APPLICATION SPECIFICATION

2.4 GHZ SMD ON-GROUND ANTENNA

1.0 SCOPE

This specification describes the antenna application and recommended PCB layout for the Molex 2.4 GHz SMD On-Ground Antenna. The information in this document is for reference and benchmark purposes only. The user is responsible for validating antenna RF performance based on users own PCB and matching circuits.

All measurements are done of the antenna mounted on the recommended PCB with VNA Agilent 5071C and OTA chamber.

Antenna illustrations in this document are generic representations. They are not intended to be an image of any antenna listed in the scope.

2.0 PRODUCT DESCRIPTION

A. DEFINITIONS OF TERMS

I. ANTENNA DESCRIPTION

The antenna design is based on carrier size 4mm × 3mm × 3mm. There is 1 feeding pad, 3 fixing pads and antenna radiator. See figure 1.

1. **FEEDING PAD**
   SMD mounted to feeding pad on PCB. The signal from the transmission line must feed into the feeding pad on the PCB.
2. **FIXING PAD**
   SMD mounted to dummy pads on PCB. Anchoring the antenna to the PCB
3. **ANTENNA RADIATOR**
   To act as a transducer that converts unguided electromagnetic wave to guided electromagnetic wave and vice versa.
4. **PICK AND PLACE FEATURE**
   To enable the antenna to be picked up by SMT machine pick up nozzle.

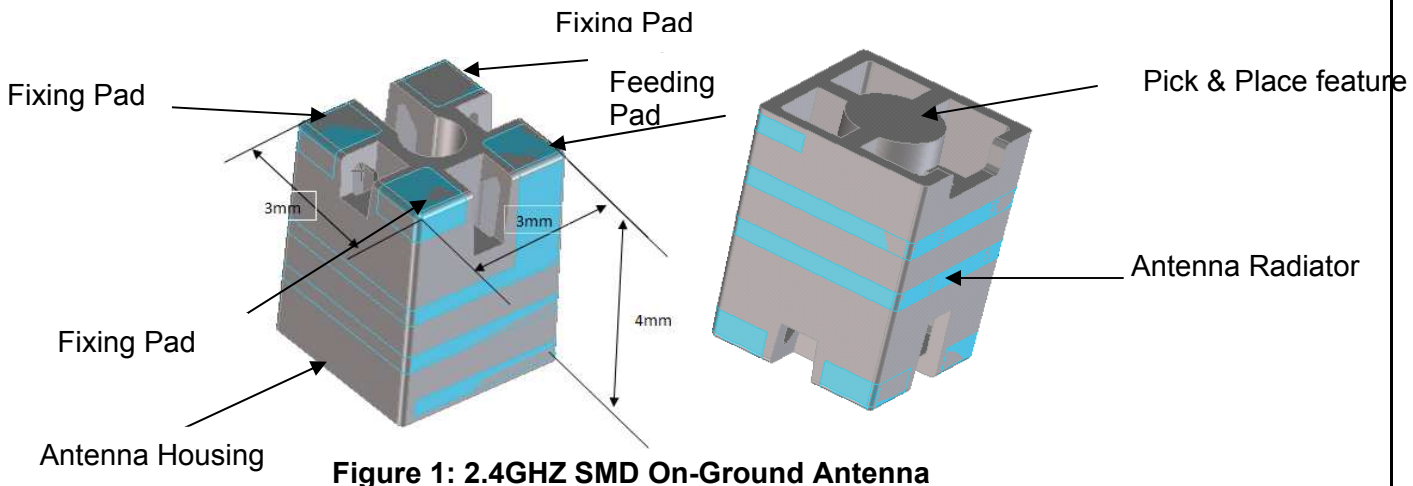

Figure 1: 2.4GHZ SMD On-Ground Antenna

B. RF SPECIFICATION OF REFERENCE IMPLEMENTATION

DESCRIPTION	TEST CONDITION	REQUIREMENT
Frequency Range	Measure antenna on recommended PCB through VNA E5071C	2400MHz-2483.5MHz
Return Loss	Measure antenna on recommended PCB through VNA E5071C	< -9 dB
Peak Gain	Measure antenna on recommended PCB through OTA chamber	3.0dBi
Avg. Total Efficiency	Measure antenna on recommended PCB through OTA chamber	Average 70%
Polarization	Measure antenna through the OTA chamber	Liner
Input Impedance	Measure antenna on recommended PCB through VNA E5071C	50Ohms

TABLE 1: Antenna Specification Using Reference Board

3.0 PROCEDURE

3.1 GENERAL REQUIREMENTS

A. REFERENCE IMPLEMENTATION

I. REFERENCE PCB DESCRIPTION

The reference design is based on a recommended double sided PCB size of 100 mm x 40 mm x 1 mm. There is 1 feeding pad, 3 fixing pads and micro-strip line on one side, and there is series matching network on the reverse side. See figure 2 and 3.

1. **FEEDING PAD**
   The signal from transmission line must feed into the feeding pad.
2. **MICRO-STRIP LINE**
   The micro-strip line provides an open end stub matching. The proposed length on the reference location on the reference PCB is 3 mm.
3. **MATCHING CIRCUIT**
   Recommended to reserve PCB space for a series matching component in case it should be needed to adjust the return loss due to loading by the devise housing and surrounding components.

