

MBN52832 Data Sheet

BLE Module



Revision History

Revision	Date	Author	Change Description
0.1	06/03/2016	RF PD	Preliminary version
0.2	06/16/2016	RF PD	Modified the module block diagram.
0.3	8/25/2016	RF PD	Update test data points
0.4	8/27/2016	RF PD	Update module dimension and reference schematic
0.5	12/15/2016	RF PD	Applied model/type name MBN52832 to the datasheet
0.6	02/01/2017	RF PD	Update Table 2.1 and Figure 2.2
0.7	03/15/2017	RF PD	Add sleep current
0.8	05/26/2017	RF PD	Table 2.2, change pin 36, 37 pin 36 description to NC. Add section 6.1 and 6.2, recommendation for host PCB landing and placement. Updated reference schematic. Add section 9 and section 10, assembly and packaging information. Add section 12 Notice. Add section 13 Preconditions to use the module.
0.9	07/27/2017	RF PD	Add regulatory section
0.92	8/23/2017	RF PD	Update layout guidance for antenna connection; updated regulatory information
1.0	10/06/2017	Murata	Update the max height to 0.9mm, remove the sample part number.
1.1	06/14/2018	Murata	Remove Section 17, support contact.
1.2	10/10/2018	Murata	Added FCC and IC certification update for portable application in section 11.6
1.3	11/13/18	Murata	Corrected typos in section 7.

Table of Contents

REVISION HISTORY	1
TABLE OF CONTENTS	2
1 INTRODUCTION	5
1.1 Features	5
1.2 Applications	5
1.3 Block Diagram	6
1.4 Acronyms	6
1.5 References	6
2 MECHANICAL SPECIFICATION	7
2.1 Module Dimensions	7
2.2 Top and Side View	7
2.3 PCB Footprint Top View	8
2.4 Pin Configuration	9
3 DC ELECTRICAL SPECIFICATION	12
3.1 Typical Power Consumption	12
4 RF SPECIFICATION	12
5 ENVIRONMENTAL SPECIFICATION	13
5.1 Absolute Maximum Rating	13
5.2 Recommended Operating Condition	13
6 POWER SEQUENCE	13
7 APPLICATION INFORMATION	14
7.1 Recommended PCB Landing Pattern	14
7.2 Host PCB Layout Recommendations	15
7.3 Layout Guidance for Using Internal PCB Antenna	16
7.4 Layout Guidance for Microstrip Design And External Antenna	17
8 APPLICATION REFERENCE	20
9 ASSEMBLY INFORMATION	21
10 PACKAGING AND MARKING INFORMATION	22
10.1 Dimensions of Tape (Plastic tape)	22
10.2 Dimensions of Reel	23
10.3 Taping Diagrams	24
10.4 Leader and Tail tape	25
10.5 Peeling Force	26
10.6 PACKAGE (Humidity proof Packaging)	26
10.7 Module Marking Information	27
10.8 Moisture Sensitivity Level	27
11 REGULATORY INFORMATION	28
11.1 FCC Notice (USA)	28
11.2 FCC Labeling Requirements	29
11.3 IC Notice (Canada)	29
11.4 IC Labeling Requirements	30
11.5 ESTI compliance (Europe)	30
11.6 RF Exposure	30

7.3 Layout Guidance for Using Internal PCB Antenna

The MBN52832 module is certified with an internal PCB antenna physically located on the module. For optimum EIRP (Effectively Isotropically Radiated Power) when use the on-board antenna, please follow the following recommendations in the customer host circuit board design.

- A. The module is recommended is be placed on top left corner of the host circuit board.
- B. Around the antenna area, all layers of the customer circuit board should be free of any metal objects. Specifically, there should be no ground plane, traces or metal shield case
- C. Host circuit PCB length is longer than 40mm to get optimal performance by using ground plane.
- D. Right side of antenna area also should be free of grounds.
- E. Metal and plastic materials should be away from the module. (more than 3mm)

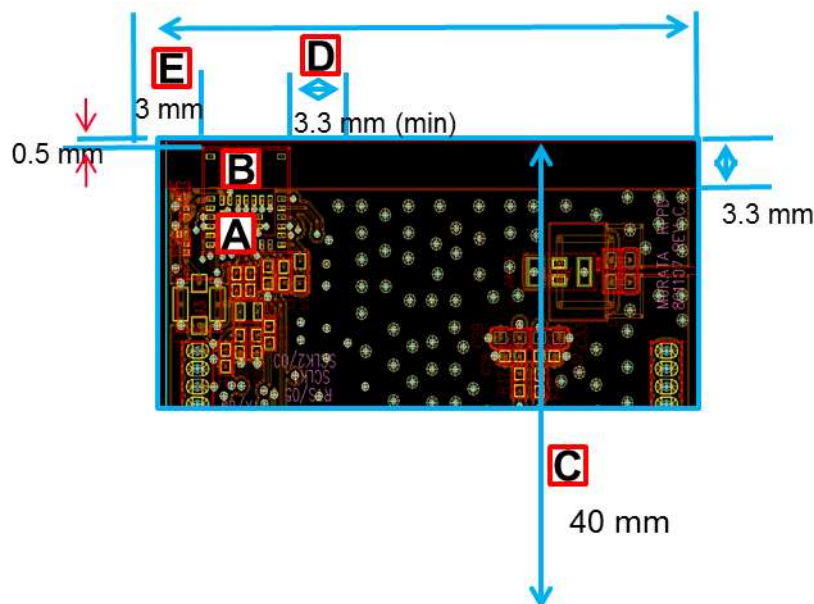


Figure 8.3 Layout Guide for Antenna Performance (1)

- F. To use this internal antenna, the integrator must provide a simple two-component matching circuit between pins 20 and 21 of the module. This circuit will connect the RF I/O of the module directly to the PCB antenna. The picture below shows the location of pins 20 and 21.

