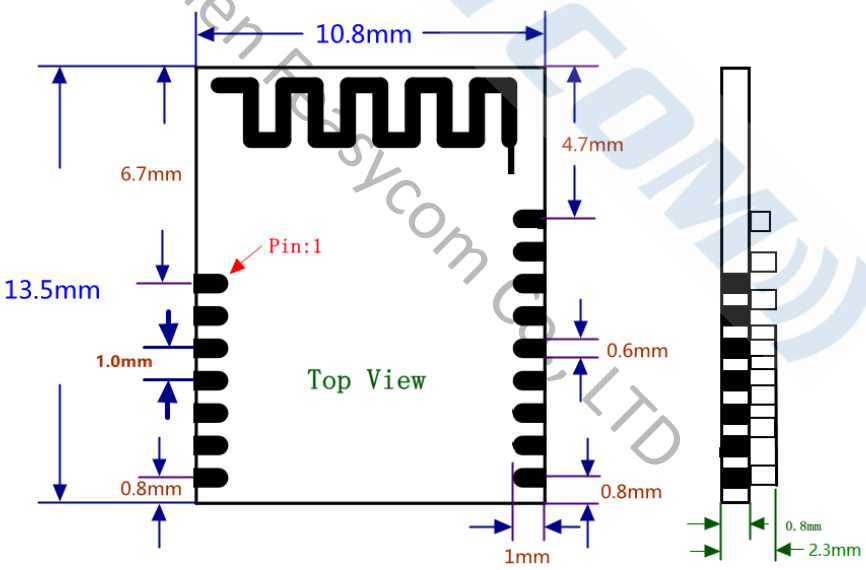

Figure 7-1： FSC-BT9104DI footprint Layout Guide (Top View)

8 HARDWARE INTEGRATION SUGGESTIONS

8.1 Soldering Recommendations

The FSC-BT9104DI is compatible with industrial standard reflow profile for Pb-free solders. The reflow profile used is dependent on the thermal mass of the entire populated PCB, heat transfer efficiency of the oven and particular type of solder paste used. Consult the datasheet of particular solder paste for profile configurations.

Feasycom will give following recommendations for soldering the module to ensure reliable solder joint and operation of the module after soldering. Since the profile used is process and layout dependent, the optimum profile should be studied case by case. Thus following recommendation should be taken as a starting point guide.

8.2 Layout Guidelines (Internal Antenna)

It is strongly recommended to use good layout practices to ensure proper operation of the module. Placing copper or any metal near antenna deteriorates its operation by having effect on the matching properties. Metal shield around the antenna will prevent the radiation and thus metal case should not be used with the module. Use grounding vias separated max 3 mm apart at the edge of grounding areas to prevent RF penetrating inside the PCB and causing an unintentional resonator. Use GND vias all around the PCB edges.

The mother board should have no bare conductors or vias in this restricted area, because it is not covered by stop mask print. Also no copper (planes, traces or vias) are allowed in this area, because of mismatching the on-board antenna.

