

FSC-BT1106RC

DATASHEET V1.0

Revision History

Version	Data	Notes	Author
V1.0	2025-05-15	Initial Version	Lu

1 INTRODUCTION

Overview

FSC-BT1106RC supporting Classic Bluetooth and BLE

By default, FSC-BT1106RC 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-BT1106RC provides an ideal solution for developers who want to integrate Bluetooth wireless technology into their design.

Features

- Qualified to Bluetooth Dual Mode V5.3 BR/EDR/BLE specification
- Class-D amplifier, differential mode output
- High speed digital peripheral interface
- Low and Flexible latency
- Supports OTA firmware upgrade
- Realtek Real-M300, 32-bit ARMv8M mainline processor
- USB Audio
- PCB antenna

Application

- USB dongles
- Stereo headset
- TWS headset

2 General Specification

Table 2-1: General Specifications

Categories	Features	Implementation
Bluetooth		
	Bluetooth Standard	Bluetooth v5.3
	Frequency Band	2402MHz~2480MHz
	Transmit Power	UP to +10 dBm
	Receiver	-94dBm/ 1 Mbps
	Interface	UART/I ² S/USB
Size		13mm × 26.9 mm × 2.4mm
Operating temperature		0°C ~+70°C,
Storage temperature		0°C ~+70°C,
Supply Voltage		3.0-3.6V
Miscellaneous	Lead Free	Lead-free and RoHS compliant
	Warranty	One Year
MSL grade:		MSL3
ESD grade:		Human Body Model: Pass ±2000 V, Charge device model: Pass ±500 V,

3 HARDWARE SPECIFICATION

3.1 Block Diagram and PIN Diagram

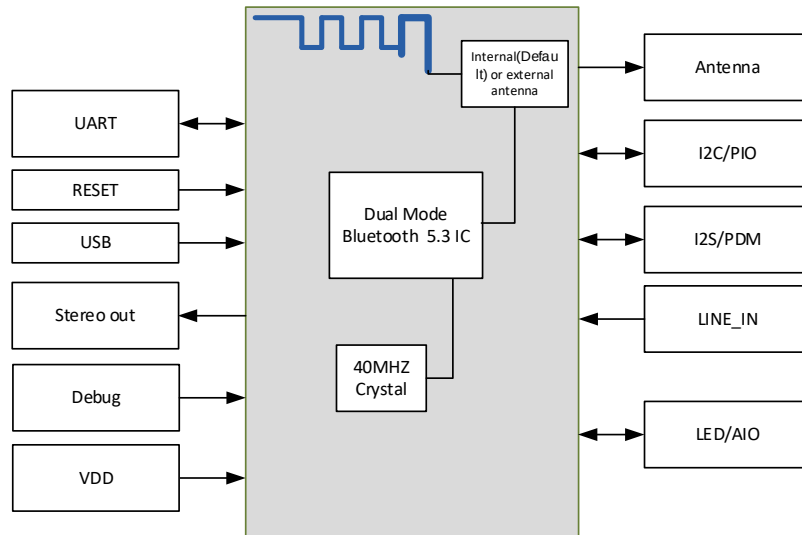


Figure 3-1:Block Diagram

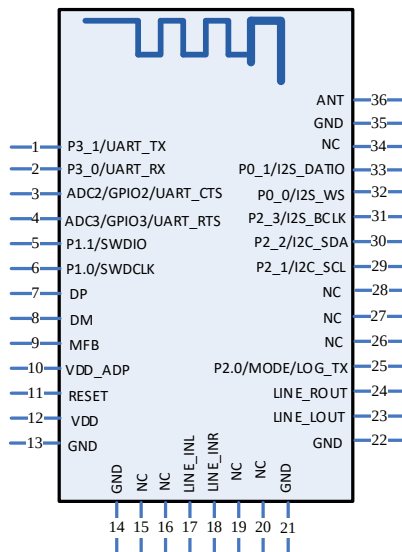


Figure 3-2:FSC-BT1106RC PIN Diagram(Top View)

