MINEWSEMI

Bluetooth LE Module

ME54BS13

Datasheet

V 1.0.0

Version Note

Version	Details	Contributor(s)	Date	Notes
1.0.0	First edit	Michelle	2026.06.23	

Part Number

Model	Hardware Code
ME54BS13	-

Click the icon to view and download the latest product documents electronically.
https://en.minewsemi.com/file/ME54BS13-nRF54LM20A_Datasheet_K_EN.pdf

ME54BS13-nRF54LM20A

Highly Flexible, Cost-Effective Ultra-Low Power Bluetooth Module

The ME54BS13 is a compact, ultra-low-power Bluetooth® Low Energy module based on Nordic Semiconductor's nRF54LM20A SoC. Equipped with a 128 MHz Arm® Cortex®-M33 processor, 2036 KB of NVM, and 512 KB of RAM, it provides a powerful and efficient platform for next-generation wireless applications.

The module integrates an onboard PCB antenna and offers up to 64 GPIOs, enabling flexible hardware design and easy product integration. With excellent RF performance, low system power consumption, and support for Bluetooth LE, Zigbee, Thread, Matter, and Aliro development, the ME54BS13 is an ideal connectivity solution for smart home, healthcare, industrial IoT, wearable, and consumer electronic devices.

FEATURES

- Bluetooth 6.0
- High cost performance
- Ultra-low Power
- Supports Development of Zigbee, Thread, Matter, Aliro and Other Protocols

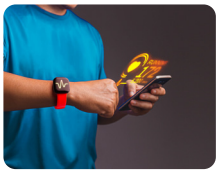
KEY PARAMETERS

ME54BS13-nRF54LM20A			
Chip Model	nRF54LM20A	**Antenna**	PCB
Module Size	16.5×12.0×2.4mm	**GPIO**	64
Flash	2036KB	**RAM**	512 KB
Receiving Sensitivity	-95.5dBm	**Transmission Power**	-10 ~ +8dBm
Current(TX)	0dBm-5mA	**Current(RX)**	3.8mA