II. REFERENCE PCB LAYOUT

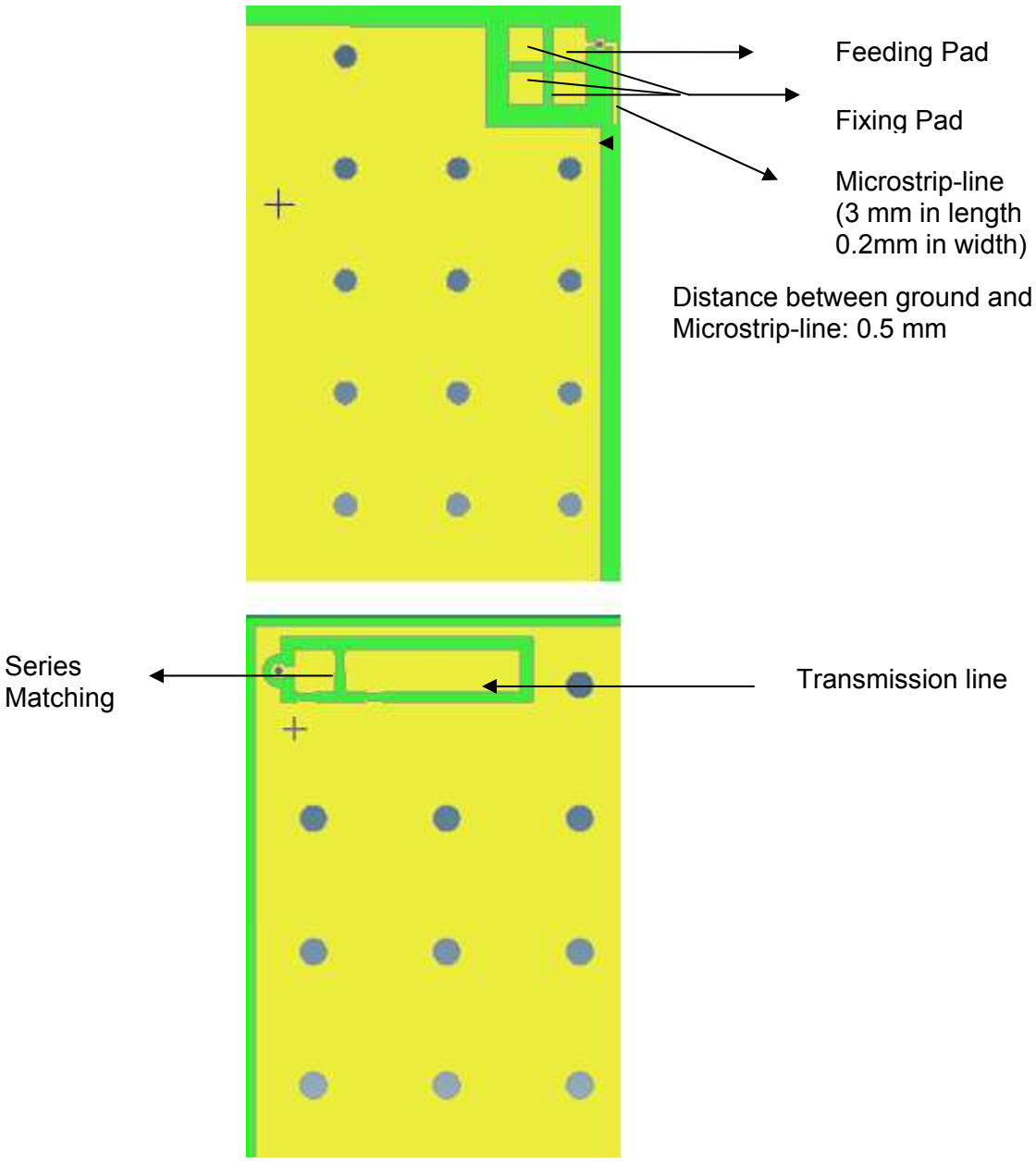

Figure 2: Recommended PCB Layout

Note: PCB size of 100 mm x 40 mm

III. REFERENCE ANTENNA LOCATION

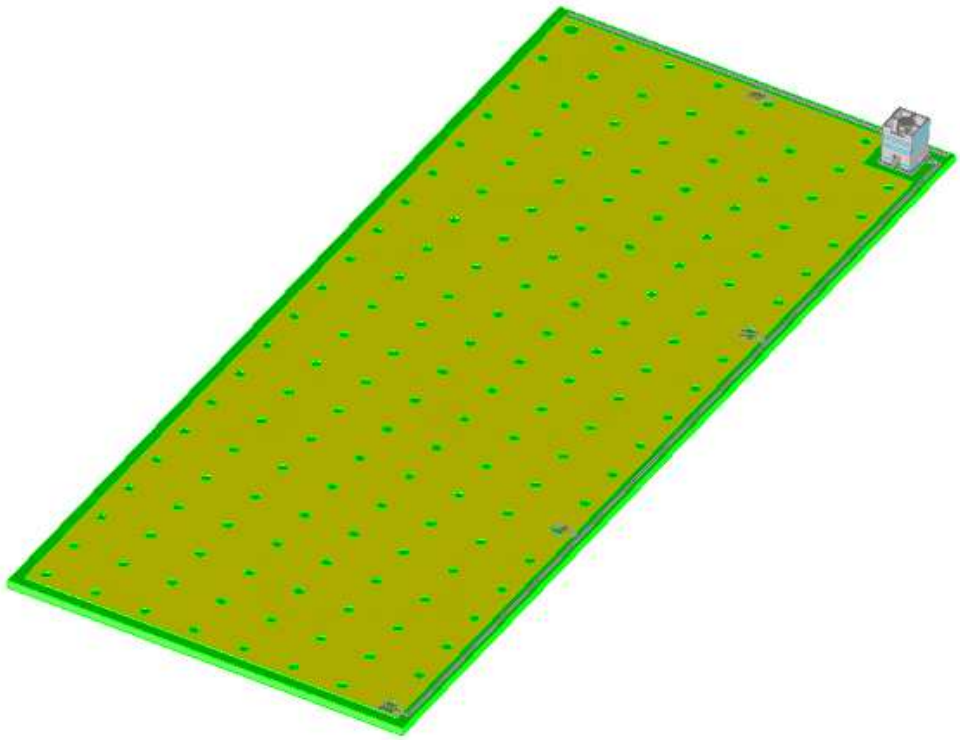

Figure 3

IV. MATCHING NETWORK DESRICPTION

The antenna is self matched with the micro-strip line. The micro-strip line provides an open end stub matching.

