MBN52832 BLE Module Layout Guide

Murata RF PD

Content

- Layout Guidance for Using Internal PCB Antenna
  - Connection
  - Location
  - Orientation
  - Antenna Performance
- Layout Guidance for Microstrip Design And External Antenna

On-board antenna Instruction

A. The module is recommended is be placed on top left corner of the host circuit board.

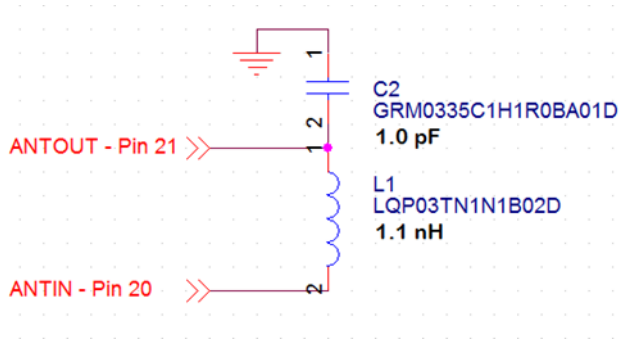
B. Around the antenna area, all layers of the customer circuit board should be free of any metal objects. Specifically, there should be no ground plane, traces or metal shield case

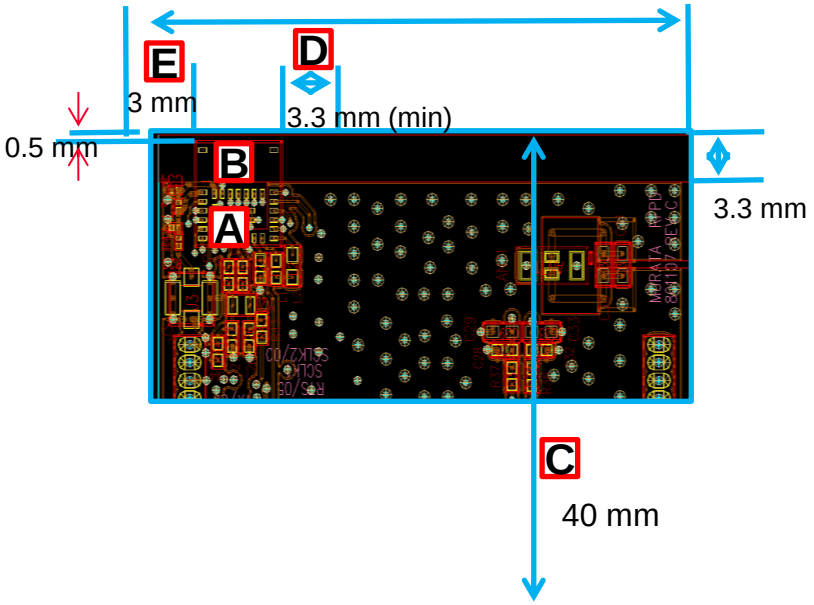
C. Host circuit PCB length is longer than 40mm to get optimal performance by using ground plane.

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)

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.





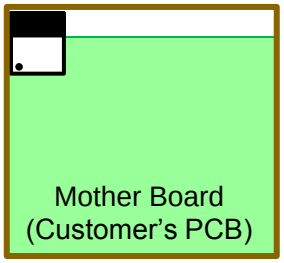
On-board antenna connection

- The Integrator must place the matching circuit shown below between Pins 20 and 21 of the Module to duplicate the structure used during FCC/IC testing of the MBN52832. Use the exact PNs provided for L1 and C2 to meet regulatory requirements.
- On the integrator’s support board, place L1 between Pin 20 and Pin 21 of the Module. Place C2 adjacent to Pin 21. The other side of C2 should be attached to RF Ground.

Module location on the PCB

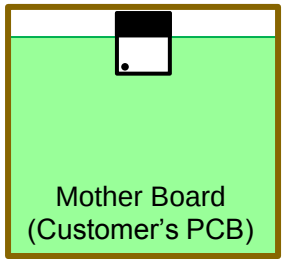
- The module should be located at the corner of PCB.

