AN9520-D

Multilayer Chip Antenna for

2.4GHz & 5~6GHz Wireless Communication

AN9520-D Multilayer Chip Antenna

◆ Features

- Light weight and low profile 9.55mm(L)X2.1mm(W)X1.0mm(H)
- Omni-directional in azimuth
- Lead (Pb) Free

◆ Applications

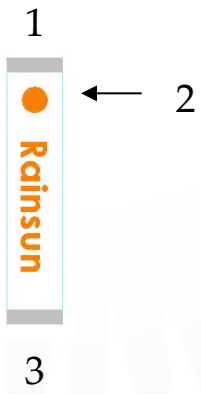
- 2.4GHz & 5~6GHzwireless communications
- 2.4GHz & 5~6GHzModules
- Bluetooth System
- 802.11a/b/g/n Wireless LAN System

Specifications

Center frequency	2.45GHz &5~6GHz
Peak gain	4.0dBi
Operation temperature	-40 ~ +85 ˚C
Storage temperature	-40 ~ +85 ˚C
VSWR	2.0 (max)
Input Impedance	50 Ohm
Power handling	3W (max)
Bandwidth	200MHz
Azimuth beamwidth	Omni-directional
Polarization	Linear

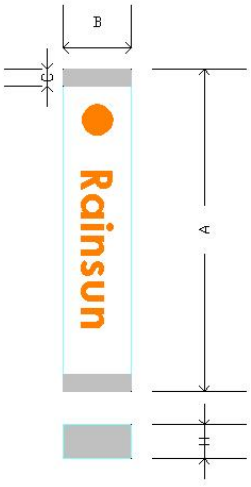
Pin configuration

Top view



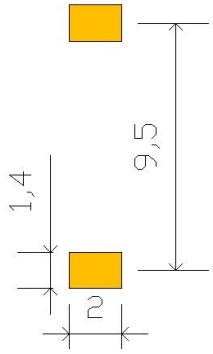
Pin No	Pin assignment
1	Feed termination
2	Feed point mark
3	Solder termination

