FSC-BT2024NI

DATASHEET V1.0

Revision History

Version	Data	Notes	Author
V1.0	2025-08-19	Initial Version	Marsh

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

FSC-BT2024NI supports Bluetooth Low Energy, and IEEE 802.15.4 for Thread and Zigbee protocols. It also allows the implementation of proprietary 2.4 GHz protocols.

By default, FSC-BT2024NI module is equipped with powerful and easy-to-use Feasycom firmware. It's easy to use and completely encapsulated. Feasycom firmware enables users to access Bluetooth functionality with simple ASCII commands delivered to the module over serial interface -it's just like a Bluetooth modem.

Therefore, FSC-BT2024NI provides an ideal solution for developers who want to integrate Bluetooth wireless technology into their design.

Features

- Bluetooth 6.0, IEEE 802.15.4-2020, and 2.4 GHz enabled transceiver
  - Up to +8 dBm configurable output power, 1 dB step size from -8 dBm to +8 dBm
  - Estimated -96 dBm sensitivity in 1 Mbps Bluetooth Low Energy mode
  - Estimated -104 dBm sensitivity in 125 kbps Bluetooth Low Energy mode(long range)
  - Estimated -101 dBm sensitivity in IEEE 802.15.4, 20 byte packet length
- Supported data rates
  - Bluetooth 6.0--2 Mbps, 1 Mbps, 500 kbps, and 125 kbps
  - IEEE 802.15.4-2020--250 kbps
  - Proprietary 2.4 GHz-4 Mbps, 2 Mbps, and 1 Mbps
- Angle of Arrival (AoA) and Angle of Departure (AoD) direction finding using Bluetooth Low Energy
- Support for NFC（Near field communication）peripherals
- 1.8 V to 3.6 V supply and I/O voltage
- PCB antenna

Application

- Asset tracking
- Lighting control
- Smart home sensors and actuators
- Gateways and hubs
- Wearable health and fitness monitoring

2 General Specification

Table 2-1：General Specifications

Categories	Features	Implementation
Bluetooth	Chip	nRF54L15
	Bluetooth	Bluetooth 6.0
	Frequency Band	2402MHz ~ 2480MHz
	Transmit Power	-8 dBm ~ 8 dBm
	Receiver	-96dBm/ 1 Mbps
	Interface	UART/I²C/NFC
Dimension		12mm × 17 mm × 1.4mm Tolerance: ±0.2mm (without shielding cover) 12mm × 17 mm × 2.0mm Tolerance: ±0.2mm (with shielding cover)
Operating temperature		-40°C ~ +85°C
Storage temperature		-40°C ~ +85°C
Supply Voltage		1.8V~3.6V
Miscellaneous	Lead Free Warranty	Lead-free and RoHS compliant One Year