A series matching is needed if the resonance frequency needs adjustment due to loading by the device housing and surrounding components effect .

A series inductor will shift the frequency downward and a series capacitor will shift the frequency upward. See figure 4 and figure 5 respectively

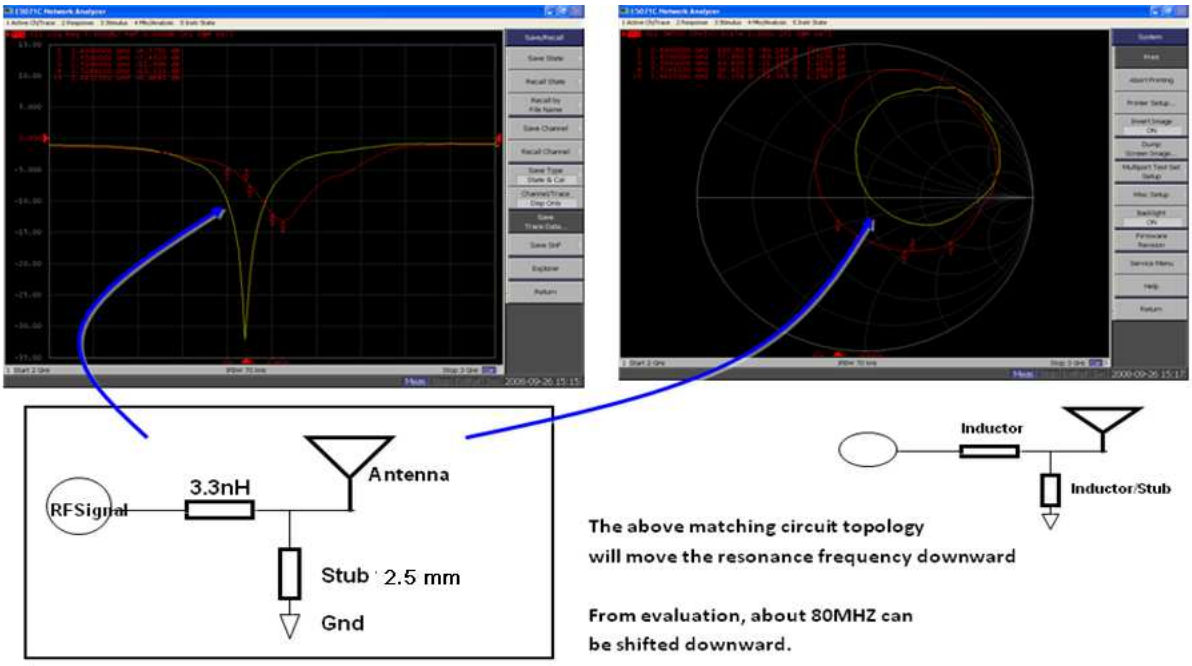

Figure 4

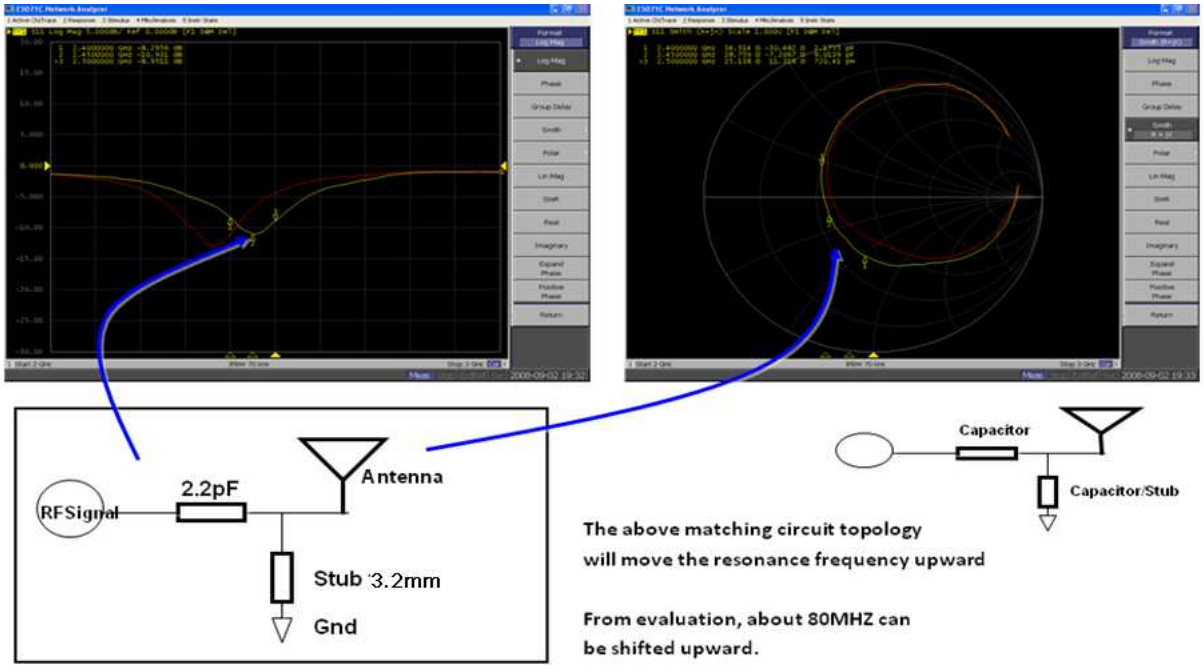

Figure 5

V. RF PERFORMANCE AS A FUNCTION OF IMPLEMENTATION

1. ANTENNA RF PERFORMANCE AS A FUNCTION OF LOCATION ON THE PCB

4 locations have been evaluated, and these locations are show in figure 8. The location which gives the best RF performance is location 1. Location 1 (corner location) is the recommended location for the antenna.