Dimensions

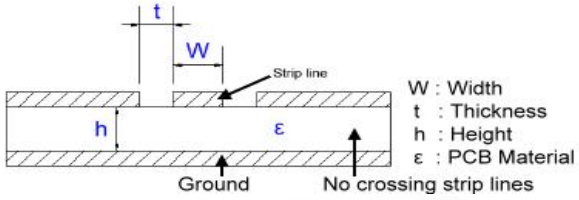


Symbol	Dimensions(mm)
A	9.55 ± 0.20
B	2.10 ± 0.10
C	0.50 ± 0.05
H	1.00 ± 0.20

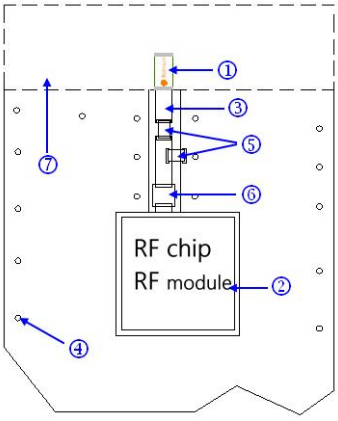
PCB foot printer



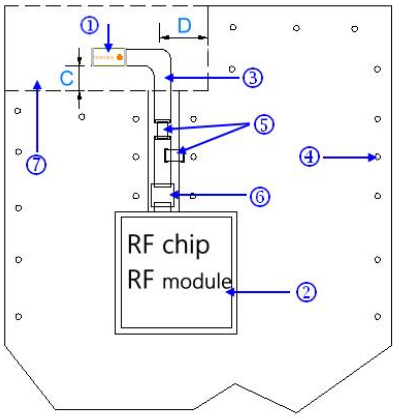
Application design guide



Best Choice



Acceptable

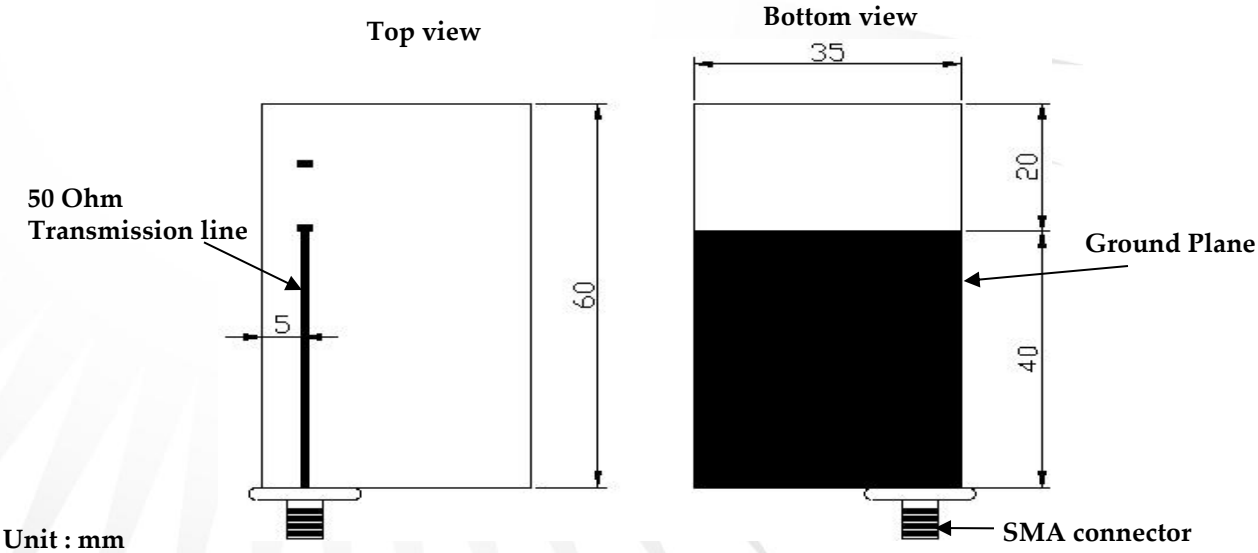


Ground plane area

Clearance area

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 clearence 3mm.
   DO NOT pleace the antenna on the PCB edge with V-cut. The antenna body size is bigger than passive component. Antenna may damage on the V-cut snap off. In PCB panel layout, CNC board edge is better then V-cut edge.
2. **Placement of RF chip set or 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 impendence 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 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.

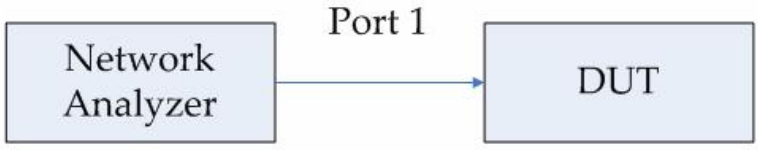
Recommended Test Board Pattern



Unit : mm
Board thickness : 0.6mm
Board material : FR4

Fig-1

Testing Setup



Measurement

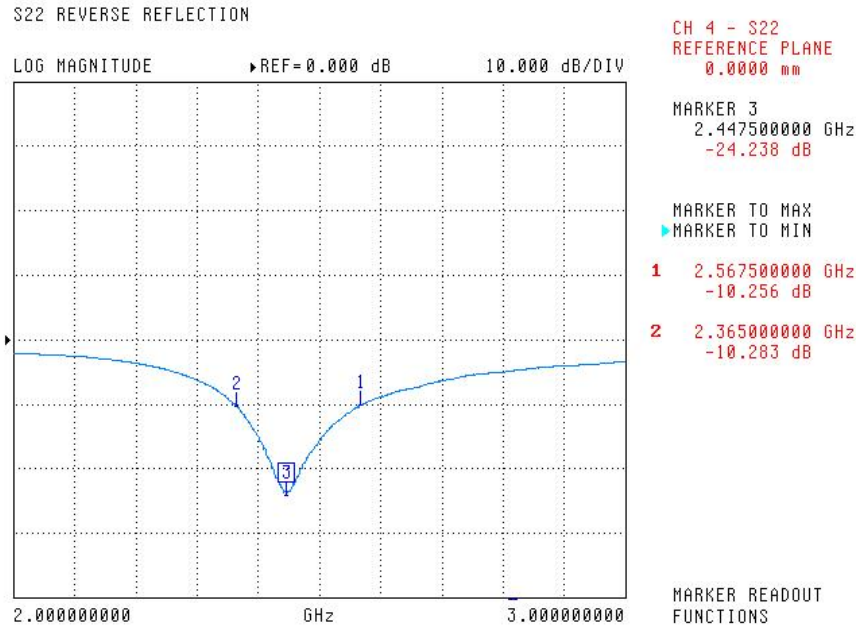
Testing Instrument:
Anritsu 37369C VNA(Vector Network Analyzer)

VNA calibrate with 1 path reflection only calibration sequence on test board feed point.