3 HARDWARE SPECIFICATION

3.1 Block Diagram and PIN Diagram

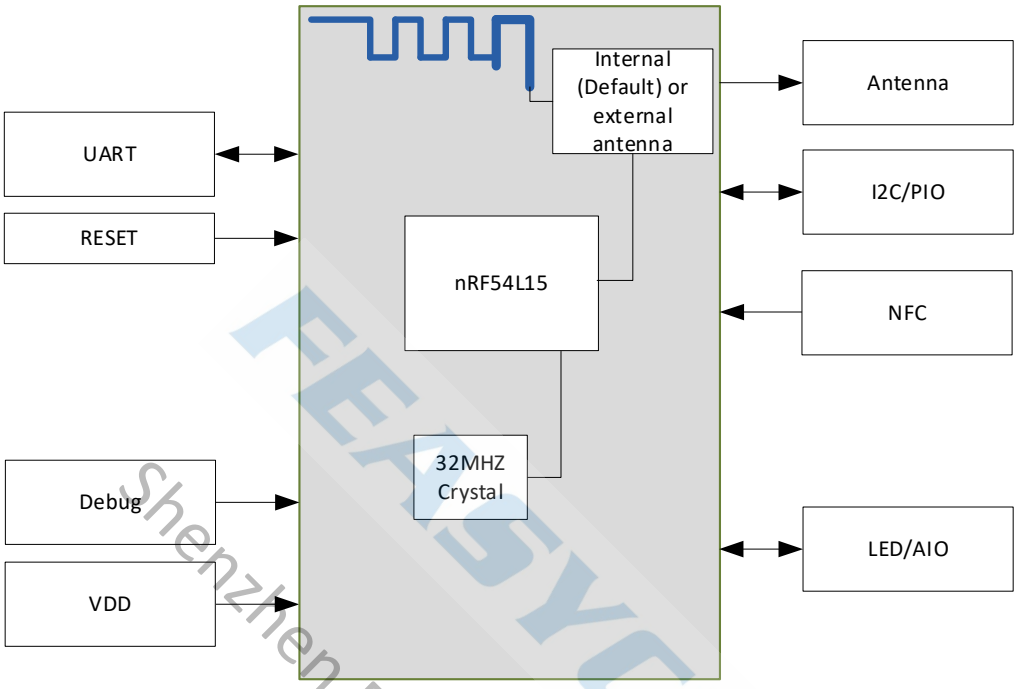

Figure 3-1:Block Diagram

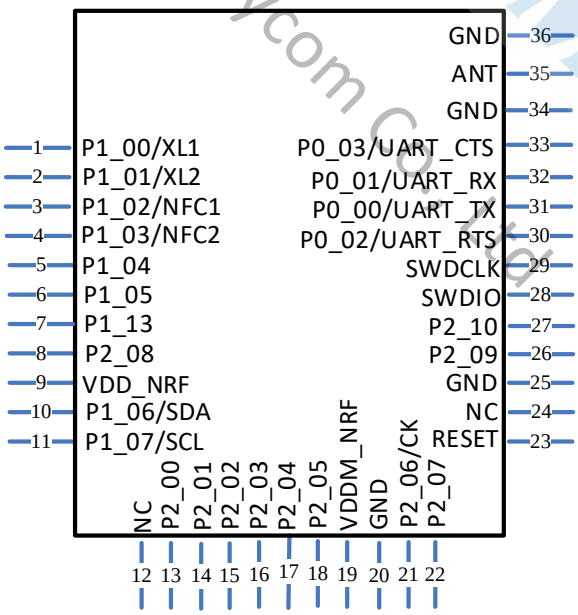

Figure 3-2:FSC-BT2024NI PIN Diagram(Top View)

3.2 Module Package Type

LCC package, as shown in the figure below.

Figure 3-2-1: FSC-BT2024NI Package Module Appearance

3.3 PIN Definition Descriptions

Table 3-2： Pin definition

Pin	Pin Name	Type	Pin Descriptions	Notes
1	P1_00/XL1	I/O	Programmable I/O Alternative function: 32.768KHZ XL1	
2	P1_01/XL2	I/O	Programmable I/O Alternative function: 32.768KHZ XL2	
3	P1_02/NFC1	I/O	Programmable I/O Alternative function: NFC1	
4	P1_03/NFC2	I/O	Programmable I/O Alternative function: NFC2	
5	P1_04	I/O	Programmable I/O	
6	P1_05	I/O	Programmable I/O	
7	P1_13	I/O	Programmable I/O	
8	P2_08	I/O	Programmable I/O	
9	VDD_NRF	I	External Input(1.8V-3.6V)	
10	P1_06/SDA	I/O	Programmable I/O Alternative function: I2C_SDA	
11	P1_07/SCL	I/O	Programmable I/O Alternative function: I2C_SCL	
12	NC			