APPLICATIONS

Smart Home

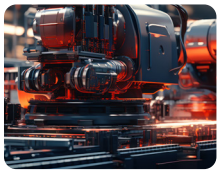
Virtual Reality and Augmented Reality



Advanced Wearables

Electric Vehicles

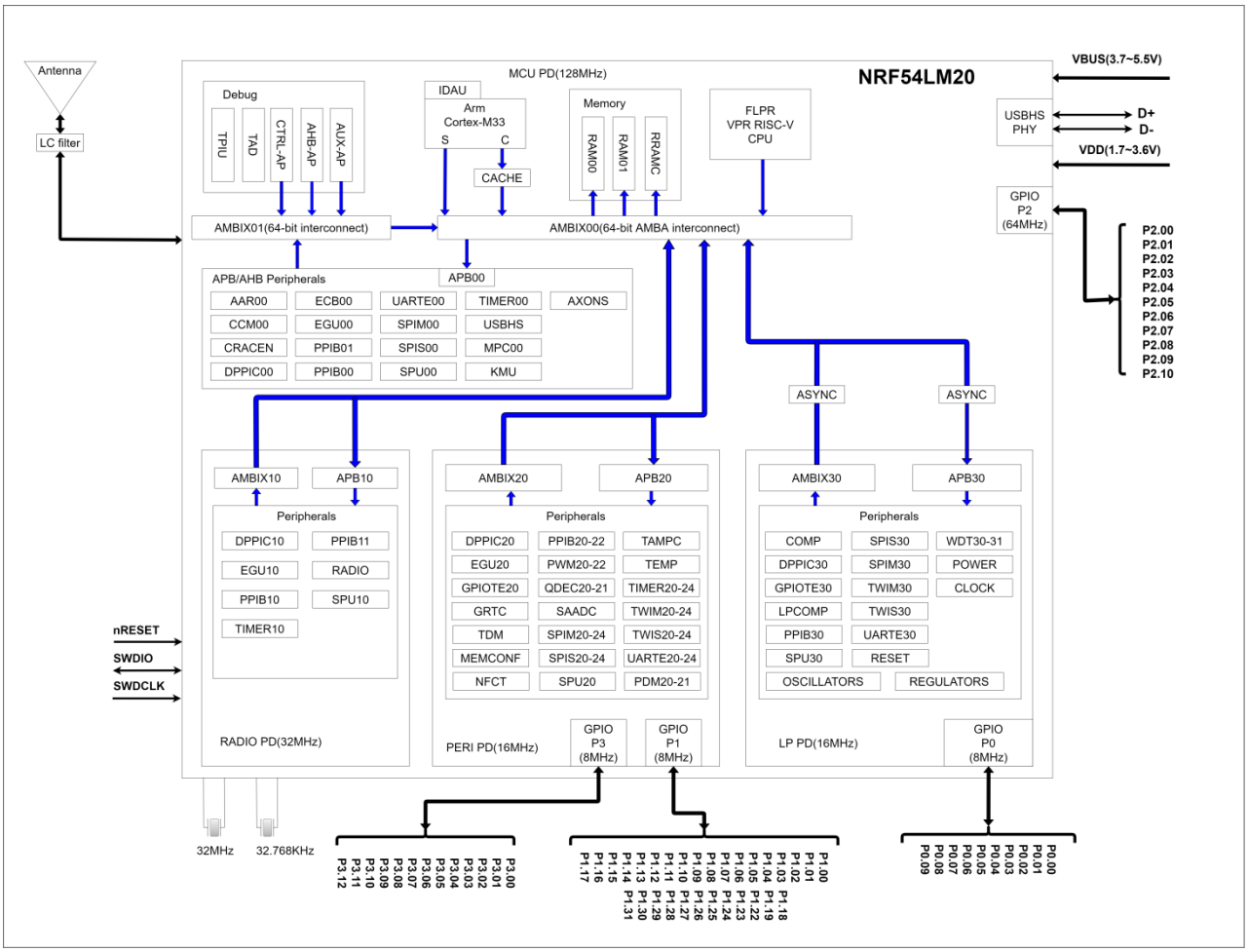
Medical Devices



Industrial IoT

INDEX

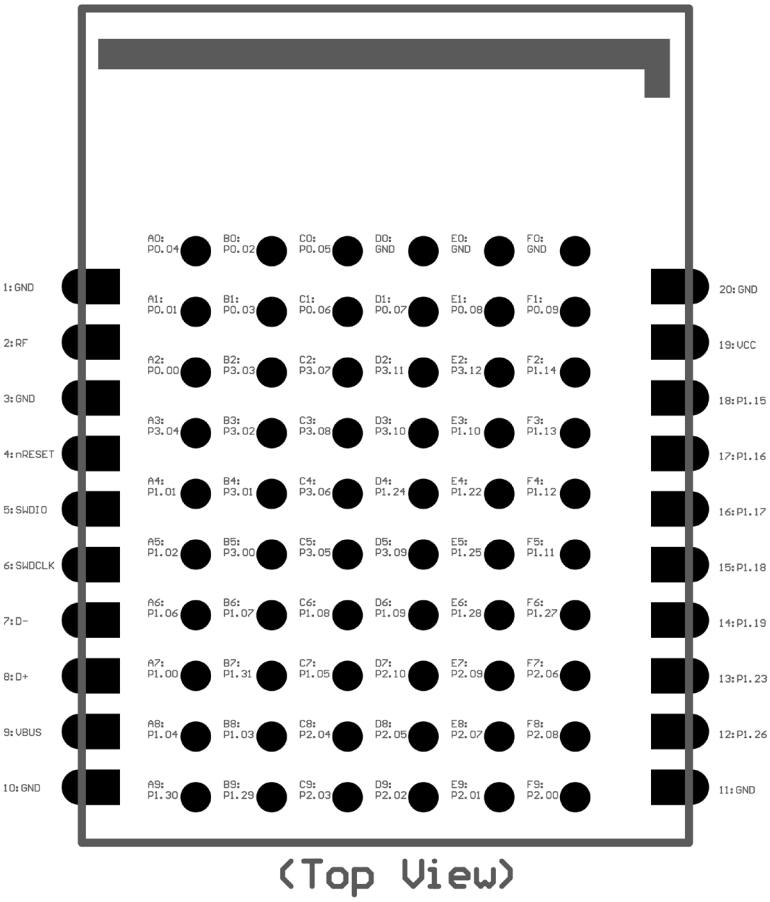
1 BLOCK DIAGRAM



2 ELECTRICAL SPECIFICATION

Parameter	Values	Notes
Working Voltage	1.7V-3.6V	To ensure RF work, it is recommended that the supply voltage should not be lower than 2.3V.
Working Temperature	-40℃~+85℃	Storage temperature: -40°C to +105°C
Transmission Power	-10 ~ +8dBm	Configurable
Current(RX)	3.8mA	RF receiving current at 1 Mbps mode
Current(TX)	5mA	RF transmission current at 0 dBm output power
Module Dimensions	16.5*12.0*2.4mm	
Quantity of IO Port	64	