3.2 PIN Definition Descriptions

Table 3-2: Pin definition

Pin	Pin Name	Type	Pin Descriptions	Notes
1	P3_1/UART_TX	I/O	Programmable I/O Alternative function:UART_TX	
2	P3_0/UART_RX	I/O	Programmable I/O Alternative function:UART_RX	
3	ADC2/GPIO2/UART_CTS	I/O	Programmable I/O	
4	ADC3/GPIO3/UART_RTS	I/O	Programmable I/O	
5	P1.1/SWDIO	I/O	Programmable I/O Alternative function: SWDIO	
6	P1.0/SWDCLK	I/O	Programmable I/O Alternative SWDCLK	
7	DP	I/O	USB Full Speed device D+	
8	DM	I/O	USB Full Speed device D-	
9	MFB	I	Button input Active Low	
10	VDD_ADP	I	USB Power(4.75~5.25V)	
11	RESET	I	RESET Active Low	
12	VDD	VDD	3V3	
13	GND	Vss	Power Ground	
14	GND	Vss	Power Ground	
15	NC			
16	NC			
17	LINE_INL	I	Line in left	
18	LINE_INR	I	Line in right	
19	NC			
20	NC			
21	GND	Vss	Power Ground	
22	GND	Vss	Power Ground	
23	LINE_LOUT	O	Line out left	
24	LINE_ROUT	O	Line out right	
25	P2.0/MODE/LOG_TX	IO	Programmable I/O	
26	NC			

27	NC		
28	NC		
29	P2_1/I2C_SCL	I/O	Programmable I/O Alternative function: I2C_SCL
30	P2_2/I2C_SDA	I/O	Programmable I/O Alternative function: I2C_SDA
31	P2_3/I2S_BCLK	I/O	Programmable I/O Alternative function: I2S_BCLK
32	P0_0/I2S_WS	I/O	Programmable I/O Alternative function: I2S_WS
33	P0_1/I2S_DATIO	I/O	Programmable I/O Alternative function: I2S_DATIO
34	NC		
35	GND	Vss	Power Ground
36	ANT	RF	Bluetooth transmit/receive.

4 MSL & ESD

Table 4-1: MSL and ESD

Parameter	Value
MSL grade:	MSL3
ESD grade	Electrostatic discharge
ESD – Human-body model (HBM) rating, JESD22-A114-F (Total samples from one wafer lot)	Pass ±2000 V
ESD – Charge-device model (CDM) rating, JESD22-C101-D (Total samples from one wafer lot)	Pass ±500 V

5 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 5-1: 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.