13	P2_00	I/O	Programmable I/O
14	P2_01	I/O	Programmable I/O Alternative function: I2S_DOUT
15	P2_02	I/O	Programmable I/O Alternative function: I2S_BCLK
16	P2_03	I/O	Programmable I/O Alternative function: I2S_DIN
17	P2_04	I/O	Programmable I/O Alternative function: I2S_WS
18	P2_05	I/O	Programmable I/O
19	VDDM_NRF	VDD	1.8V-3.6V
20	GND	Vss	Power Ground
21	P2_06/CK	I/O	Programmable I/O
22	P2_07/SWO	I/O	Programmable I/O
23	RESET	I	RESET Active Low
24	NC		
25	GND	Vss	Power Ground
26	P2_09	I/O	Programmable I/O
27	P2_10	I/O	Programmable I/O-
28	SWDIO	I/O	Debug function: SWDIO
29	SWDCLK	I/O	Debug function: SWCLK
30	P0_02/UART_RTS	I/O	Programmable I/O Alternative function: UART_RTS
31	P0_00/UART_TX	I/O	Programmable I/O Alternative function: UART_TX
32	P0_01/UART_RX	I/O	Programmable I/O Alternative function: UART_RX
33	P0_03/UART_CTS	I/O	Programmable I/O Alternative function: UART_CTS
34	GND	Vss	Power Ground
35	ANT	RF	Bluetooth transmit/receive
36	GND	Vss	Power Ground

4 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 7-1: 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.

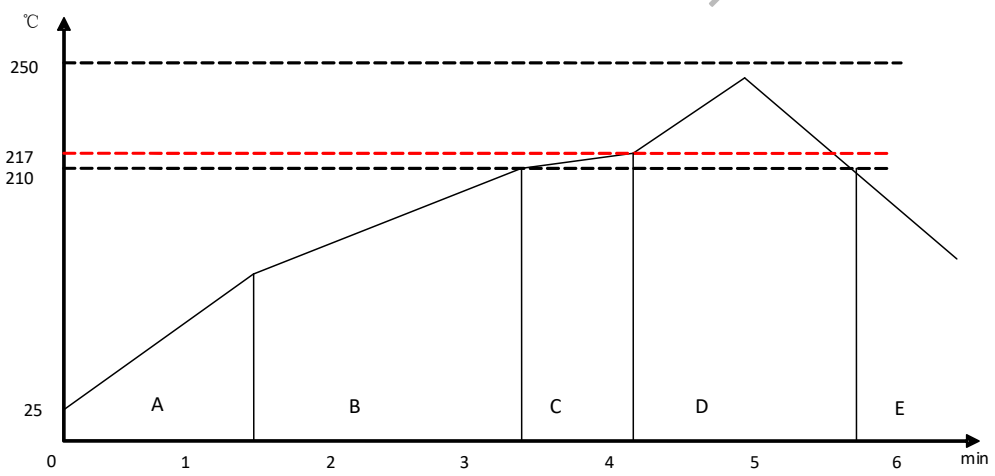

Figure 7-1: Typical Lead-free Re-flow

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.**

5 MECHANICAL DETAILS

5.1 Mechanical Details

- Dimension: 12mm(W) x 17mm(L) x 1.4mm(H) Tolerance: ±0.2mm (without shielding cover)
- 12mm(W) x 17mm(L) x 2.0mm(H) Tolerance: ±0.2mm (with shielding cover)
- Module size: 12mm X 17mm Tolerance: ±0.2mm
- Pad size: 1.7mmX0.5mm Tolerance: ±0.2mm
- Pad pitch: 0.9mm Tolerance: ±0.1mm

(Residual plate edge error: < 0.5mm)

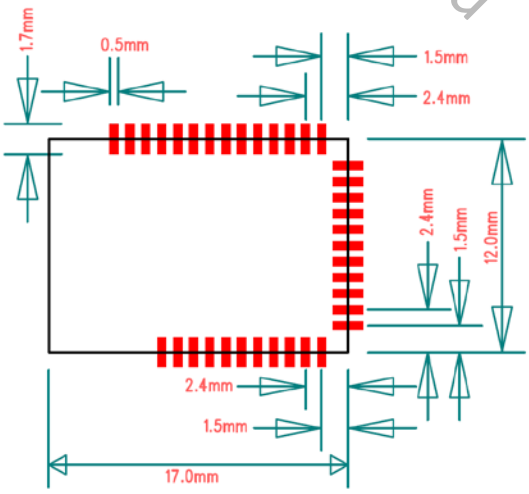

Figure 8-1： FSC-BT2024NI footprint Layout Guide (Top View)

6 HARDWARE INTEGRATION SUGGESTIONS

6.1 Soldering Recommendations

FSC-BT2024NI 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.