3 PIN DESCRIPTION



4 PIN DEFINITION

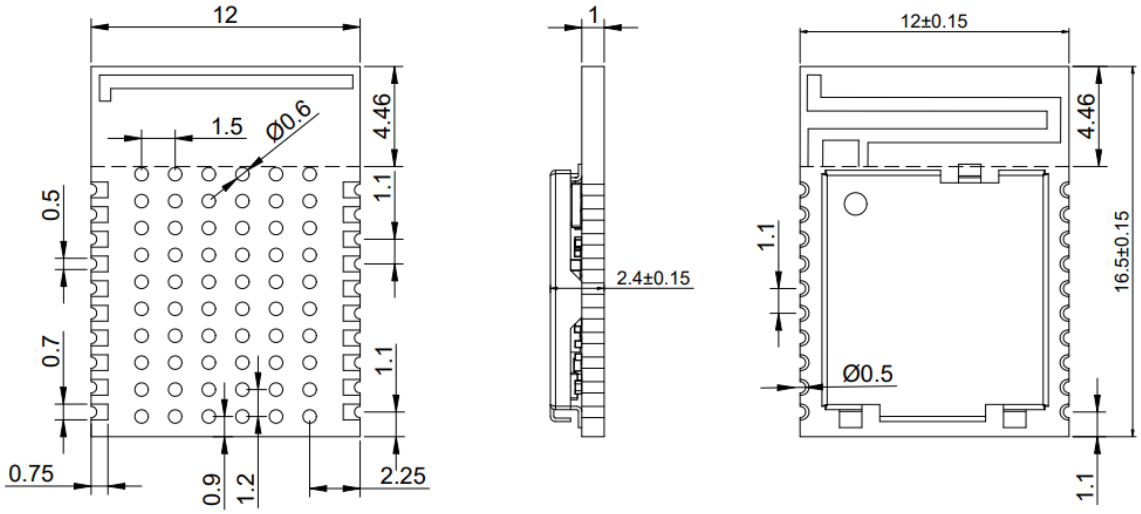
Pin Number	Available pins	Pad type	Definition
1	GND	GND	
2	RF	External antenna pin	By default, the onboard antenna is used and this pin is left unconnected. To use an external antenna, reconfigure the RF matching resistor and connect the antenna through this pin. Please contact our sales team before ordering to confirm the required antenna configuration.
3	GND	GND	
4	nRESET	Reset	Low level reset, high level work
5	SWDIO	Programming data pin	For flashing firmware
6	SWCLK	Programming clock pin	For flashing firmware

Pin Number	Symbol	Type	Definition
7	D-	USB	USB D-
8	D+	USB	USB D+
9	VBUS	Power supply	5V USB power supply
10/11	GND	GND	
12	P1.26	GPIO	General Purpose I/O
13	P1.23	GPIO	General Purpose I/O
14	P1.19	GPIO	General Purpose I/O
15	P1.18	GPIO	General Purpose I/O
16	P1.17	GPIO	General Purpose I/O
17	P1.16	GPIO	General Purpose I/O
18	P1.15	GPIO	General Purpose I/O
19	VCC	VCC	
20	GND	GND	
A0	P0.04	GPIO	General Purpose I/O
A1	P0.01	GPIO	General Purpose I/O
A2	P0.00	GPIO	General Purpose I/O
A3	P3.04	GPIO	General Purpose I/O
A4	P1.01	GPIO/NFC1	General Purpose I/O/NFC input
A5	P1.02	GPIO/NFC2	General Purpose I/O/NFC input
A6	P1.06	GPIO/AIN4	General Purpose I/O/Analog input
A7	P1.00	GPIO/AIN0	General Purpose I/O/Analog input
A8	P1.04	GPIO/AIN6	General Purpose I/O/Analog input
A9	P1.30	GPIO/AIN2	General Purpose I/O/Analog input
B0	P0.02	GPIO	General Purpose I/O
B1	P0.03	GPIO	General Purpose I/O
B2	P3.03	GPIO	General Purpose I/O
B3	P3.02	GPIO	General Purpose I/O
B4	P3.01	GPIO	General Purpose I/O