Module is located at the Left corner



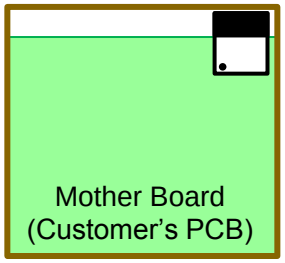
Good

Module is located at the Middle edge



Not Recommended

Module is located at the Right corner

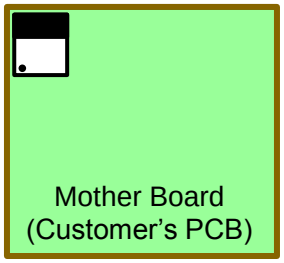


Acceptable

Keeping away ground plane

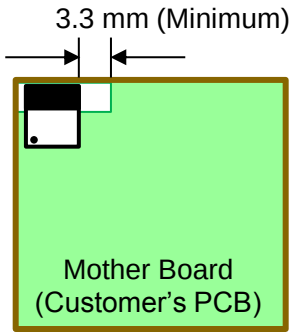
- Ground plane should be kept away from the antenna area.

Ground plane exists even under Antenna Area



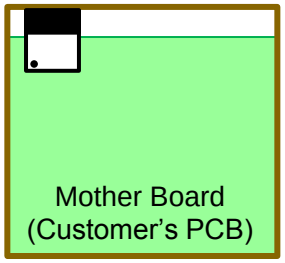
Not Recommended

Ground plane is close to the Antenna Area



Not Recommended

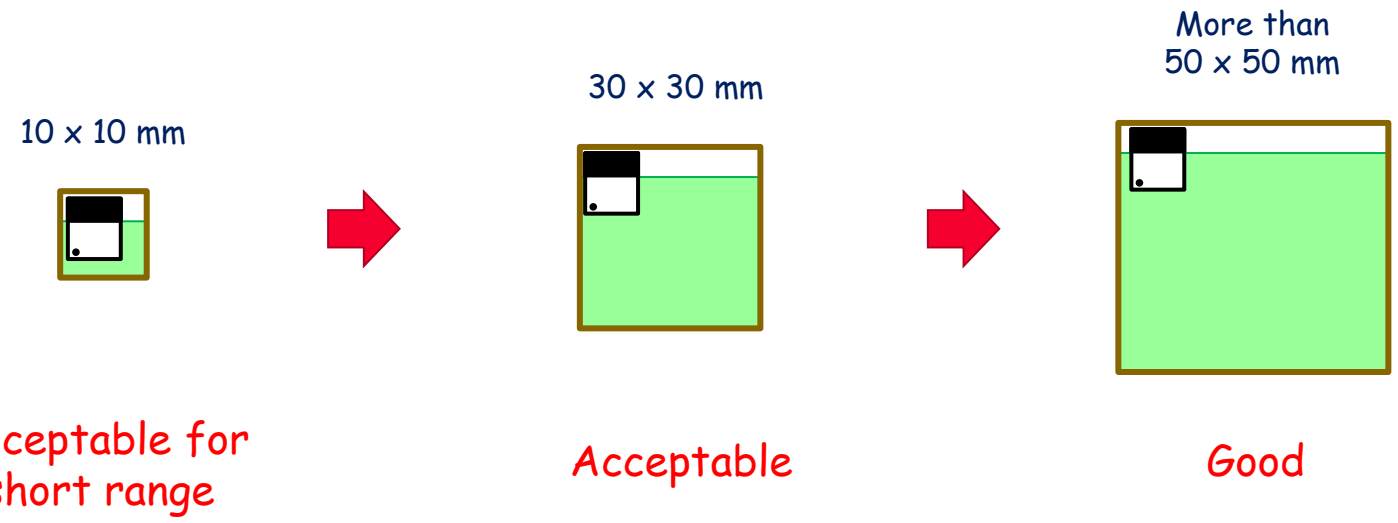
Ground plane is kept away from the Antenna Area