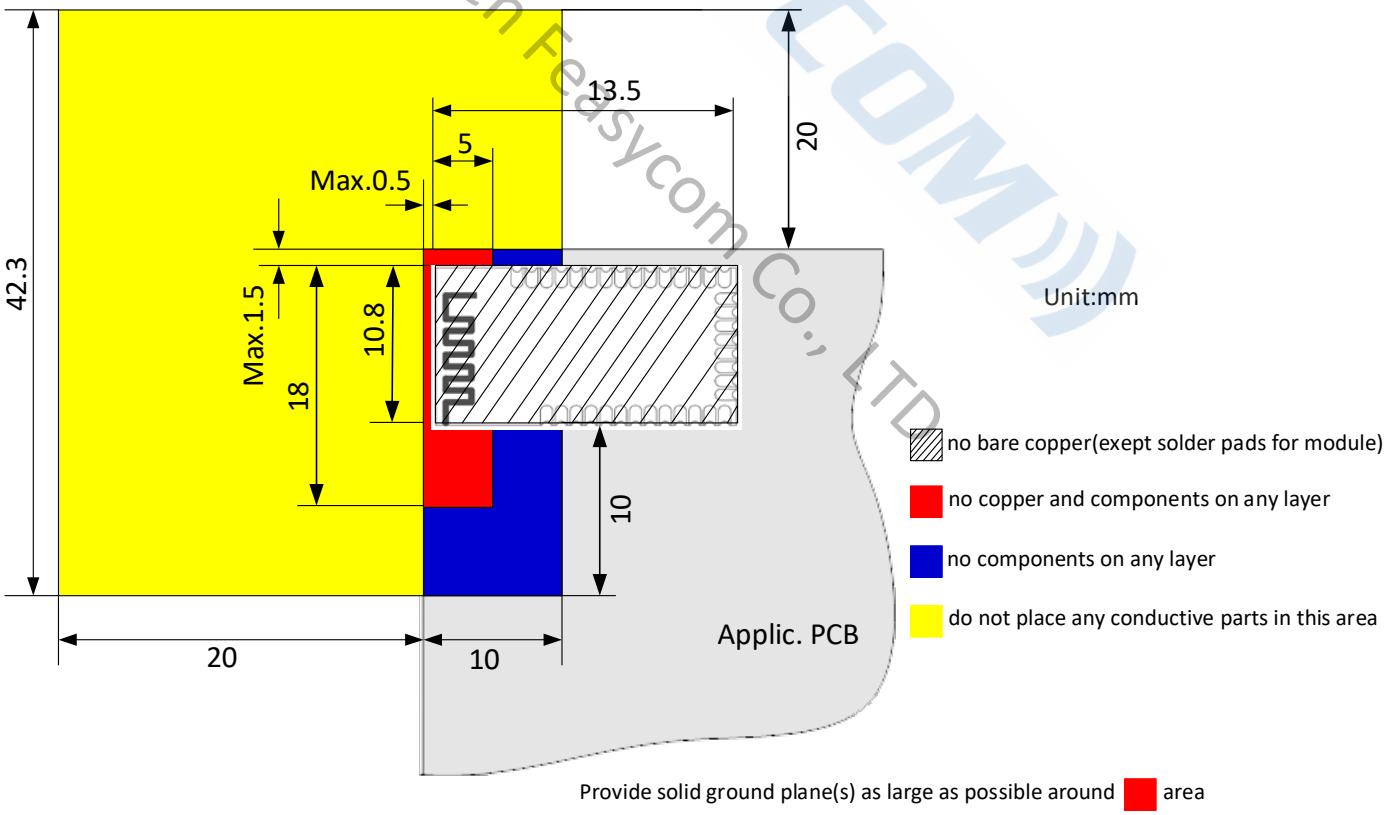

Figure 8-2: Restricted Area (Design schematic, for reference only. Unit: mm)

Following recommendations helps to avoid EMC problems arising in the design. Note that each design is unique

and the following list do not consider all basic design rules such as avoiding capacitive coupling between signal lines. Following list is aimed to avoid EMC problems caused by RF part of the module. Use good consideration to avoid problems arising from digital signals in the design.

Ensure that signal lines have return paths as short as possible. For example if a signal goes to an inner layer through a via, always use ground vias around it. Locate them tightly and symmetrically around the signal vias. Routing of any sensitive signals should be done in the inner layers of the PCB. Sensitive traces should have a ground area above and under the line. If this is not possible, make sure that the return path is short by other means (for example using a ground line next to the signal line).

8.3 Layout Guidelines (External Antenna)

Placement and PCB layout are critical to optimize the performances of a module without on-board antenna designs. The trace from the antenna port of the module to an external antenna should be 50Ω and must be as short as possible to avoid any interference into the transceiver of the module. The location of the external antenna and RF-IN port of the module should be kept away from any noise sources and digital traces. A matching network might be needed in between the external antenna and RF-IN port to better match the impedance to minimize the return loss.