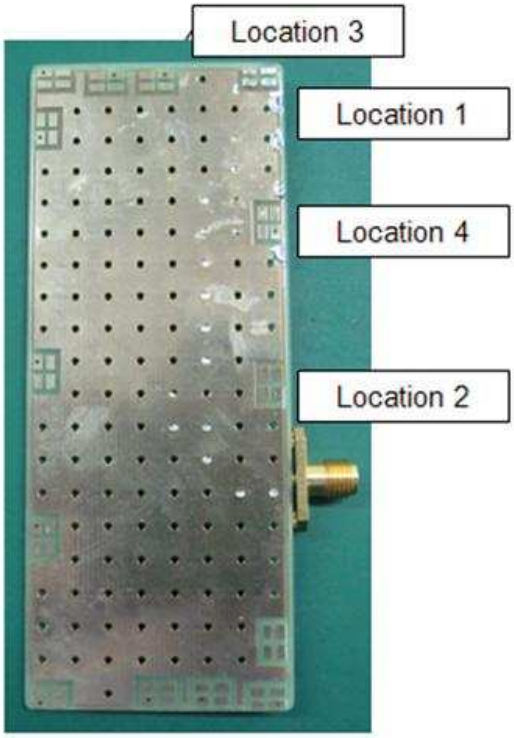

Figure 8

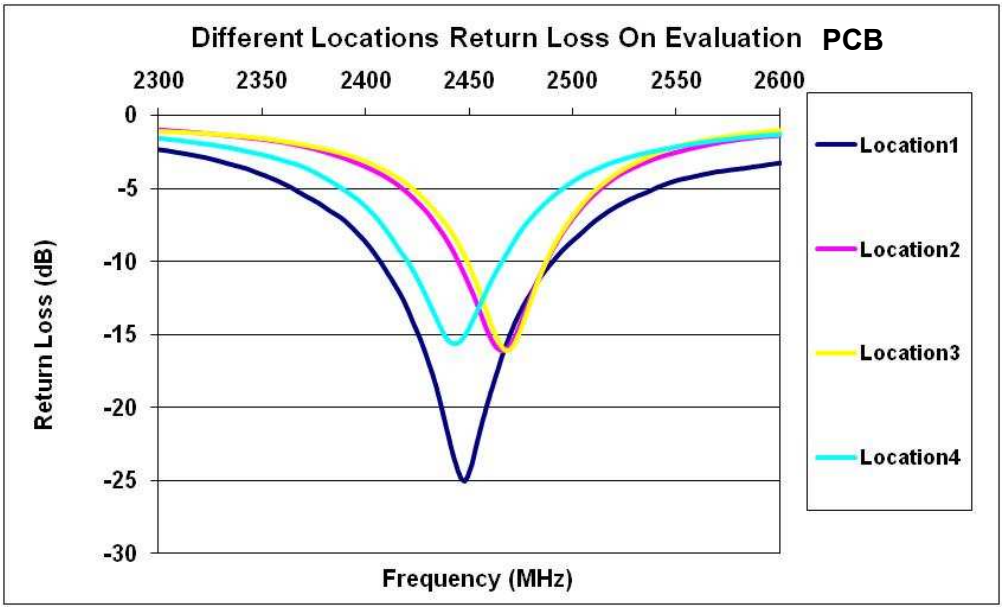

Figure 9: Return loss of comparison of different location on PCB

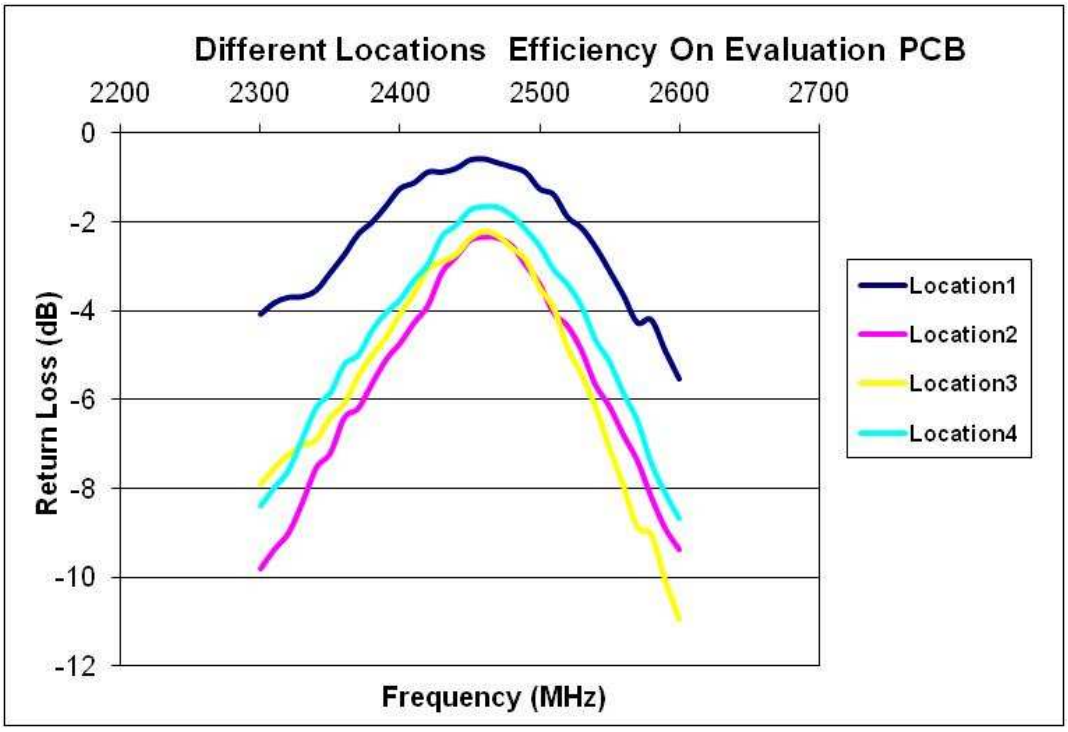

Figure 10: Total efficiency of comparison of different location on PCB

2. ANTENNA RF PERFORMANCE AS AN EFFECT OF NEARBY SHIELDING CAN

An evaluation was done with 3 different distances from the antenna which located at the recommended location. The 3 distances is as follow: 1mm, 3mm and 5mm.