Pin Number	Symbol	Type	Definition
B5	P3.00	GPIO	General Purpose I/O
B6	P1.07	GPIO	General Purpose I/O
B7	P1.31	GPIO/AIN1	General Purpose I/O/Analog input
B8	P1.03	GPIO/AIN7	General Purpose I/O/Analog input
B9	P1.29	GPIO/AIN3	General Purpose I/O/Analog input
C0	P0.05	GPIO	General Purpose I/O
C1	P0.06	GPIO	General Purpose I/O
C2	P3.07	GPIO	General Purpose I/O
C3	P3.08	GPIO	General Purpose I/O
C4	P3.06	GPIO	General Purpose I/O
C5	P3.05	GPIO	General Purpose I/O
C6	P1.08	GPIO	General Purpose I/O
C7	P1.05	GPIO/AIN5	General Purpose I/O/Analog input
C8	P2.04	GPIO	General Purpose I/O
C9	P2.03	GPIO	General Purpose I/O
D0	GND	GND	
D1	P0.07	GPIO	General Purpose I/O
D2	P3.11	GPIO	General Purpose I/O
D3	P3.10	GPIO	General Purpose I/O
D4	P1.24	GPIO	General Purpose I/O
D5	P3.09	GPIO	General Purpose I/O
D6	P1.09	GPIO	General Purpose I/O
D7	P2.10	GPIO	General Purpose I/O
D8	P2.05	GPIO	General Purpose I/O
D9	P2.02	GPIO	General Purpose I/O
E0	GND	GND	
E1	P0.08	GPIO	General Purpose I/O
E2	P3.12	GPIO	General Purpose I/O

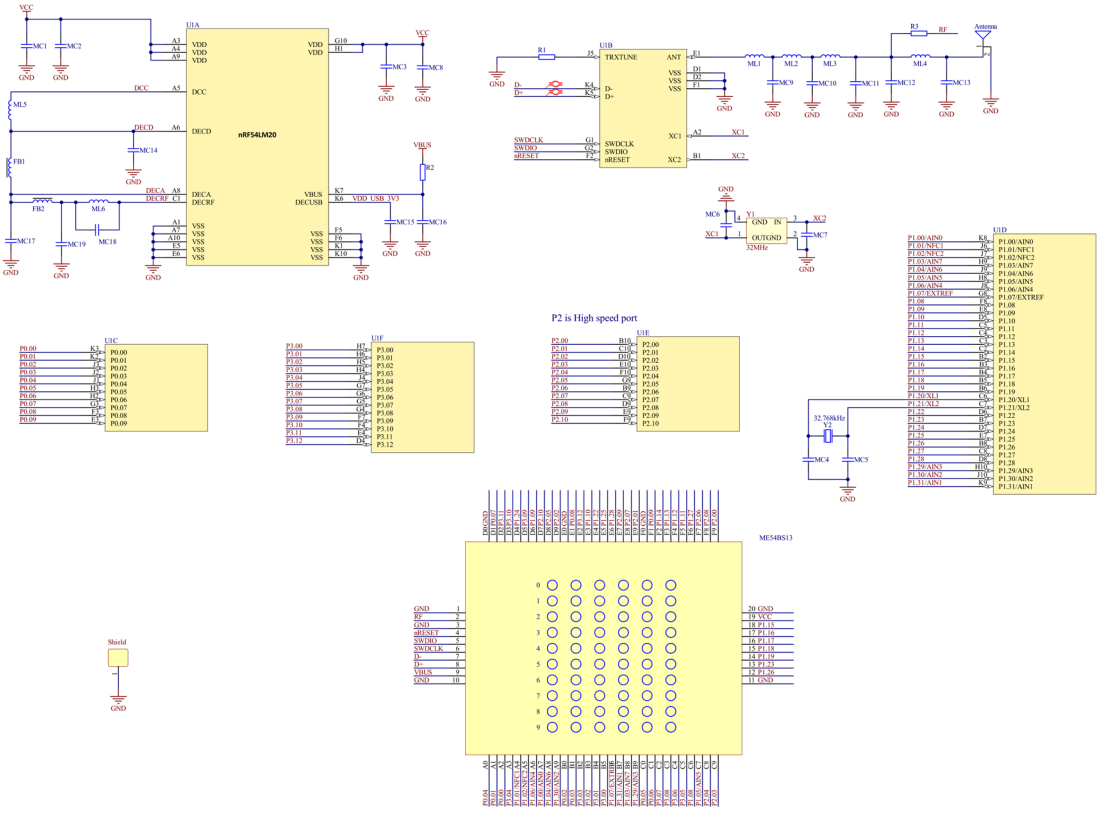
Pin Number	Symbol	Type	Definition
E3	P1.10	GPIO	General Purpose I/O
E4	P1.22	GPIO	General Purpose I/O
E5	P1.25	GPIO	General Purpose I/O
E6	P1.28	GPIO	General Purpose I/O
E7	P2.09	GPIO	General Purpose I/O
E8	P2.07	GPIO	General Purpose I/O
E9	P2.01	GPIO	General Purpose I/O
F0	GND	GND	
F1	P0.09	GPIO	General Purpose I/O
F2	P1.14	GPIO	General Purpose I/O
F3	P1.13	GPIO	General Purpose I/O
F4	P1.12	GPIO	General Purpose I/O
F5	P1.11	GPIO	General Purpose I/O
F6	P1.27	GPIO	General Purpose I/O
F7	P2.06	GPIO	General Purpose I/O
F8	P2.08	GPIO	General Purpose I/O
F9	P2.00	GPIO	General Purpose I/O

5 MECHANICAL DRAWING



Default unit: mm Default tolerance: ±0.15

6 ELECTRICAL SCHEMATIC



Notice: Before placing an order, please confirm the specific configuration required with the salesperson.