6.2 Layout Recommendations for Product Design Structure

The onboard antenna of this module is a specially designed antenna. Its optimal performance characteristics are highly dependent on the actual product's structure, materials, module placement, the shape of the baseboard, and even the thickness and dimensions of the baseboard. Therefore, the customer's baseboard design must strictly adhere to this guide to achieve the best RF performance and complete real-world distance testing and validation.

6.2.1 Module Layout Recommendations

Recommendation 1:

Place the module in the middle of the main board **(the customer's baseboard must be hollowed out)**. The TOP layer layout is shown in Figure 9-2-1:

- The upper edge of the module should align with the edge of the baseboard.
- The left edge of the module should be 11.1mm from the board edge, and the right edge of the module should be 12.1mm from the hollowed edge of the baseboard.
- No copper pour or traces are allowed on any layers beneath the IPEX connector.
- The hollowed-out area on the baseboard should measure 35.2x3.3mm.
- The clearance area on the baseboard should measure 21.3x2.0mm.

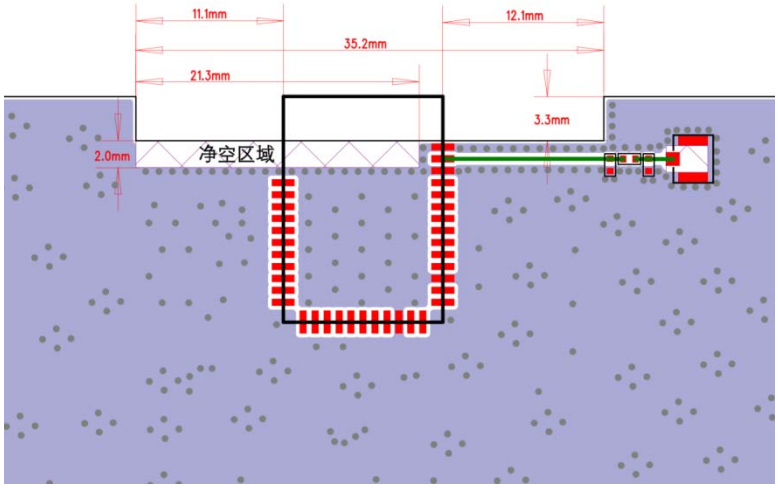

Figure 9-2-1：Module Layout - Baseboard TOP Layer

When the module is placed in the middle of the baseboard, the L2/L3/Ln...../Bottom layer layout is shown in Figure 9-2-2:

- The clearance area on L2/L3/Ln...../Bottom layers should measure 21.3x2.0mm.
- No copper pour or traces are allowed on any layers beneath the IPEX connector.

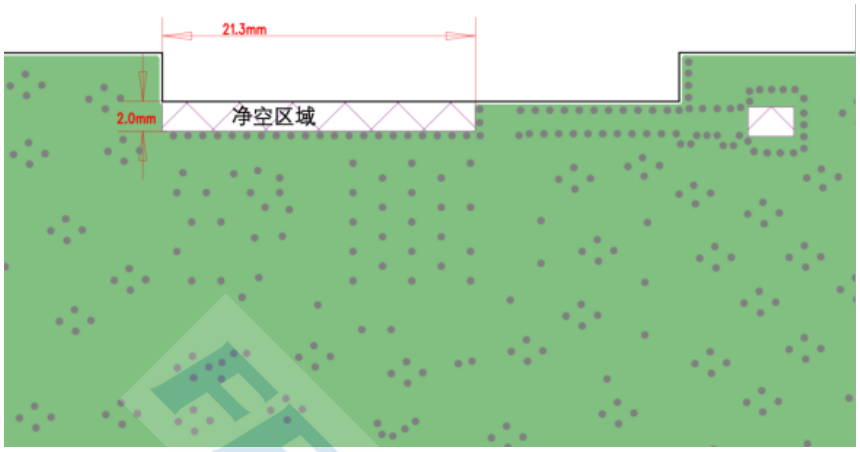

Figure 9-2-2： Module Layout - Baseboard L2/L3/BOTTOM Layers

Recommendation 2:

Similar to Recommendation 1, place the module at the edge of the baseboard **(the customer's baseboard must be hollowed out)**. The TOP layer layout is shown in Figure 9-2-3:

- The upper edge of the module should align with the edge of the baseboard.
- The left edge of the module should be 11.1mm from the board edge, and the right edge of the module should be 12.1mm from the hollowed edge of the baseboard.
- No copper pour or traces are allowed on any layers beneath the IPEX connector.
- The hollowed-out area on the baseboard should measure 35.2x3.3mm.
- The clearance area on the baseboard should measure 21.3x2.0mm.

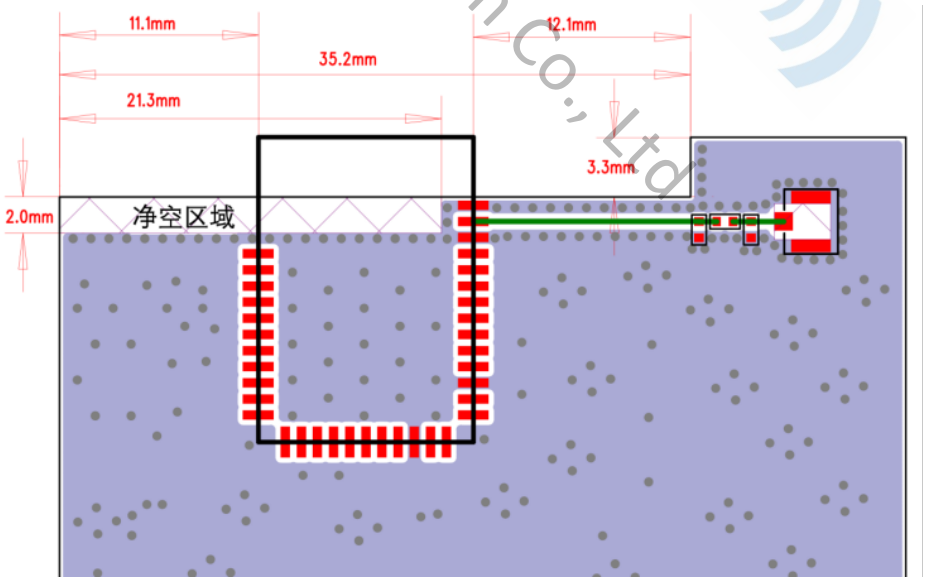

Figure 9-2-3： Module Layout - Baseboard TOP Layer

When the module is placed at the corner of the main board, the L2/L3/Ln...../Bottom layer layout is shown in Figure 9-2-4:

- The clearance area on L2/L3/Ln...../Bottom layers should measure 21.23x2.0mm.
- No copper pour or traces are allowed on any layers beneath the IPEX connector.