From the study, we can say that the minimum distance a shielding can (30mm x 30mm x 2mmm) should be placed from the antenna is 5mm. At distances less than 5mm the antenna performance will be significantly degraded. Refer to figure 11,12 and 13.

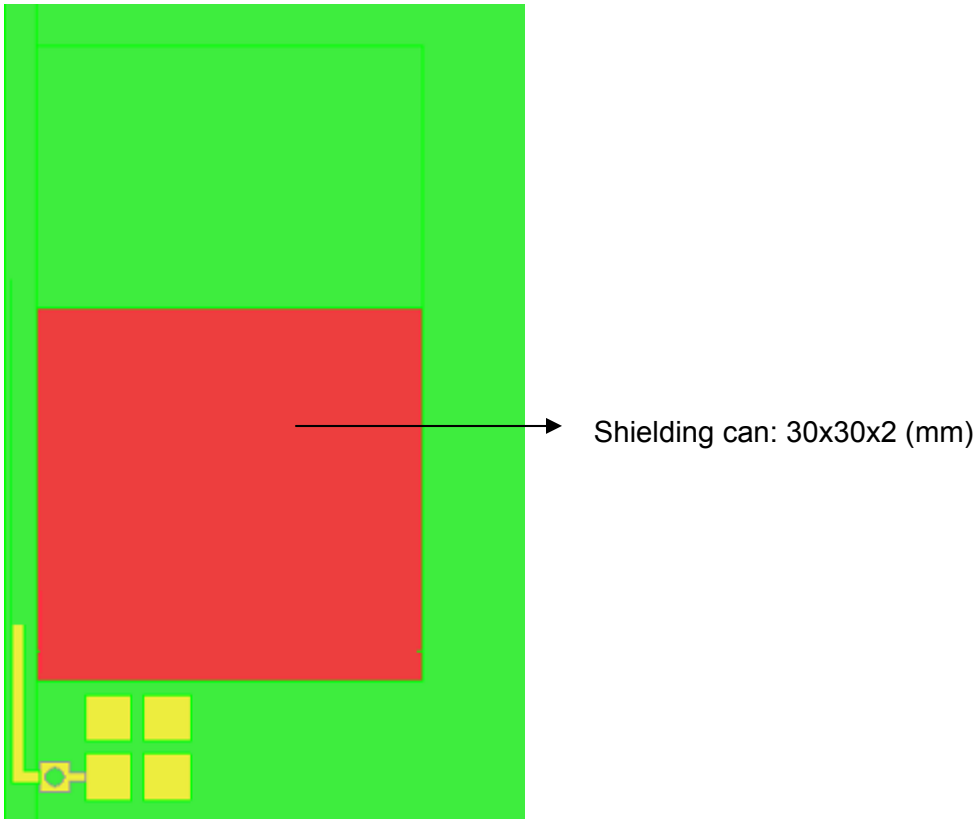

Figure 11

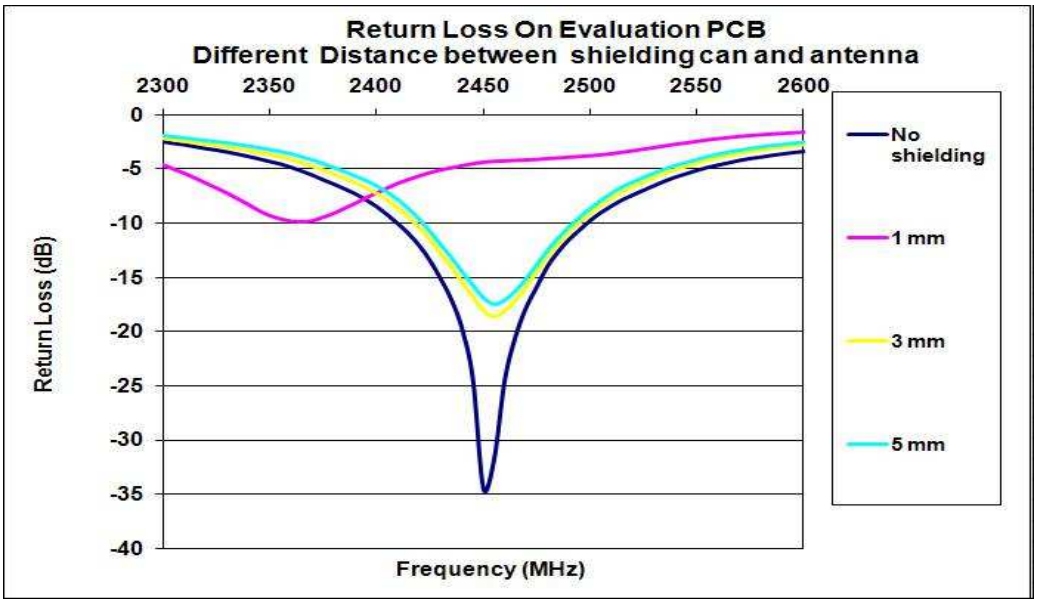

Figure 12: Return loss comparison of shielding can distance from antenna

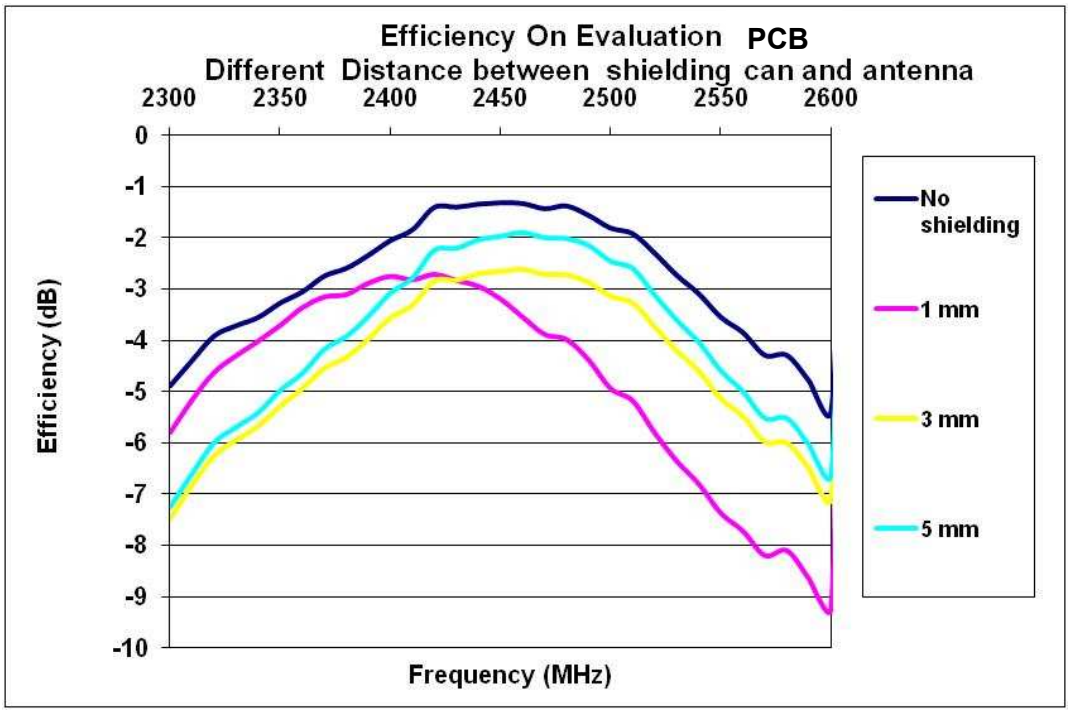

Figure 13: Total efficiency comparison of shielding can distance from antenna

3. RF PERFORMANCE AS AN EFFECT OF NEARBY BATTERY

An evaluation was done with 3 different distances from the antenna which located at the recommended location. The 3 distances is as follow: 1mm, 3mm and 5mm.

From the study, we can say that the minimum distance a battery (30mm x 60mm x 3mmm) should be placed from the antenna is 5mm. At distances less than 5mm the antenna performance will be degraded significantly. Refer to figure 14, 15 and 16.