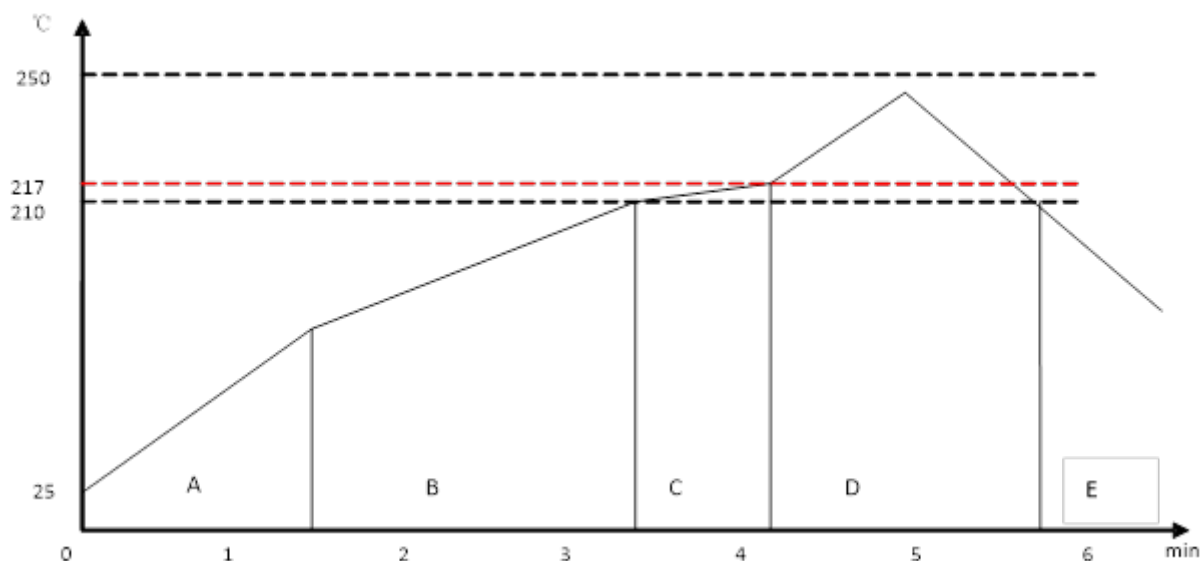


Figure 5-1: Typical Lead-free Re-flow

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**

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.

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

6.1 Mechanical Details

- Dimension: 13mm(W) x 26.9mm(L) x 2.4mm(H) Tolerance: $\pm 0.2\text{mm}$
- Module size: 13mm X 26.9mm Tolerance: $\pm 0.2\text{mm}$
- Pad size: 2.2mmX0.8mm Tolerance: $\pm 0.2\text{mm}$
- Pad pitch: 1.5mm Tolerance: $\pm 0.1\text{mm}$
- **(Residual plate edge error: $< 0.5\text{mm}$)**

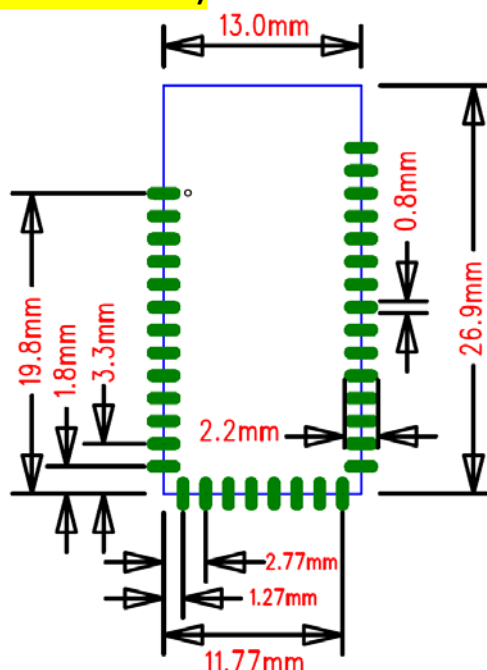


Figure 6-1: FSC-BT1106RC footprint Layout Guide (Top View)

7 HARDWARE INTEGRATION SUGGESTIONS

7.1 Soldering Recommendations

The FSC-BT1106RC is compatible with the industrial standard reflow profile for Pb-free soldering. The specific reflow profile depends on factors such as the thermal mass of the fully populated PCB, the oven's heat transfer efficiency, and the type of solder paste used. Please refer to the datasheet of the specific solder paste for precise profile configurations.

Feasycom provides the following recommendations to ensure reliable solder joints and optimal module performance after soldering. However, since the reflow profile is influenced by the process and layout, it is advisable to optimize the profile based on individual cases. The recommendations below should serve as an initial guide.

7.2 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.