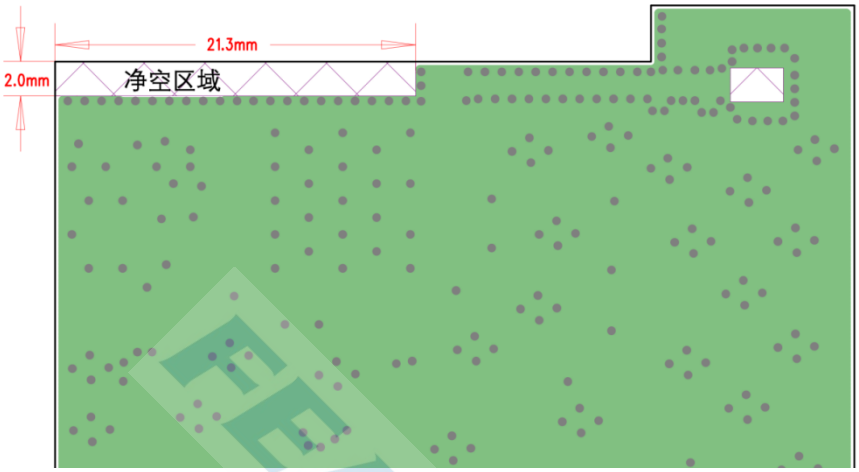

Figure 9-2-4： Module Layout - Main Board L2/L3/Ln...../BOTTOM Layers

6.2.2 Special Trace Recommendations

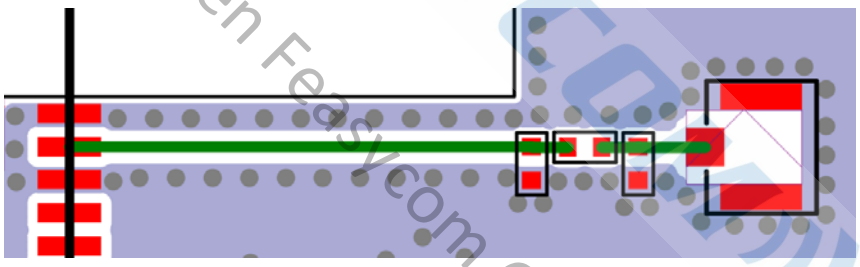

Figure 9-2-5： External Antenna Trace Schematic

The signal transmission line from the module to the antenna matching circuit should be a 50-ohm characteristic impedance microstrip line. The width of the microstrip line and the spacing from the ground copper must be determined based on the specific PCB layer stack-up. No intersecting lines are allowed between the microstrip line and the ground. All layers beneath the IPEX connector must be cleared (as shown by the purple cross-hatched area under the connector).

6.3 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.

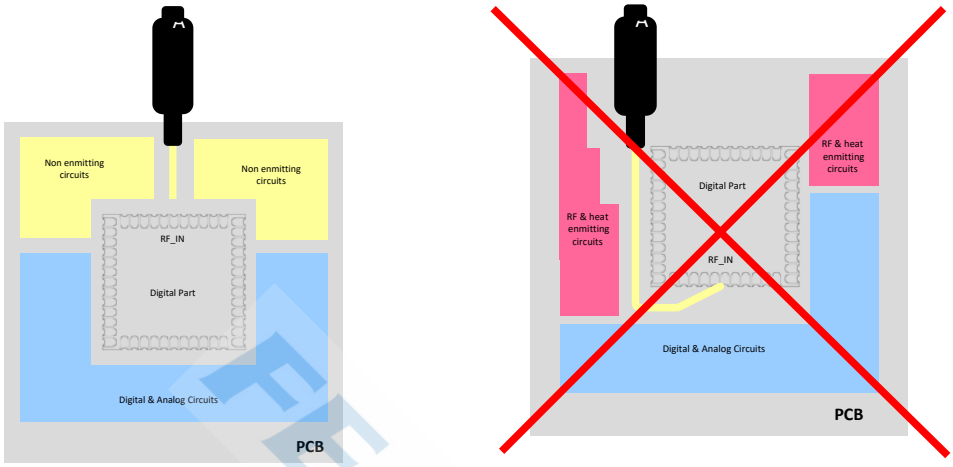

Figure 9-3: Placement the Module on a System Board

6.3.1 Antenna Connection and Grounding Plane Design

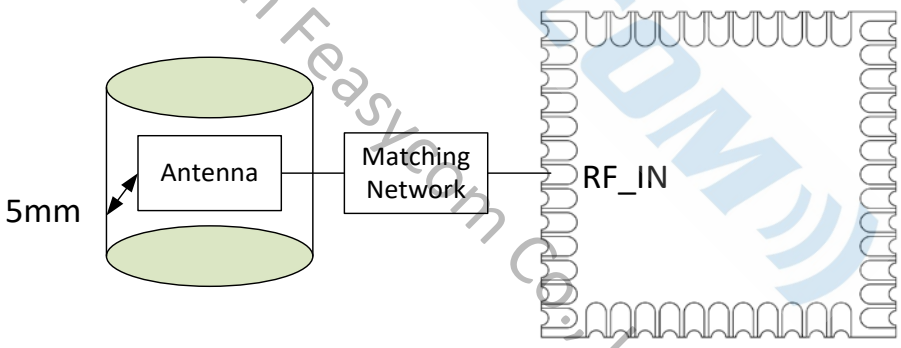

Figure 9-31-0: Leave 5mm Clearance Space from the Antenna

General design recommendations are:

- 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.