As indicated in below, RF critical circuits of the module should be clearly separated from any digital circuits on the system board. All RF circuits in the module are close to the antenna port. The module, then, should be placed in this way that module digital part towards your digital section of the system PCB.

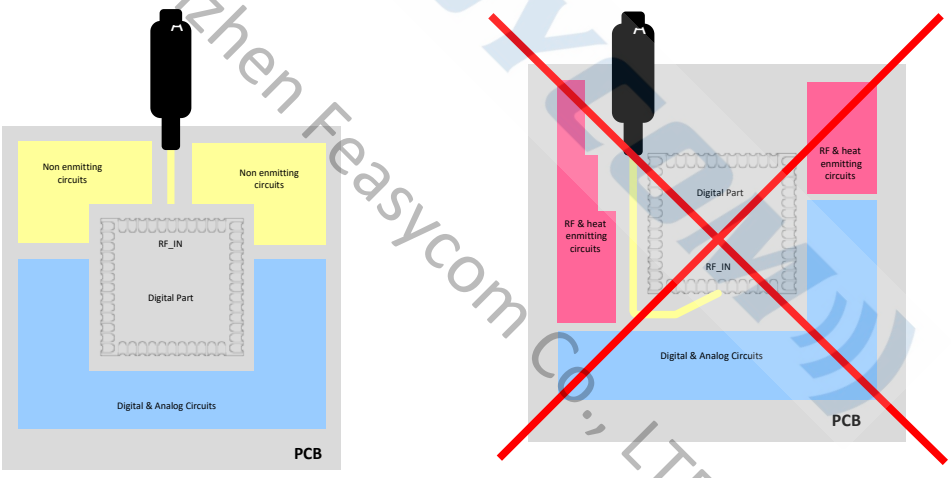

Figure 8-3: Placement the Module on a System Board

8.3.1 Antenna Connection and Grounding Plane Design

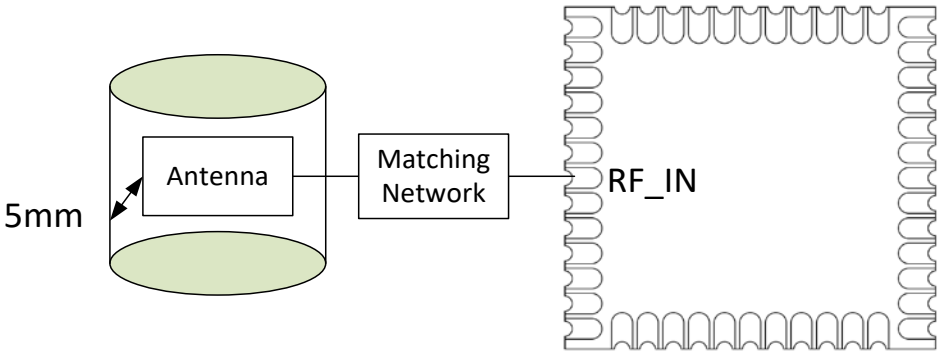

Figure 8-3-1: Leave 5mm Clearance Space from the Antenna

General design recommendations include

- The length of the trace or connection line should be kept as short as possible.
- Distance between connection and ground area on the top layer should at least be as large as the dielectric thickness.
- Routing the RF close to digital sections of the system board should be avoided.
- To reduce signal reflections, sharp angles in the routing of the micro strip line should be avoided. Chamfers or fillets are preferred for rectangular routing; 45-degree routing is preferred over Manhattan style 90-degree routing.