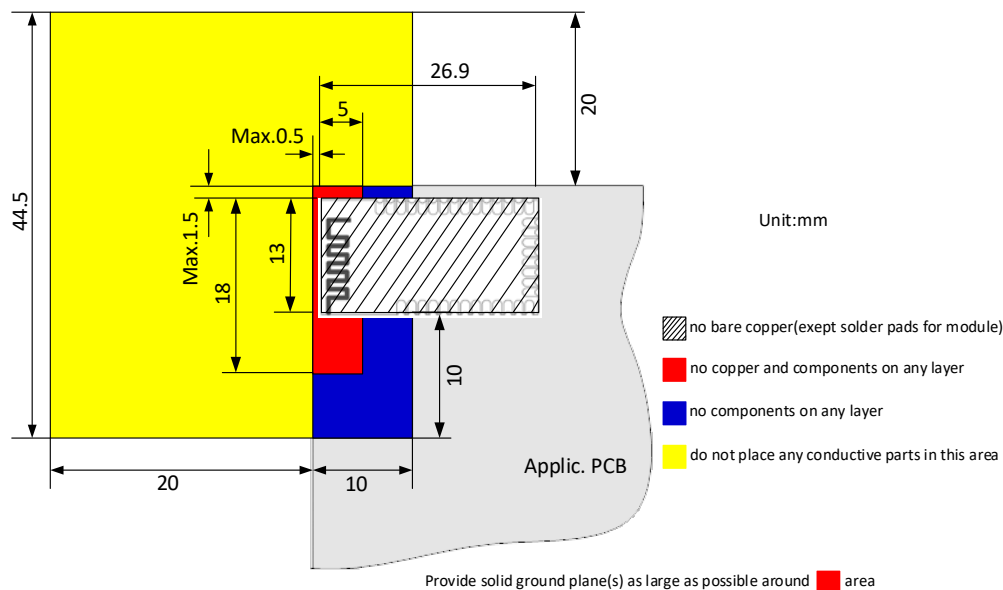


Figure 7-2: Restricted Area (Design schematic, for reference only. Unit: mm)

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).

7.3 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.

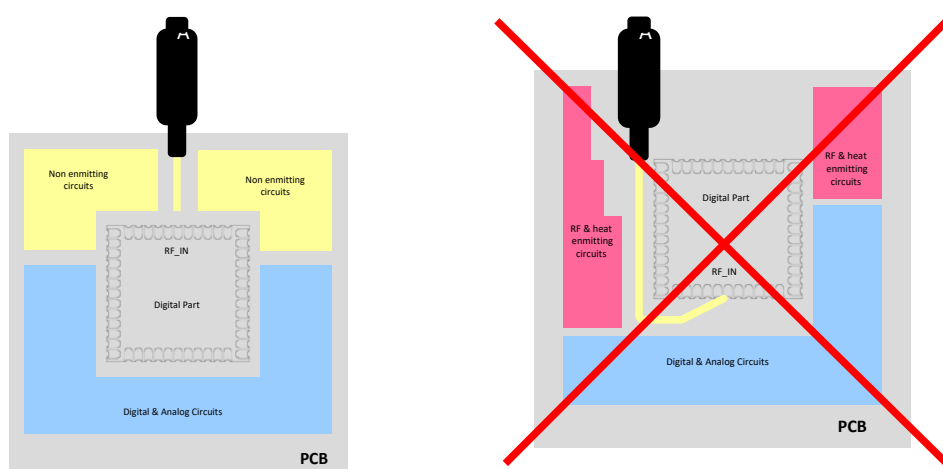


Figure 7-3: Placement the Module on a System Board

7.3.1 Antenna Connection and Grounding Plane Design

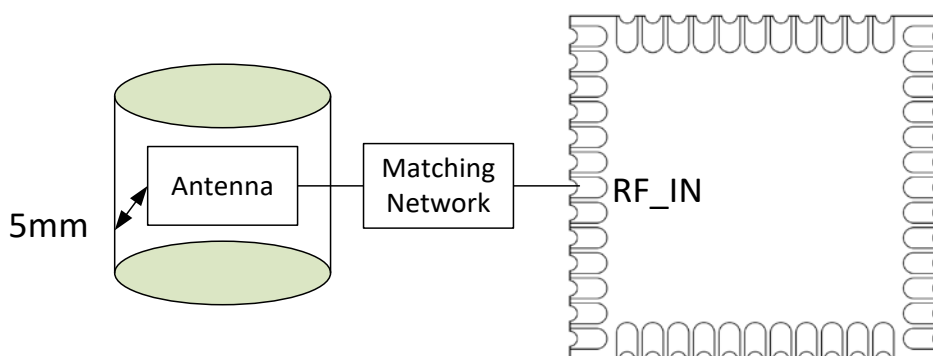


Figure 7-3-1: Leave 5mm Clearance Space from the Antenna

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.