Good

Size of PCB

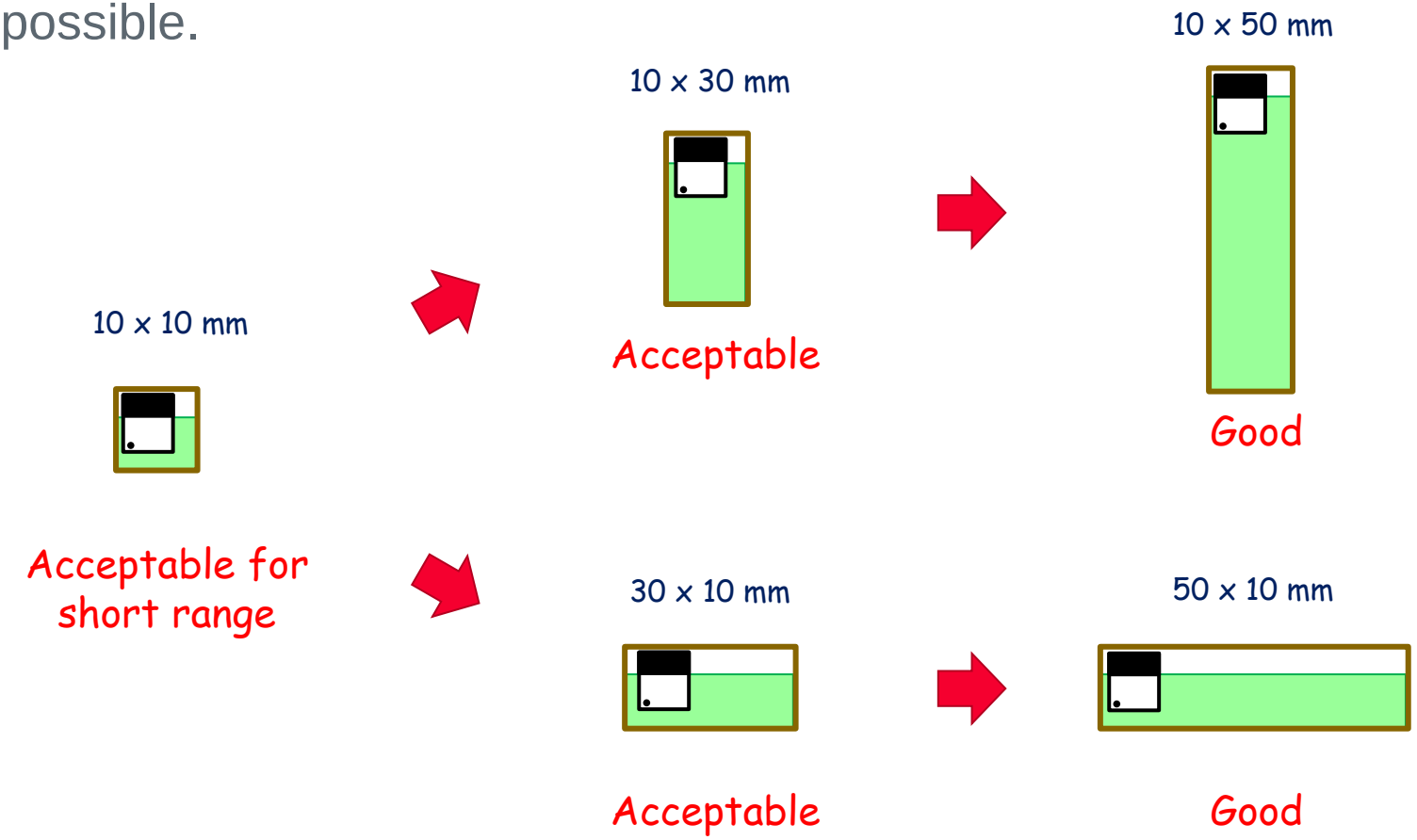
- Size of PCB is better around 50mm.



- If installed on a small PCB, connecting PCB's ground to the common ground of the unit is preffered because total ground can be seen larger.

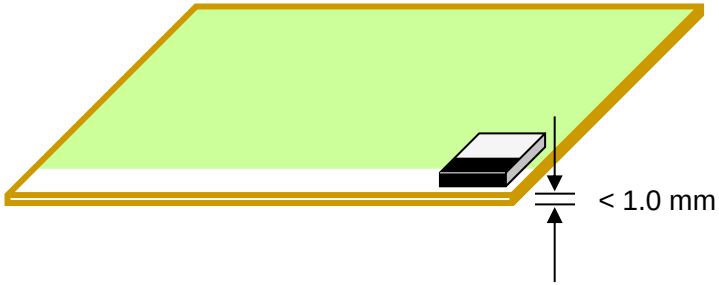
PCB Orientation

- In case of slim PCB, the dimension of Y should be as long as possible.