7 PCB DESIGN

7.1 Disclaimer

This document is provided as a recommended design reference for wireless modules and does not constitute any explicit or implied guarantee of performance or compliance. Due to differences in PCB layout, enclosure structure, antenna selection, and application environments of the end product, this document cannot guarantee RF performance in all scenarios. Users must conduct sufficient RF performance and compliance verification after completing the final product design and shall bear full responsibility for design and application outcomes. Our company reserves the right to modify the contents of this document without prior notice.

7.2 Core Rules for Module Placement

Board Edge Priority Rule

The RF output/antenna side of the module must face the edge of the main PCB to ensure that antenna radiation is directed outward from the board without obstruction. Placing the module in a closed central area of the PCB is strictly prohibited.

Keep-Out Area Rule

A 0.5 mm area around the module body must be kept clear. Non-ground components and non-ground signal traces are prohibited within this area. Within 4 mm around the RF side of the module, copper pours, unrelated components, and non-RF signal traces are strictly prohibited.

Interference Isolation Rule

The module must maintain a minimum safe distance from strong interference sources. Adjacent placement on the same layer is prohibited. Isolation using different PCB layers and shielding covers is recommended. General isolation requirements are as follows:

Interference Source Type	Minimum Recommended Isolation Distance
DC-DC switching power supplies, power inductors, power transformers	20mm
USB 3.0 / HDMI 2.0 / DDR / High-speed SDIO interfaces	20mm
MCU high-frequency clock circuits, Ethernet PHY chips	15mm
Displays, cameras, FPC cables with wiring	25mm

Structural Clearance Rule

The minimum distance between the module and metal screws, clips, standoffs, and other structural parts must be ≥5 mm. At least 4 mm of SMT soldering and rework clearance must be reserved around the module. The RF side of the module must not be enclosed inside a fully metal cavity.

Multiple Modules on the Same PCB

When placing multiple wireless modules of the same or different types on the same PCB, the minimum spacing between modules must be ≥50 mm to avoid mutual interference.

7.3 GENERAL PCB DESIGN REQUIREMENTS

Grounding Design

The main PCB must maintain a complete and continuous ground plane. Splitting the ground plane within the module projection area or RF surrounding area is strictly prohibited. All module ground pins must be soldered. At least one grounding via should be placed near each ground pad and connected to the main ground plane. A complete ground plane with dense grounding vias may be placed under the non-RF area of the module to improve shielding performance.

Power Supply Design

Decoupling capacitor combinations must be placed close to the module power input pins. The trace length between capacitor pads and power pins should be ≤0.5 mm, and at least two grounding vias should be placed near the ground terminal. For switching power supply applications, a π-type filter circuit footprint must be reserved near the module power input pins.

RF Trace Design

The entire RF path must maintain a strict 50 Ω characteristic impedance (tolerance ±10%). RF traces should be as short as possible and routed preferably in straight lines. Sharp or right-angle corners are prohibited; curved routing with a radius ≥3 times the trace width is recommended. Branch traces and stub traces are prohibited. A continuous reference ground plane must exist beneath the RF traces, and no other signal traces should be routed within 3 times the trace width on either side. Ground shielding should be applied throughout the RF routing.

Control Interface Design

Control signal traces such as UART, SPI, and I2C should be routed away from RF areas with short and simple routing paths and continuous ground shielding. Parallel routing adjacent to RF traces is prohibited. ESD protection devices must be placed close to external PCB connectors and must not be placed near module pins.

Footprint Design

Strictly use the standard PCB footprint provided by our company. Unauthorized modification of pad size or spacing is prohibited to ensure soldering reliability.

7.4 DESIGN RESTRICTIONS

Do not place the RF side of the module facing the inside of the PCB or away from the board edge.

Do not place copper pours, components, or fully enclosed metal housings without openings over the antenna area.

Do not route RF traces across split ground planes, without a complete reference ground, or with right-angle corners.

Do not place modules adjacent to strong interference sources or split the ground plane in the module area.

Do not leave module ground pins unconnected or inadequately grounded.