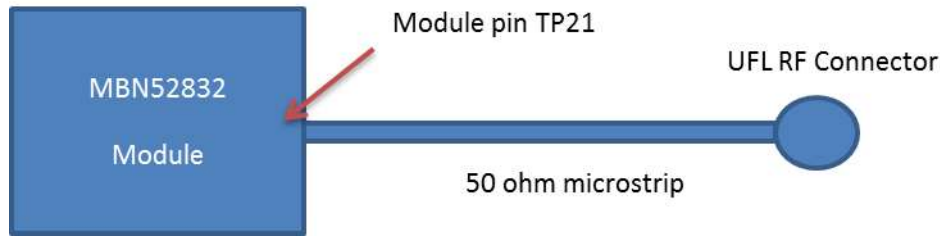


Figure 8.5 Layout Guide for External Antenna (1)

The design of the 50 ohm microstrip on the customer's PCB is crucially important. Compliant operation of the MBN is dependent on proper construction of this 50 ohm line and the following guidelines must be followed to ensure legal operation of the product.

The diagram below shows the required microstrip structure to be routed between module pin TP21 and the UFL connector.

The top PCB trace carries the RF energy from module to UFL connector. The Layer2 ground plane provides a return path for the circuit. The Dielectric material (along with the dimensions of the microstrip structures) determines the characteristic impedance of the microstrip transmission line.

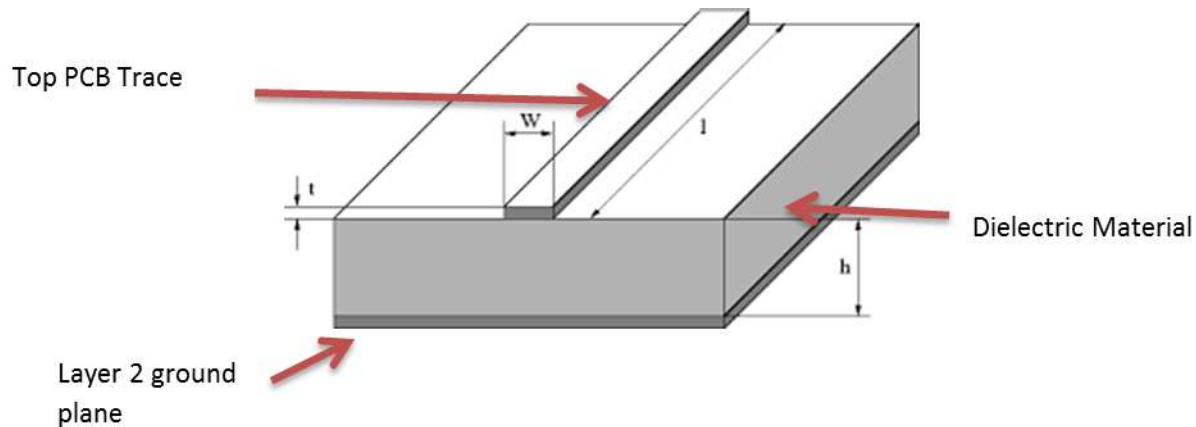


Figure 8.6 Layout Guide for External Antenna (2)

Note the representative dimensions shown in the drawing above. It is imperative that the module customer (the integrator) use the exact dimensions we recommend to ensure a 50-ohm impedance for this transmission line.

The following dimensions and/or ratios should be used to set the microstrip impedance to 50 ohms.

Dielectric (PCB) Material – We recommend standard FR-4 PCB material. Other dielectrics will work but will require recalculation of microstrip dimensions. The following guidance is predicated on the use of FR-4 Dielectric. If FR-4 is not used for PCB material, please contact Murata Electronics at (678) 684-2009 to determine new dimensions for microstrip structure.

H (Dielectric Height) – this is the thickness of dielectric between the trace layer (layer 1) and the ground plane on layer 2. Note that layer 2 must be electrical ground. We recommend a dielectric thickness of 8-15 mils. This range provides the customer with some flexibility in board construction.

t (trace thickness) – Microstrip impedance is not severely affected by the thickness dimension. Standard 1oz or 2oz copper deposition is recommended. Equivalent thickness is 1-2 mils.

W (trace width) – this is the crucial dimension. This width must be set correctly to obtain the desired 50 ohms impedance. When using FR-4 dielectric, the width (W) of the microstrip trace should be set to:

$$W = H * 1.8$$

Where W is microstrip trace width and H is Dielectric height. Note that both values must be measured in identical units (mils or mm)

Example:

$$H = 12 \text{ mils}, W = 12 * 1.8 = 21.6 \text{ mils}$$

$$H = 0.4 \text{ mm}, W = 0.4 * 1.8 = 0.72 \text{ mm}$$

l (trace length) – the impedance of the microstrip line is not dependent on its length. However, regulatory and performance limitations practically determine the actual length to be used by the customer (integrator). The length of this microstrip line must be longer than 7 mm to mimic the length used during FCC/IC certification of the MBN52832 module. Lengths longer than 7 mm are acceptable although additional signal loss will occur as a result. Given these restrictions, Murata recommends microstrip trace lengths between 7 mm and 25 mm.

In any event, the microstrip line must operate over the same Dielectric-Ground Plane configuration shown above to act as a 50 ohm transmission line. Do not run the microstrip trace through sections of PCB that do not have the Dielectric-Ground plane configuration shown above.

A reliable 50 ohm transmission line will be produced if the above guidance is closely followed. Any deviations from the guidance above may cause the module to operate in noncompliant manner. Any implementation questions or concerns should be directed to Murata module technical support. .