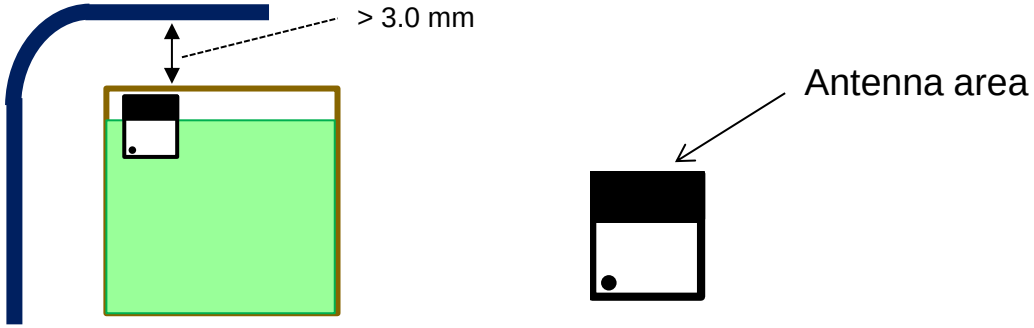


Other suggestions

- The thickness of PCB should be as thin as possible.



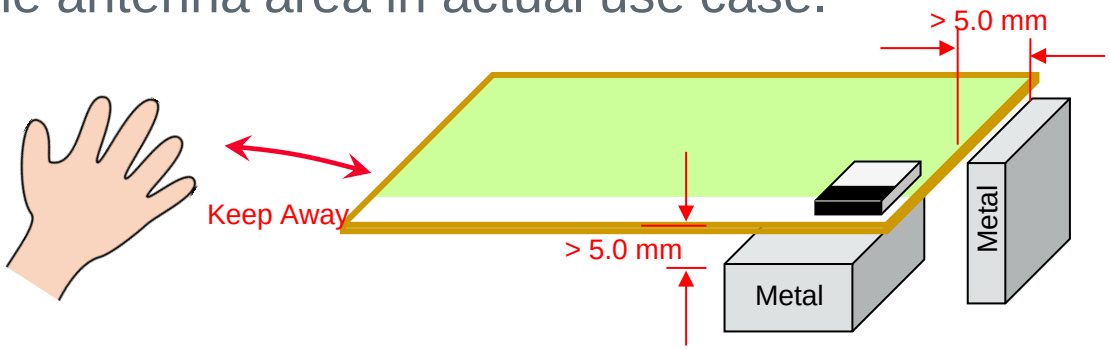
- Plastic case should be kept away from the antenna area as far as possible.



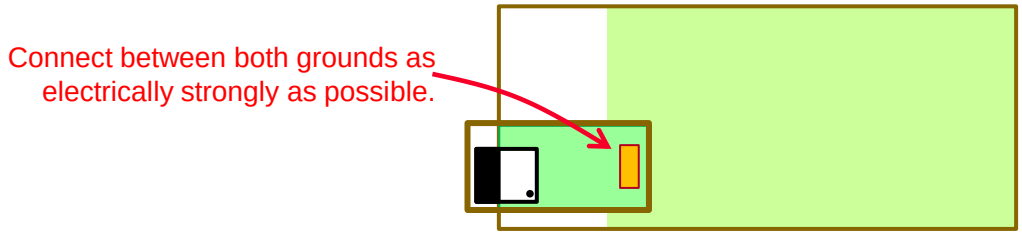
- Any metal elements should not be close to the antenna area.
- User's hands and human body should be kept away from the antenna area in actual use case.