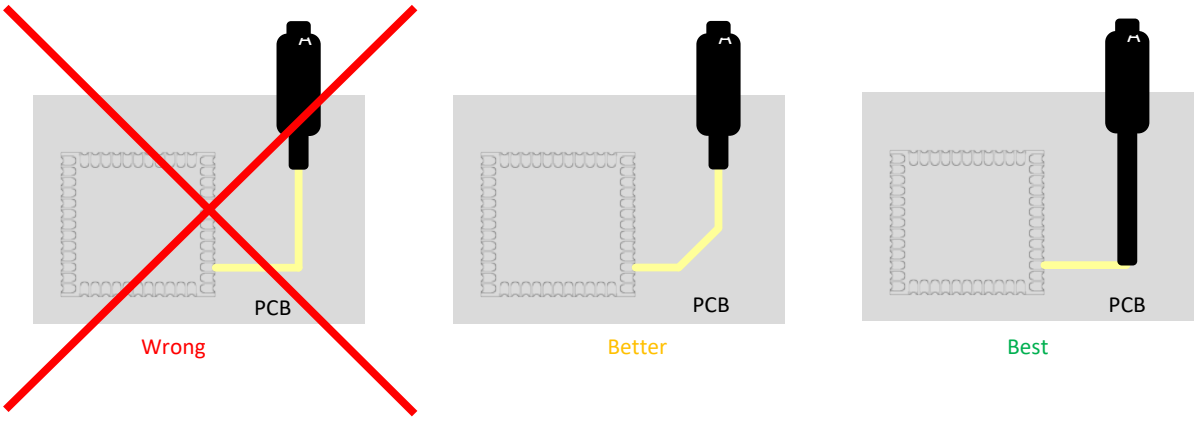

Figure 9-31-1: Recommended Trace Connects Antenna and the Module

- 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 via as possible to connect the ground planes.

7 PRODUCT PACKAGING INFORMATION

7.1 Default Packing

a, Tray vacuum

b, Tray Dimension: 230mm * 180mm* 8mm

Figure 10-1: Tray vacuum

7.2 Packing box(Optional)

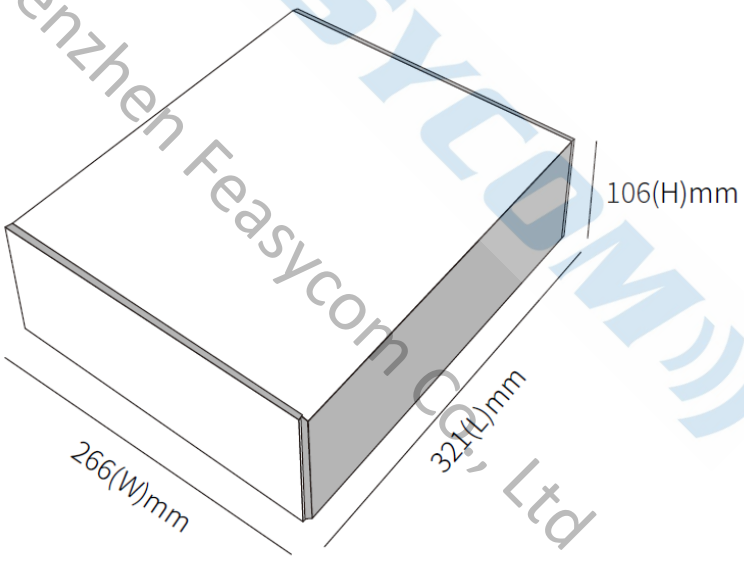

Figure 10-2: Packing box(Optional)

8 APPLICATION SCHEMATIC