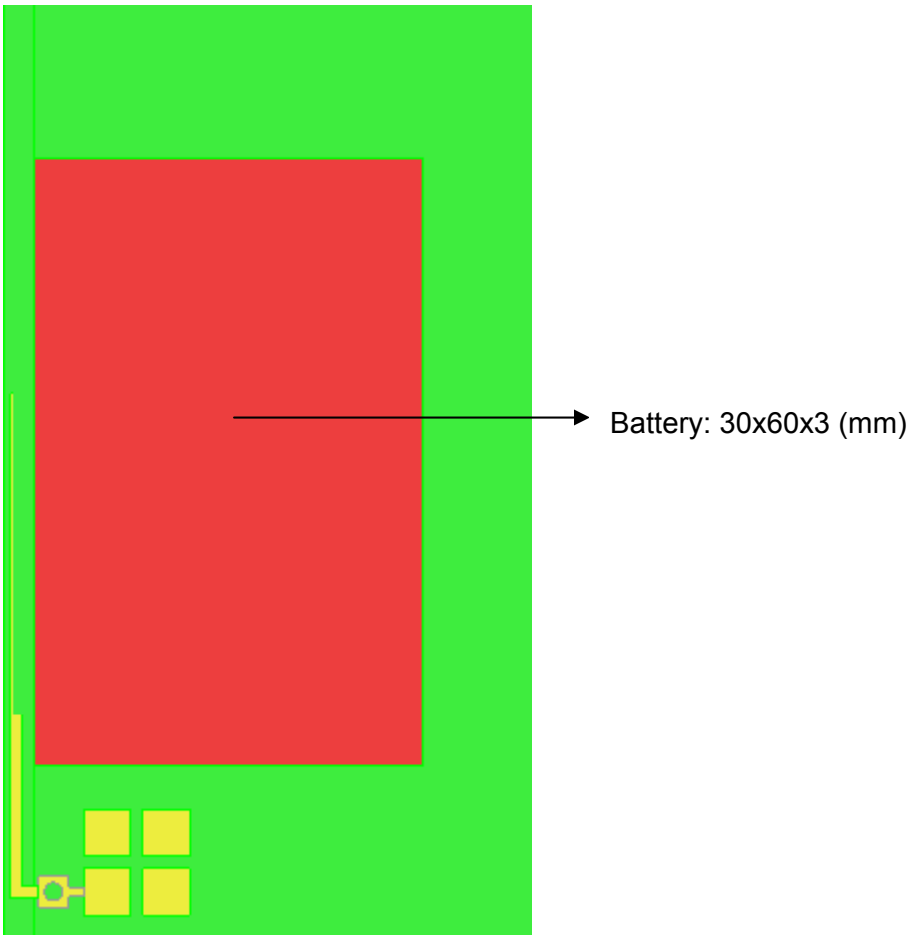

Figure 14

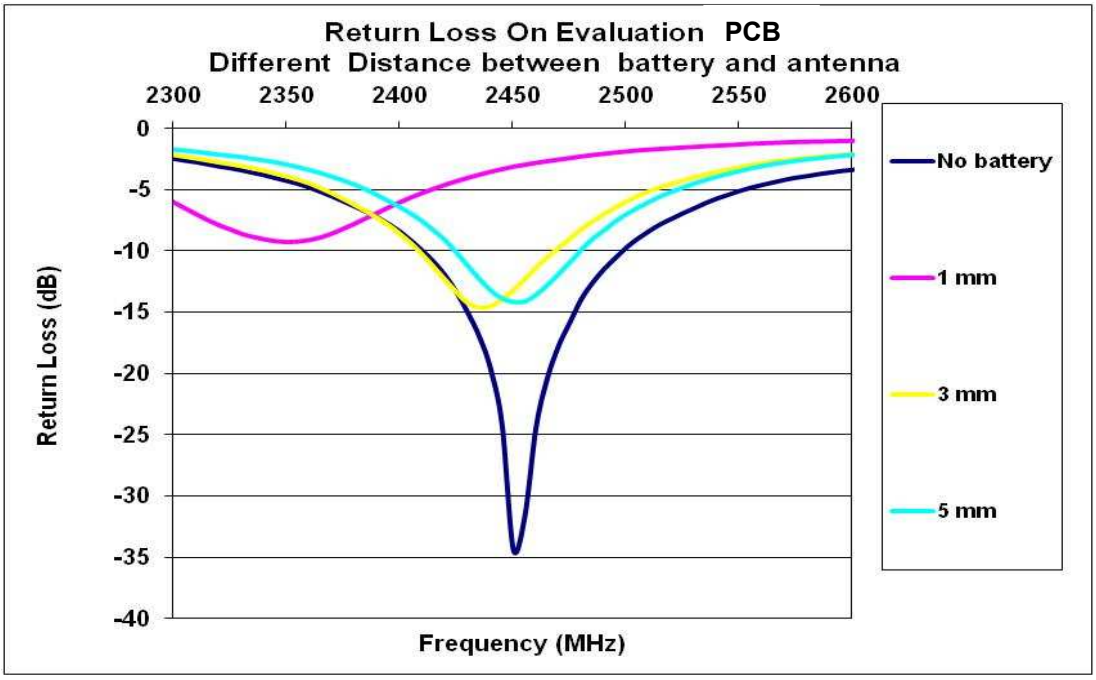

Figure 15: Return loss comparison of battery distance from antenna

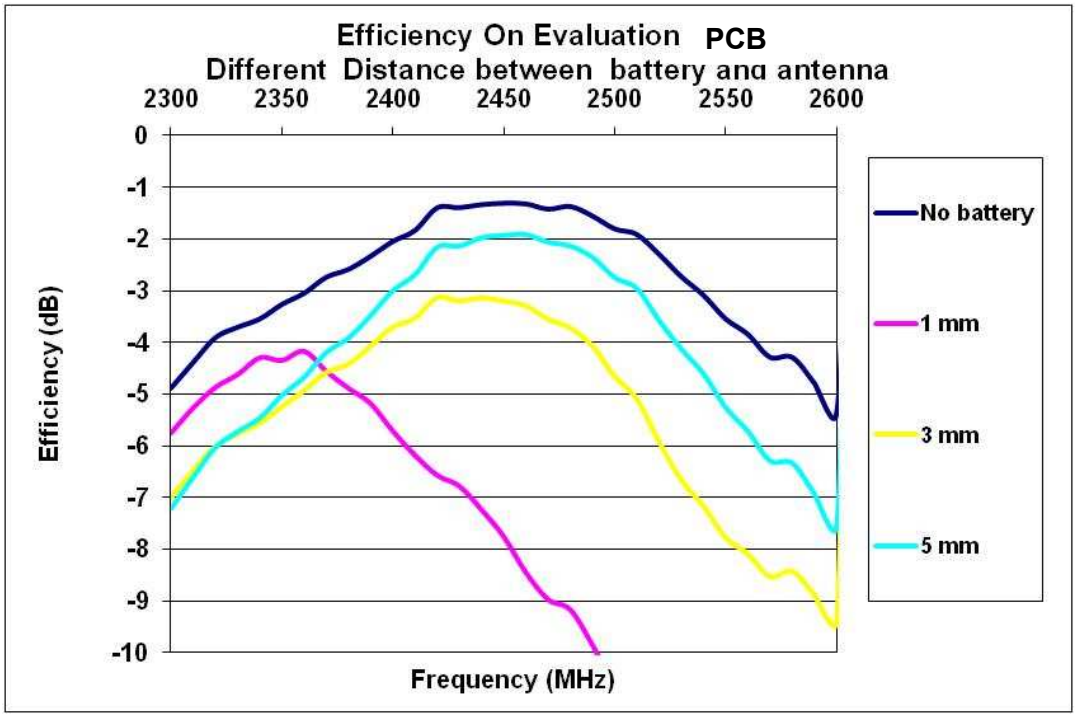