Other suggestions

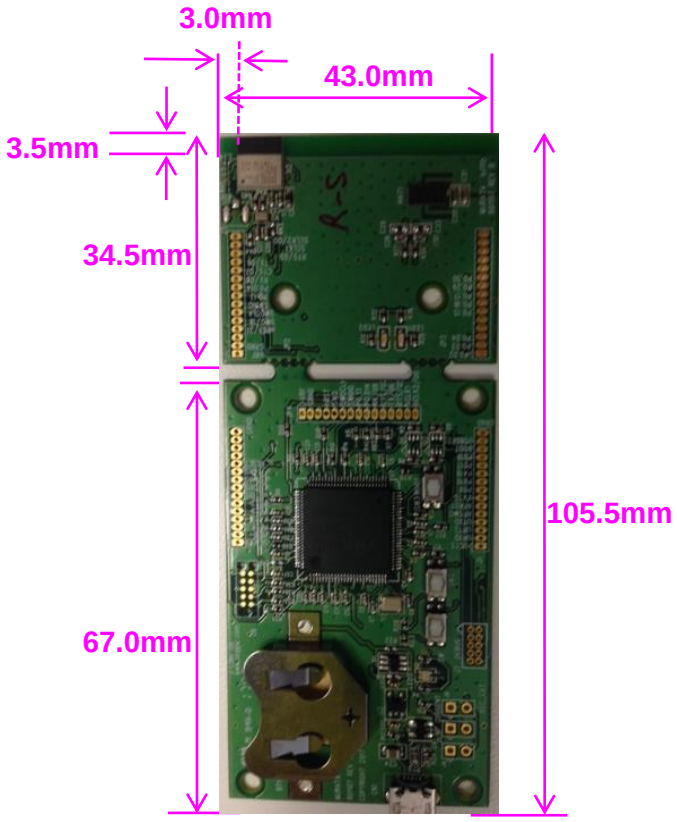
- Any metal elements (ex. connector, battery, etc.) should not be close to the antenna area. It should be at least 5mm away.
- User's hands and human body should also be kept away from the antenna area in actual use case.



- If the module is located on a small daughter board, please make ground connected strongly and regard ground plane including entire main board.

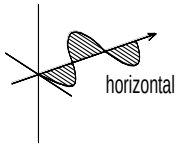


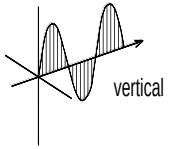
MBN52832 Development Kit



MBN52832 Antenna Performance

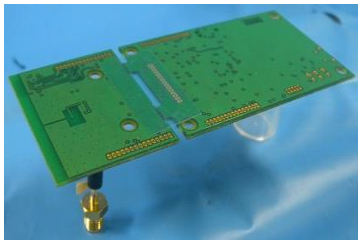
Measurement setup



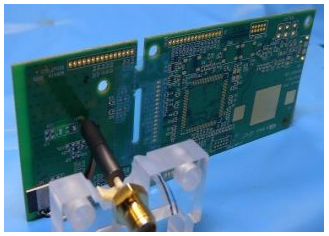


XY plane

YZ plane

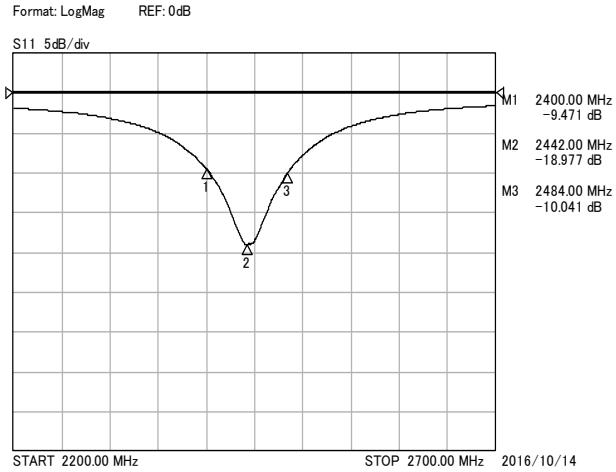


ZX plane



MBN52832 Antenna Performance

Return Loss



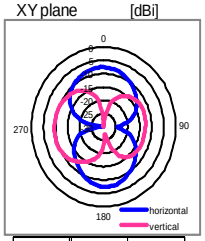
Efficiency

[dBi] [dB]

