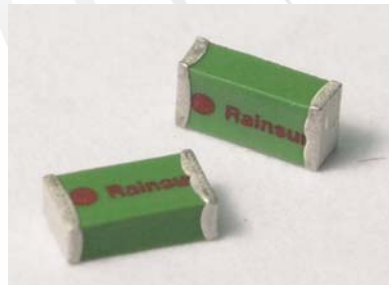


# Chip Antenna application note

## Multilayer Chip Antenna for Wireless Communication



## **1.Introduction**

Rainsun Chip antenna series is high quality RF component for use in wireless communication device. It consists of the necessary antenna elements such as stripe line and high frequency material. It enable customers to quickly and easily develop application for wireless products such as WLAN, Bluetooth, Zigbee and others.

### **1.1 Purpose**

This document describes design guide for chip antenna.

## **2. Test environment**

### **2.1 Instrument**

The chip antenna specification test base on S-Parameter(S11-Return loss). The parameter tested on Vector Network Analyzer (VNA). VNA also provide the smith chart and other relate antenna specification.

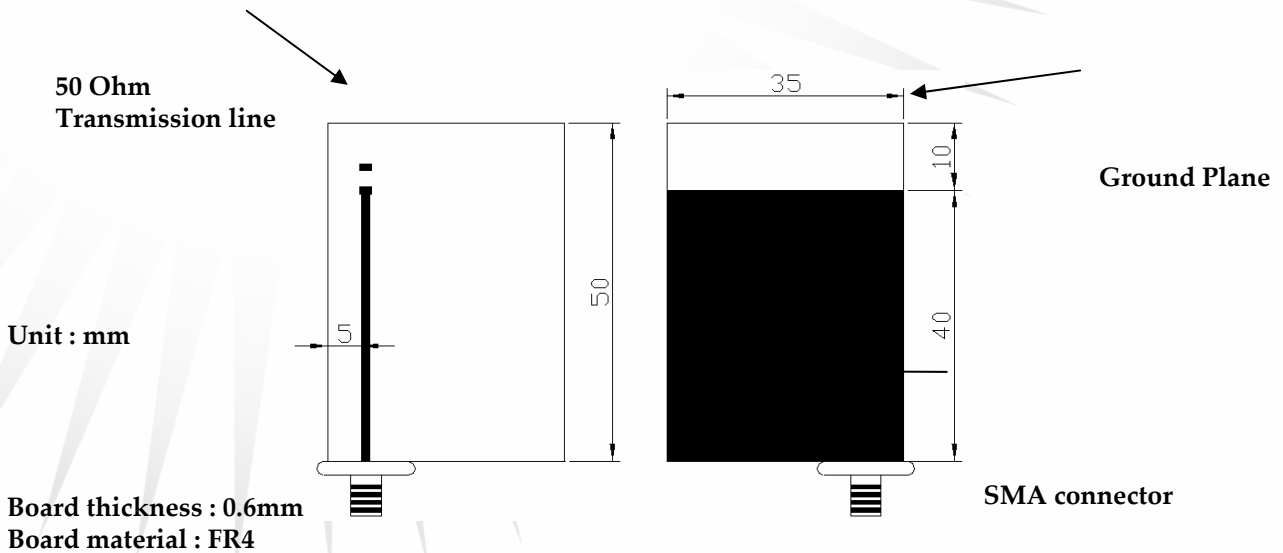
### **2.1 Target Device**

- Standard antenna test board with SMA connector.
- customer product PCB ( contain Rainsun antenna foot printer)

### 3. Antenna Test

#### 3.1 Standard antenna test board

AN3216 antenna test board dimension is 35mmx50mm. The board thickness is 0.6mm.



#### 3.2 Antenna measurement

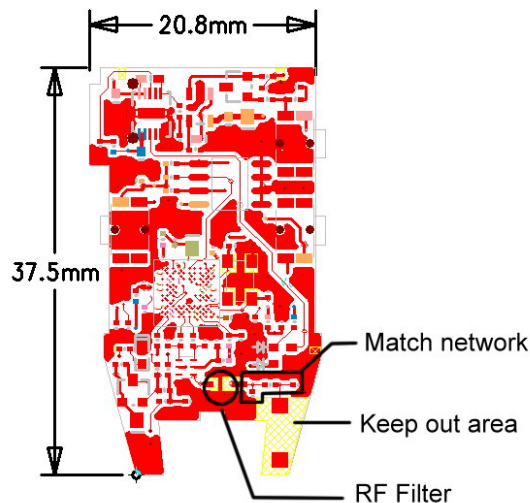
Connect the test board to VNA RF port and measure the S11 parameter. Center frequency depends on target application. For most 2.4GHz, the center frequency usually set to 2.45GHz. The band width need 100Mhz(at least) to cover the 2.4GHz to 2.5GHz working frequency.