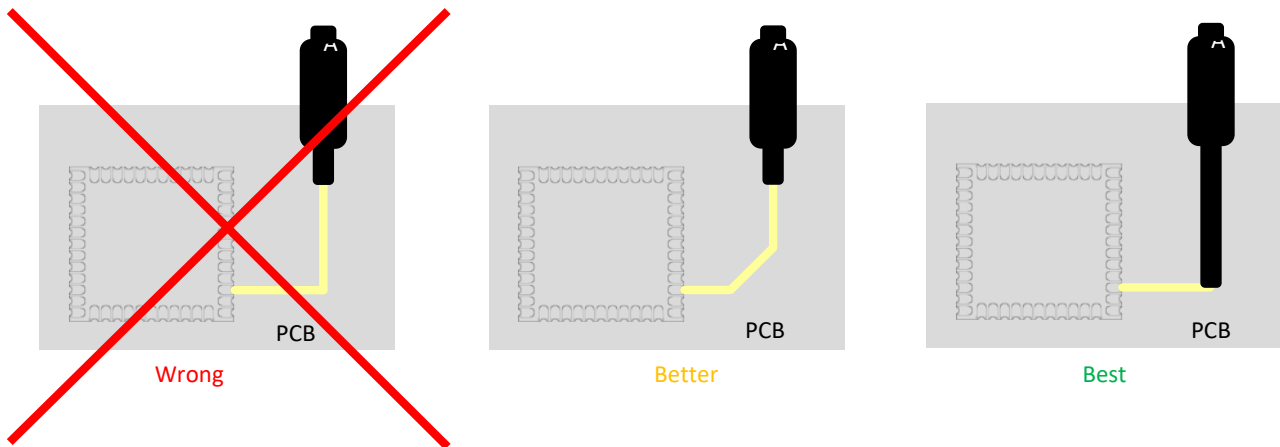


Figure 7-3-2: Recommended Trace Connects Antenna and the Module

- 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 via as possible to connect the ground planes.

8 PRODUCT PACKAGING INFORMATION

8.1 Default Packing



Figure 8-1: Tray Dimension: 140mm * 265mm Tray vacuum

8.2 Packing box (Optional)

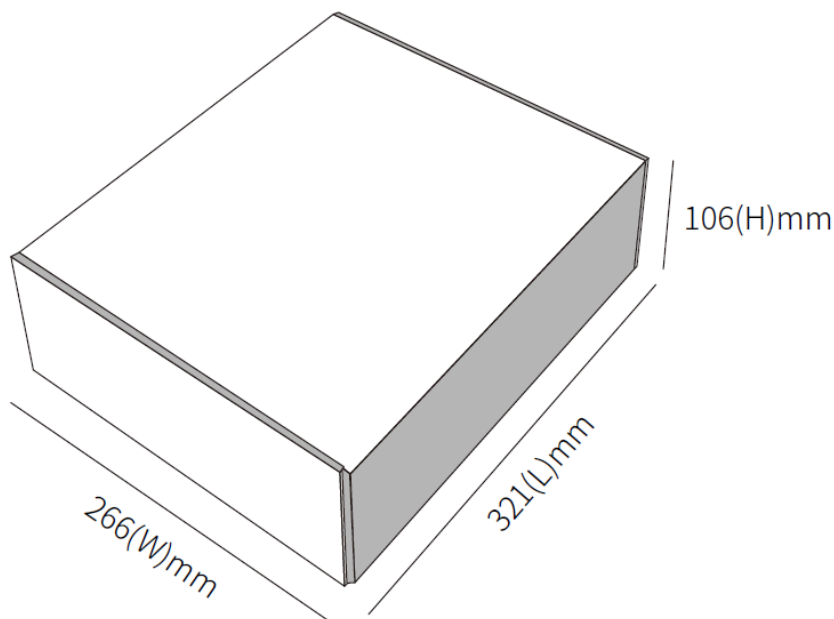


Figure 8-2: Packing box(Optional)