LINEAR POLARIZATION		XY-plane hor.	XY-plane ver.	YZ-plane hor.	YZ-plane ver.	ZX-plane hor.	ZX-plane ver.	Total Efficiency
2400 MHz	MAX.	-8.1	-9.6	-2.8	-19.6	-5.1	-4.7	
	AVE.	-11.5	-13.5	-7.5	-24.1	-10.8	-6.6	-6.4
2442 MHz	**MAX.**	**-7.3**	**-9.5**	**-2.5**	**-22.0**	**-4.8**	**-4.4**	
	AVE.	**-11.0**	**-13.3**	**-7.3**	**-25.0**	**-10.7**	**-6.3**	**-6.2**
2484 MHz	MAX.	-7.8	-10.1	-2.9	-22.4	-5.7	-4.9	
	AVE.	-11.3	-14.0	-7.9	-25.1	-11.5	-6.8	-6.8

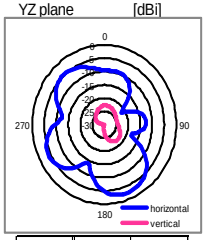
Radiation Pattern

XY plane [dBi]



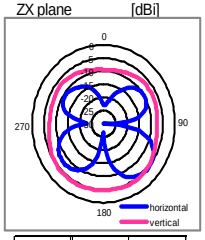
	HOR.	VER.
MAX	-7.3	-9.5
AVE	-11.0	-13.3

YZ plane [dBi]



	HOR.	VER.
MAX	-2.5	-22.0
AVE	-7.3	-25.0

ZX plane [dBi]



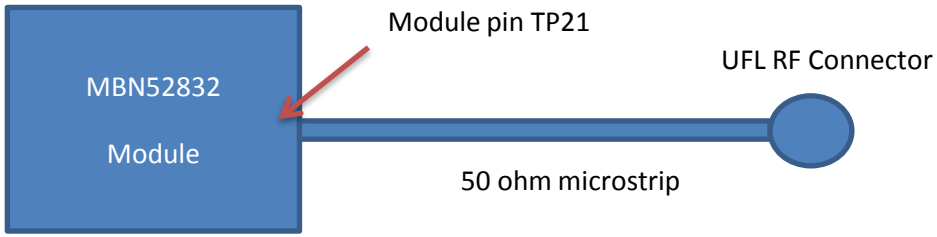
	HOR.	VER.
MAX	-4.8	-4.4
AVE	-10.7	-6.3

Microstrip Design And External Antenna

- The MBN52832 module is certified with an internal PCB antenna and two external antennas; a 7 dBi omni antenna and a 6 dBi patch antenna.
- Either patch or omni antenna should be connected to the MBN module using 50 ohm microstrip and a U.FL RF connector as shown below. This microstrip and U.FL connector are placed on the customer's PCB and are external to the MBN module. The patch or omni antenna is then connected to this UFL Connector via a 50 ohm RF adapter cable.

Part Number	Vendor	Gain (dBi)	Type	Connector	Remarks
S2406PL	Cushcraft	6.0	Patch	U.FL	Necessary for FCC/IC compliance only.
HG2407RD-RSP	L-Comm	7.0	Dipole	U.FL	Necessary for FCC/IC compliance only.

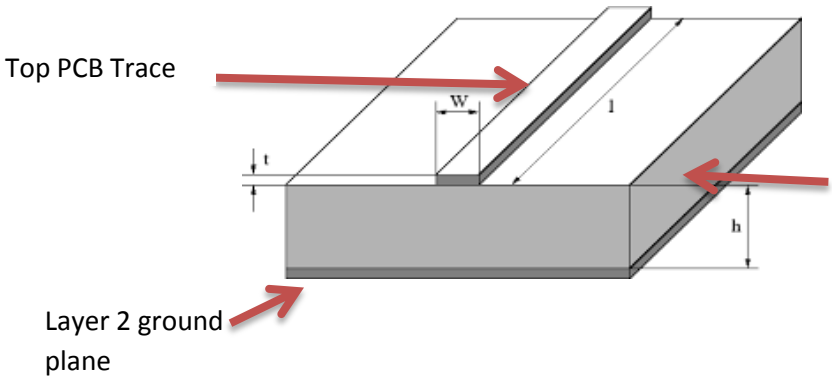
Microstrip



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.

Microstrip



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 mils, W = 12 * 1.8 = 21.6 mils
H = 0.4 mm W = 0.4 * 1.8 = 0.72 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.