7.5 MODULE PLACEMENT EXAMPLE

For end products, antenna position significantly affects overall system performance. The module should preferably be placed at the edge or corner of the PCB to maximize the clearance area and reduce interference from surrounding components. The PCB area directly beneath the module antenna should be hollowed out to keep the antenna region suspended, while no signal traces, metal objects, or other interference sources should exist within 3–5 mm around the antenna area. The figure below demonstrates an example of proper module placement on the main PCB for optimal RF performance. The first recommended layout method should be used whenever possible, as the module antenna design has been tuned and optimized based on this configuration.

Correct Example

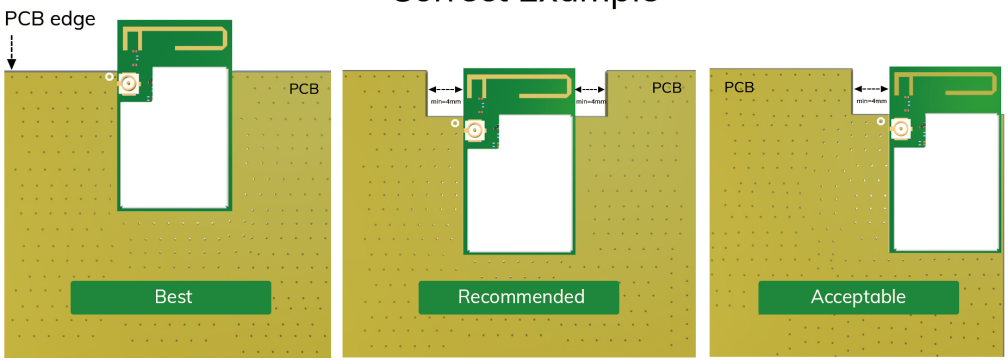


Figure 1 Figure 2 Figure 3

Incorrect Example

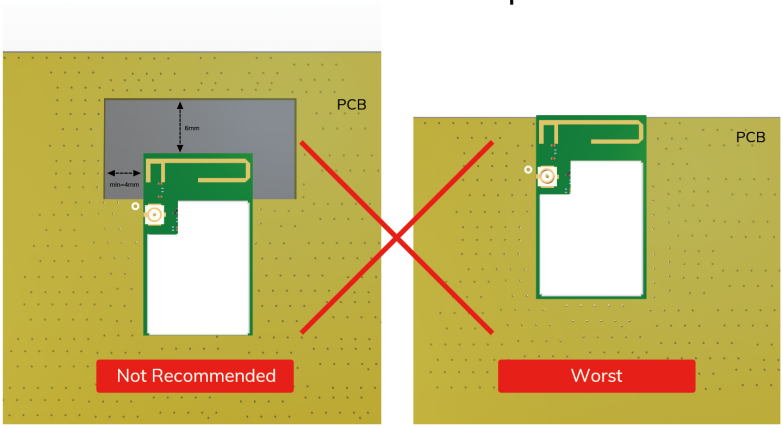
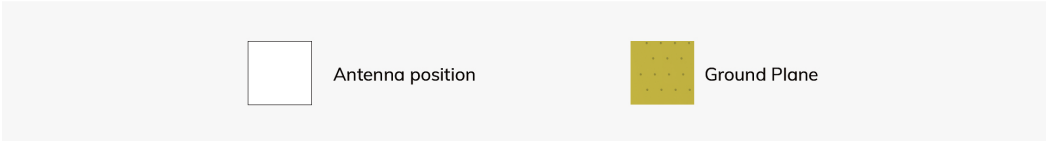