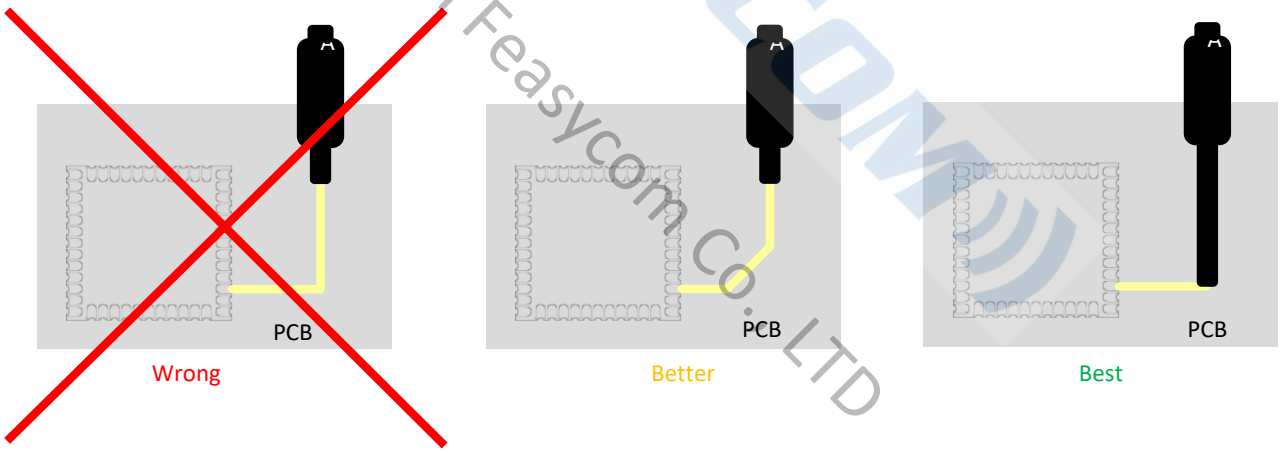

Figure 8-3-2: Recommended Trace Connects Antenna and the Module

- Routing of the RF-connection underneath the module should be avoided. The distance of the micro strip line to the ground plane on the bottom side of the receiver is very small and has huge tolerances. Therefore, the impedance of this part of the trace cannot be controlled.
- Use as many vias as possible to connect the ground planes.

9 PRODUCT PACKAGING INFORMATION

9.1 Default Packing

a, Tray vacuum

b, Tray Dimension: 210mm * 180mm

Figure 9-1: Vacuum Tray

9.2 Packing box (Optional)

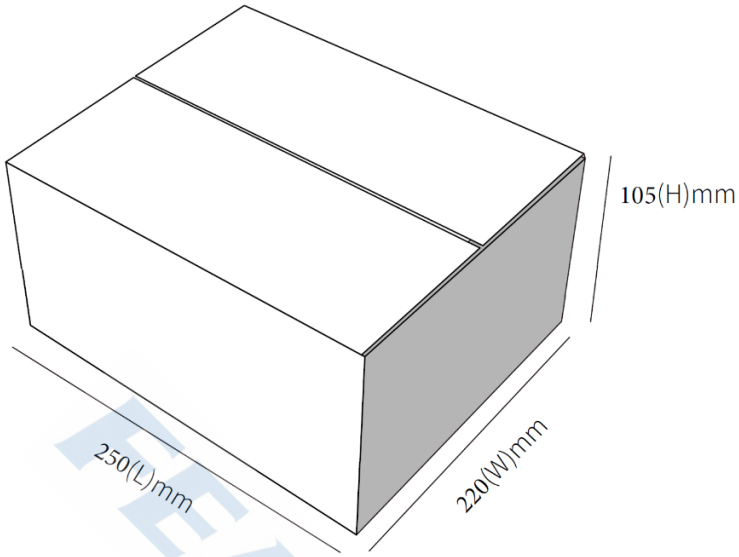

Figure 9-2: Packing box(Optional)

** If other packing is required, please confirm with the customer*

** Packing: 1000pcs per carton (Minimum packing quantity)*

** The outer packing size is for reference only, please refer to the actual size*

10 APPLICATION SCHEMATIC