9 FCC NOTE

15.19 (a)(3)

Un-license band: This device complies with Part 15 of the FCC Rules.

Operation is subject to the following two conditions: (1) this device may not cause harmful interference, and (2) this device must accept any interference received, including interference that may cause undesired operation.

For All device

(15.21) Warning: Changes or modifications to this unit not expressly approved by the party responsible for compliance could void the user authority to operate the equipment.

For Class B digital device or peripheral or cordless phone 15.105

This equipment has been tested and found to comply with the limits for a Class B digital device, pursuant to Part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference in a residential installation. This equipment generates, uses and can radiate radio frequency energy and, if not installed and used in accordance with the instructions, may cause harmful interference to radio communications.

However, there is no guarantee that interference will not occur in a particular installation. If this

equipment does cause harmful interference to radio or television reception, which can be determined by turning the equipment off and on, the user is encouraged to try to correct the interference by one or more of the following measures:

- ◆ Reorient or relocate the receiving antenna.
- ◆ Increase the separation between the equipment and receiver.
- ◆ Connect the equipment into an outlet on a circuit different from that to which the receiver is needed.
- ◆ Consult the dealer or an experienced radio/TV technician for help.

Warning: Changes or modifications to this unit not expressly approved by the party responsible for compliance could void the user authority to operate the equipment.

1. To comply with FCC RF exposure compliance requirements, a separation distance of at least 20 cm must be maintained between the antenna of this device and all persons.
2. This Transmitter must not be co-located or operating in conjunction with any other antenna or transmitter

This device is intended only for OEM integrators under the following conditions:

- 1) The antenna must be installed such that 20 cm is maintained between the antenna and users. For laptop installations, the antenna must be installed to ensure that the proper spacing is maintained in the event the users places the device in their lap during use (i.e. positioning of antennas must be placed in the upper portion of the LCD panel only to ensure 20 cm will be maintained if the user places the device in their lap for use) and
- 2) The transmitter module may not be co-located with any other transmitter or antenna. As long as the 2 conditions above are met, further transmitter testing will not be required. However, the OEM integrator is still responsible for testing their end-product for any additional compliance requirements required with this module installed (for example, digital device emissions, PC peripheral requirements, etc.).

IMPORTANT NOTE: In the event that these conditions can not be met (for example certain laptop configurations or co-location with another transmitter), then the FCC authorization is no longer considered valid and the FCC ID can not be used on the final product. In these circumstances, the OEM integrator will be responsible for re-evaluating the end product (including the transmitter) and obtaining a separate FCC authorization. End Product Labeling This transmitter module is authorized only for use in devices where the antenna may be installed such that 20 cm may be maintained between the antenna and users (for example access points, routers, wireless ASDL modems, certain laptop configurations, and similar equipment).

The final end product must be labeled in a visible area with the following: "Contains TX FCC ID:

TN7FSCBT1106RC".

This device complies with Part 15 of the FCC Rules. Operation is subject to the following two conditions: (1) this device may not cause harmful interference, and (2) this device must accept any interference received, including interference that may cause undesired operation.

Le present appareil est conforme aux CNR d'ISED applicables aux appareils radio exempts de licence. L'exploitation est autorisee aux deux conditions suivantes : (1) le dispositif ne doit pas produire de brouillage prejudiciable, et (2) ce dispositif doit accepter tout brouillage recu, y compris un brouillage susceptible de provoquer un fonctionnement indésirable