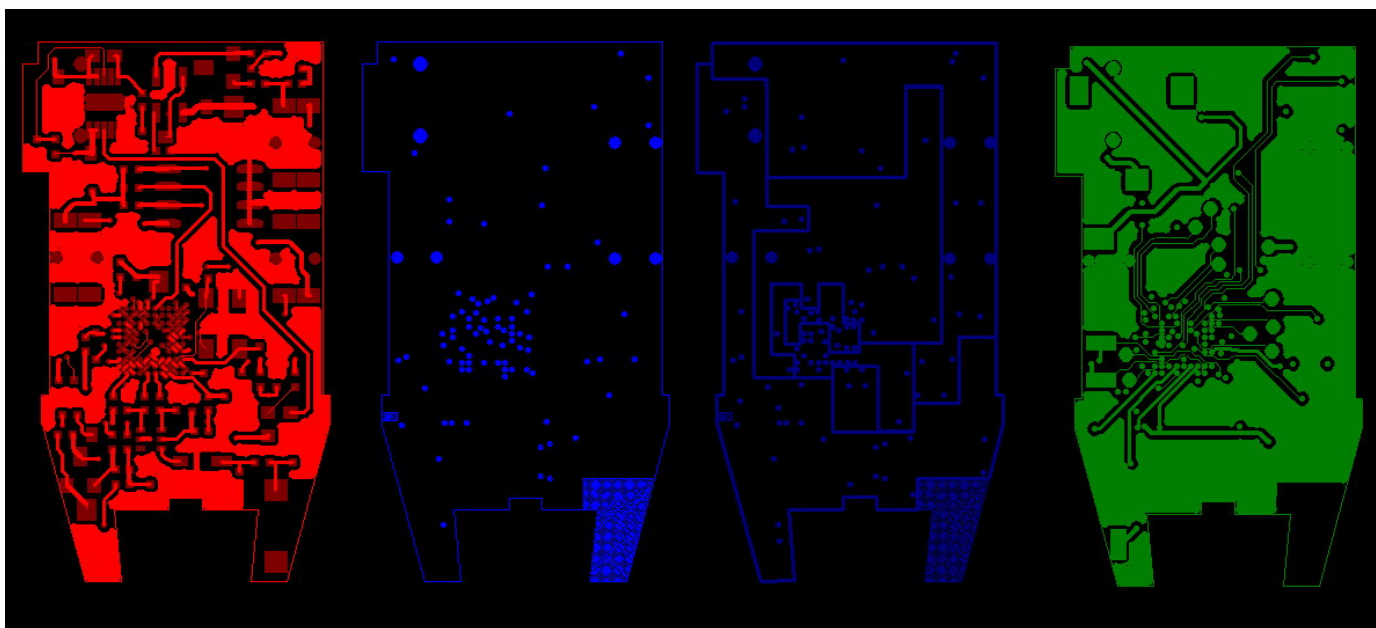
**Testing Instrument: Anritsu 37369C**  
VNA(Vector Network Analyzer)

### 3.3 Customer product PCB

Here is an example design for Bluetooth headset. The board size and antenna placement shows as figure. Between RF filter and antenna, there are 3 components. That is matching circuit. This is very important for antenna performance. The antenna center frequency can be fine tune by match schematic. Different PCB board size may effect antenna center frequency or performance.



In this example, the PCB is 4 layer design. Top layer for component place and routing. 2nd layer is ground plane. 3rd layer is power plane. 4th layer is routing layer. PCB thickness is 0.8mm.



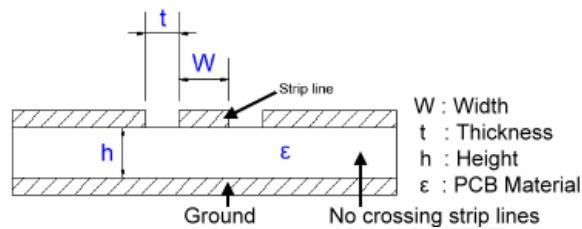
Top layer

GND plane

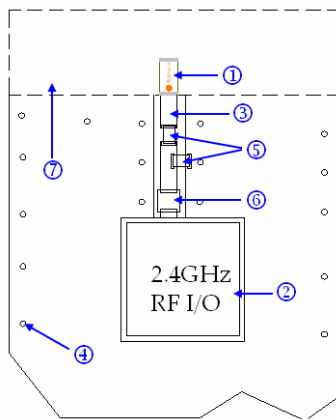
Power plane

Bottom layer

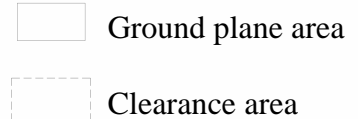
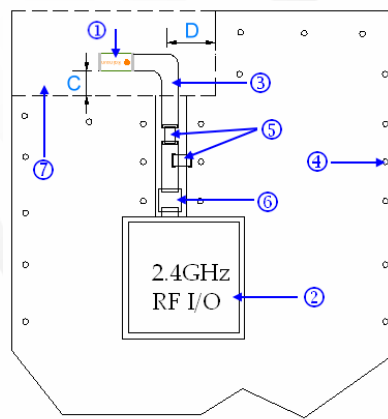
## 4. Application design guide



### Best Choice



### Acceptable



#### 1. Placement of the antenna

The antenna shall be placed on a area without underlying ground plane at the edge of the PCB oriented as above. Ground plane area surrounding the antenna should be with minimum clearance 3mm.

#### 2. Placement of 2.4 GHz module

To avoid losses in the strip line, the module shall be placed as close to the antenna as possible.

#### 3. Strip line

The strip line impedance must be dimensioned according to your specific PCB (see fig.2) to 50 Ohm. No crossing strip lines are allowed between the strip line and its ground plane.

#### 4. Via Connections on PCB

To avoid spurious effects via connections must be made to analogue ground. Via connection depends on PCB layout design. Figure 2 for reference only.

#### 5. Component matching

Component values are depending on antenna placement, PCB dimensions and location of other components. PCB dimension and antenna location will effect the antenna frequency.

#### 6. DC Block

It might be needed depending on RF Module or chip hardware design.

#### 7. Clearance

No components allowed within the clearence area with a minimum distance to other components. The minimum distance is 3mm.