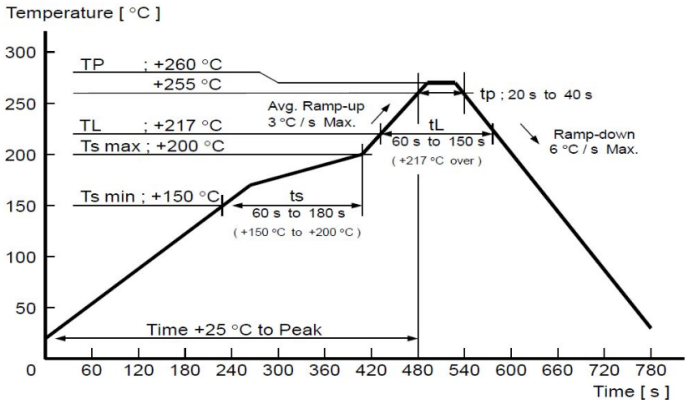
Figure 4 Figure 5



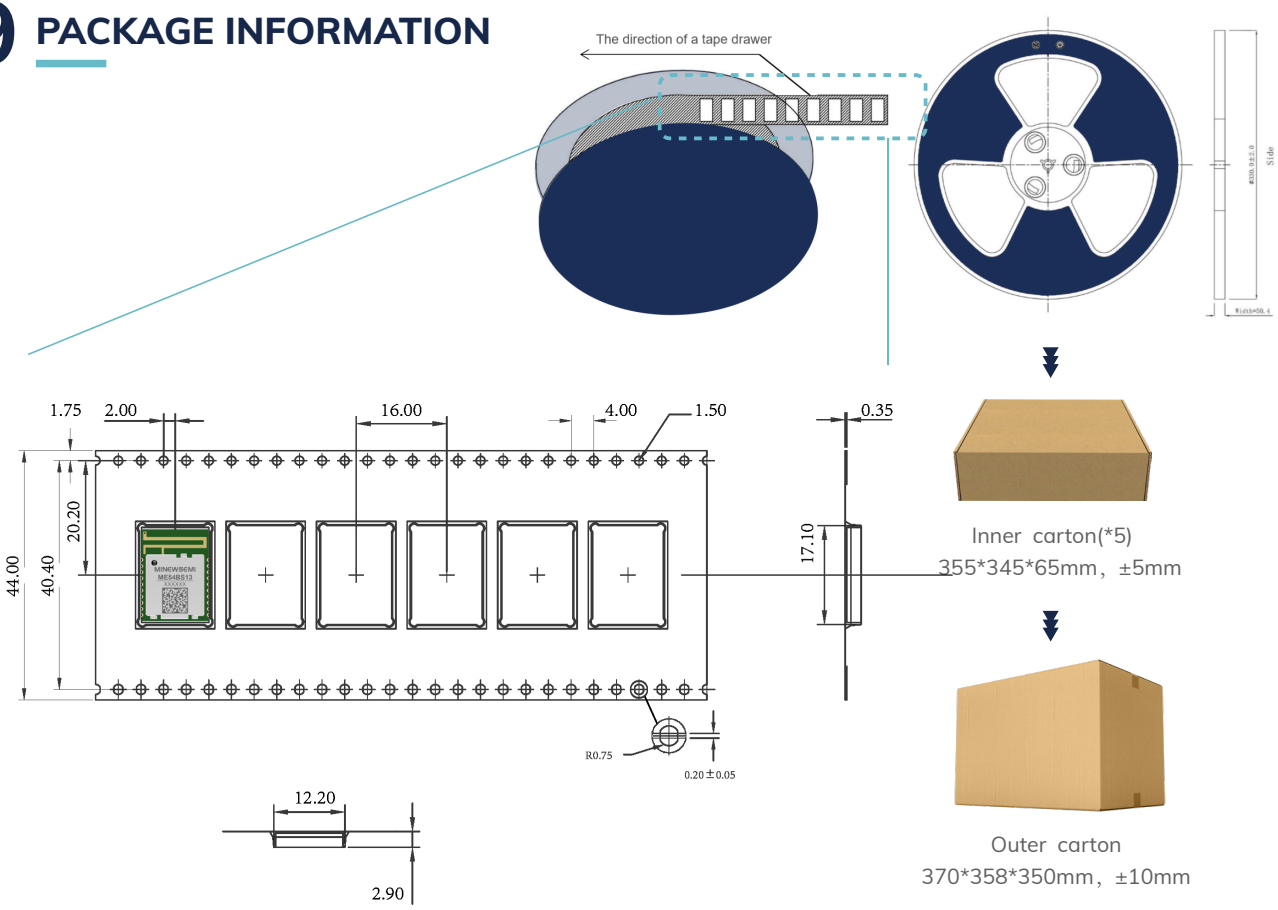
8 REFLOW AND SOLDERING

1) Perform SMT according to the reflow oven temperature profile provided below, with a maximum temperature of 260°C;
2) Follow IPC/JEDEC standards; Peak temperature: < 260°C; Number of reflows: ≤2 times; For SMT involving double-sided placement, it is recommended that the module side undergoes reflow soldering only once. For any special processes, please contact our company.

3) For SMT modules, a stencil thickness of 0.13–0.15 mm is recommended; round holes should not be enlarged, and the outer edge of half-hole pad openings should extend 1–1.3 mm.
4) After opening, if the entire package is not used at once, it should be stored in a vacuum to prevent long-term exposure to air, which can cause moisture absorption and pad oxidation. If there is a gap of 7 to 30 days before reuse, it is recommended to bake the tape at 65-70°C for 24 hours without unrolling it before returning to SMT.
5) ESD protection measures should be implemented before using SMT.