Figure 16: Total efficiency comparison of battery distance from antenna

3.2 ASSEMBLY INSTRUCTIONS

A. RECOMMENDED SMT REFLOW PROFILE

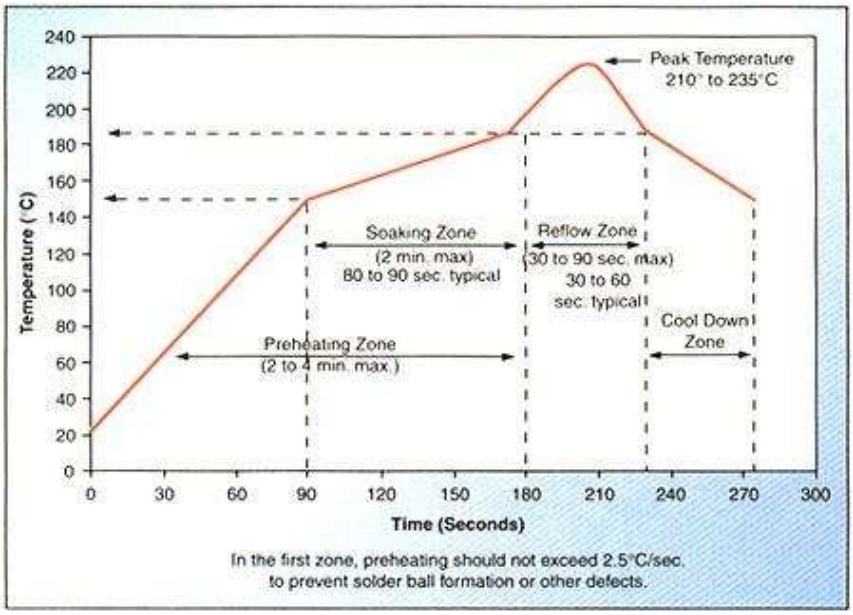

Figure 17

B. MECHANICAL INTERFACE

I. GENERAL DESCRIPTION

The overall antenna size is 4mm × 3mm × 3mm