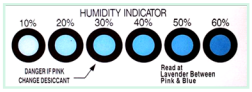
9 PACKAGE INFORMATION



Remarks

General material list for FCL packaging:

Carrier tape packaging tray



Humidity Indicator
(1 pcs/bag)

Desiccant
(placed in a vacuum bag)

Vacuum bag

Inner carton(*5)
355*345*65mm，±5mm

Outer carton
370*358*350mm，±10mm

Other:

Moisture-proof label (attached to the vacuum bag)

Certification label (attached to the vacuum bag)

Outer box label

Default unit：mm Default tolerance：±0.1

Packing detail	Specification	Net weight	Gross weight	Dimension
ME54BS13	1100 PCS	880 g	2276 g	W=44mm，T=0.35mm

Note: Default weight tolerance all are within 10g（except the special notes）

10 STORAGE CONDITIONS

- **Please use this product within 6 months after signing the receipt.**
  - This product should be stored without opening the package at an ambient temperature of 5~35°C and a humidity of 20~70%RH.
  - This product should be left for more than 6 months after receipt and should be confirmed before use.
  - The product must be stored in a non-corrosive gas (CI2, NH3, SO2, NOx, etc.).
  - To avoid damaging the packaging material, do not apply any excessive mechanical shocks, including but not limited to sharp objects adhering to the packaging material and product dropping.

- **This product is suitable for MSL2 (based on JEDEC standard J-STD-020).**
  - After opening the package, the product must be stored at ≤30°C/<60%RH. It is recommended to use the product within 3-6 months after opening the package.
  - When the color of the indicator in the package changes, the product should be baked before welding.

- **Baking is not required for one year if exposure is limited to <30°C and 60%RH. Refer to MSL2 for exposure criteria for moisture sensitivity level. If exposed to (≥168h@85°C/60%RH) conditions or stored for more than one year, recommended baking conditions.**
  1. 120 +5/-5°C, 8 hours, 1 time
  Products must be baked individually on heat-resistant trays because the materials (base tape, reel tape, and cover tape) are not heat-resistant, and the packaging material may be deformed at temperatures of 120℃;
  2、90℃ +8/-0℃，24hours，1times
  The base tape can be baked together with the product at this temperature. Please pay attention to the uniformity of heat.

11 HANDLING CONDITIONS

• Be careful in handling or transporting products because excessive stress or mechanical shock may break products.
• Handle with care if products may have cracks or damages on their terminals. If there is any such damage, the characteristics of products may change. Do not touch products with bare hands that may result in poor solder ability and destroy by static electrical charge.

12 QUALITY

Cognizant of our commitment to quality, we operate our own factory equipped with state-of-the-art production facilities and a meticulous quality management system. We hold certifications for ISO9001, ISO14001, ISO27001, OHSA18001, BSCI.

Every product undergoes stringent testing, including transmit power, sensitivity, power consumption, stability, and aging tests. Our fully automated module production line is now in full operation, boasting a production capacity in the millions, capable of meeting high-volume production demands.

13 COPYRIGHT STATEMENT

This manual and all the contents contained in it are owned by Shenzhen Minewsemi Co., Ltd. and are protected by Chinese laws and applicable international conventions related to copyright laws.

The certified trademarks included in this product and related documents have been licensed for use by MinewSemi. This includes but is not limited to certifications such as BQB, RoHS, REACH, CE, FCC, BQB, IC, SRRC, TELEC, WPC, RCM, WEEE, etc. The respective textual trademarks and logos belong to their respective owners. For example, the Bluetooth® textual trademark and logo are owned by Bluetooth SIG, Inc. Other trademarks and trade names are those of their respective owners. Due to the small size of the module product, the "®" symbol is omitted from the Bluetooth Primary Trademarks information in compliance with regulations.

The company has the right to change the content of this manual according to the technological development, and the revised version will not be notified otherwise. Without the written permission and authorization of the company, any individual, company, or organization shall not modify the contents of this manual or use part or all of the contents of this manual in other ways. Violators will be held accountable in accordance with the law.

14 RELATED DOCUMENTS

- MinewSemi_Product_Naming_Reference_Manual
  https://en.minewsemi.com/file/MinewSemi_Product_Naming_Reference_Manual_EN.pdf
- MinewSemi_Connectivity_Module_Catalogue
  https://en.minewsemi.com/file/MinewSemi_Connectivity_Module_Catalogue_EN.pdf

For product change notifications and regular updates of Minewsemi documentation, please register on our website: **www.minewsemi.com**

MINEWSEMI
Innovative IoT Module Expert

SHENZHEN MINEWSEMI CO., LTD.

0086-755-2801 0353
https://minewsemi.com
minewsemi@minew.com
https://store.minewsemi.com
Gangzhilong Technology Park, Qinglong Road, Longhua District, Shenzhen