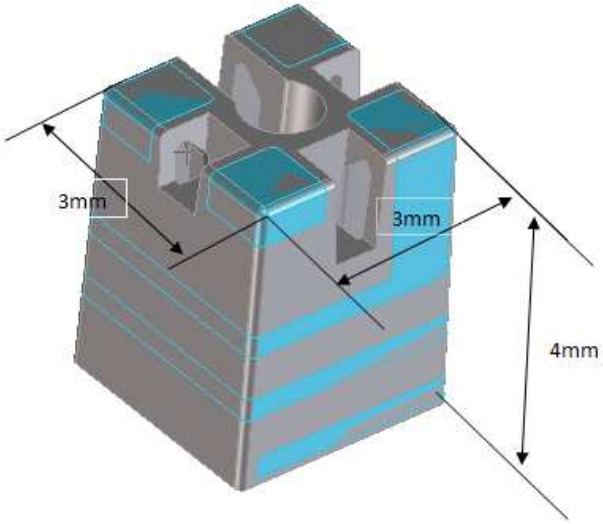

Figure 18

II. STRUCTURE FUNCTIONAL DESCRIPTION

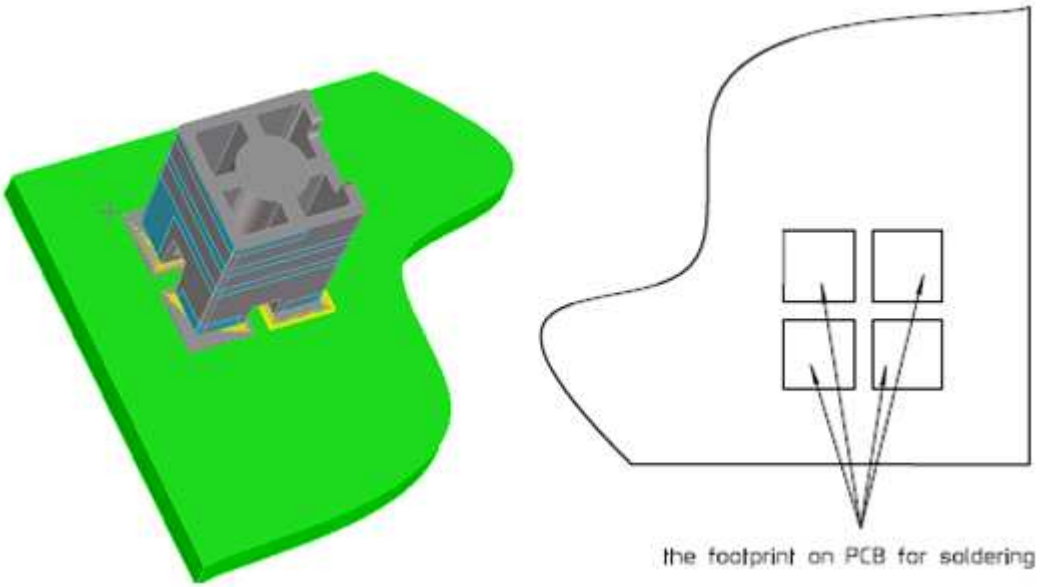

Figure 19

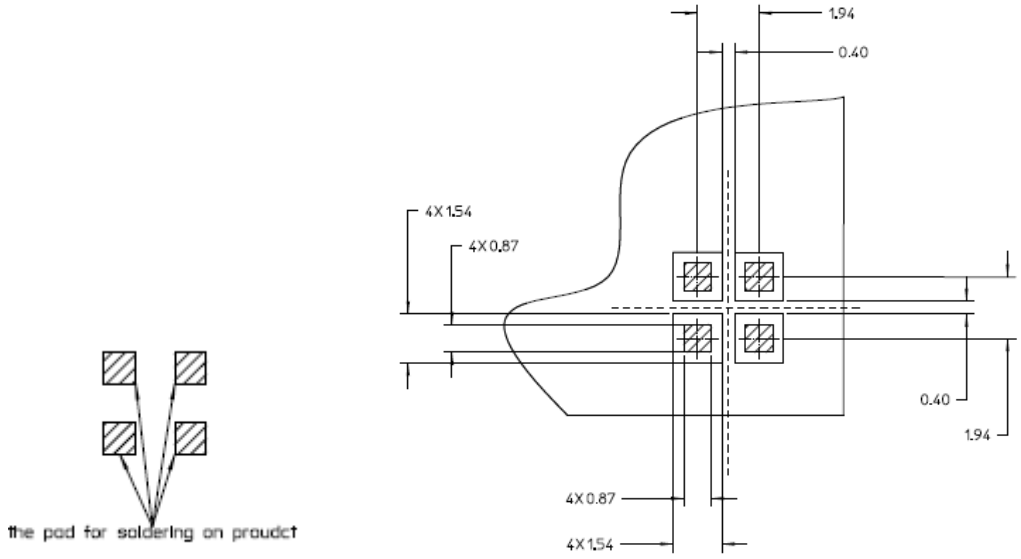

Figure 20: Footprint on PCB for soldering