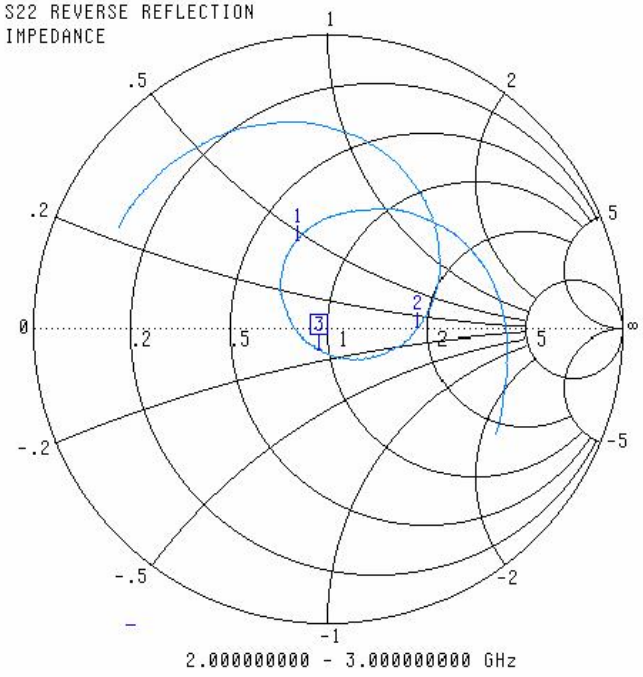
The test board dimension and it's layout is the same as Fig-1.

Typical Electrical Characteristics

Return loss



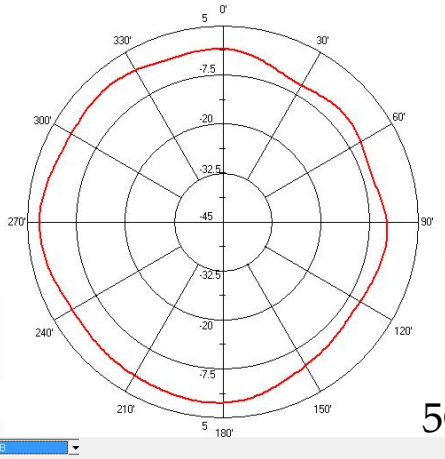
Smith Chart



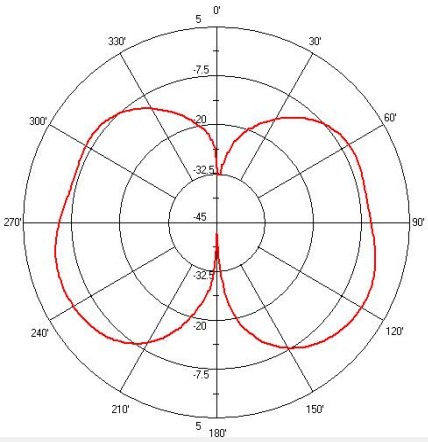
Marker data:
1 : f=2.567 GHz
2 : f=2.365 GHz
3 : f=2.447 GHz

Typical Radiation Patterns

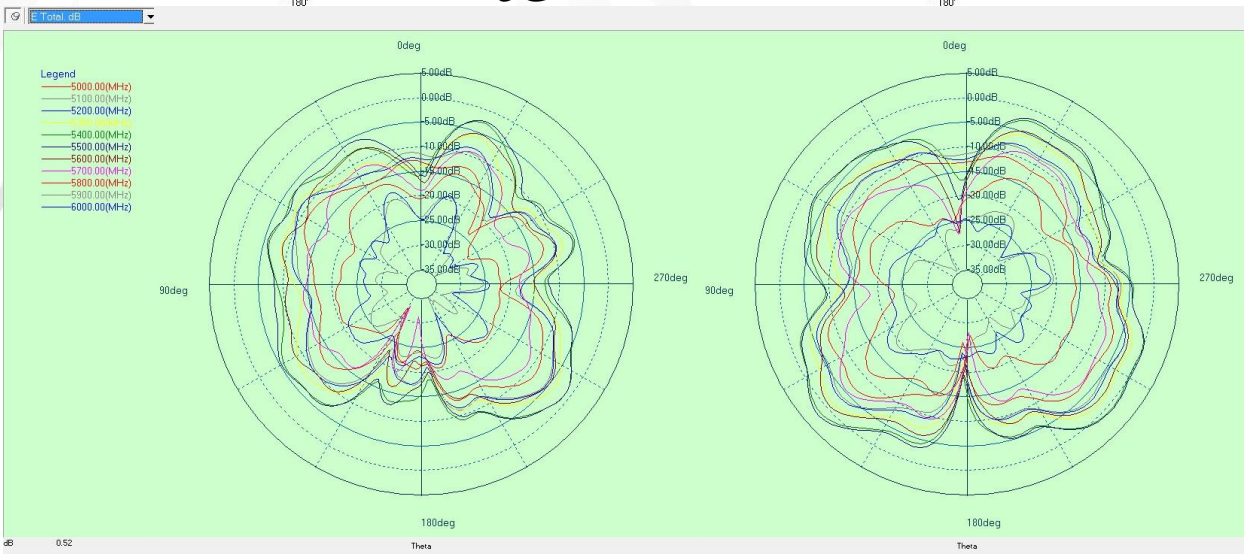
2.45 GHz H-Plane



2.45 GHz E-Plane



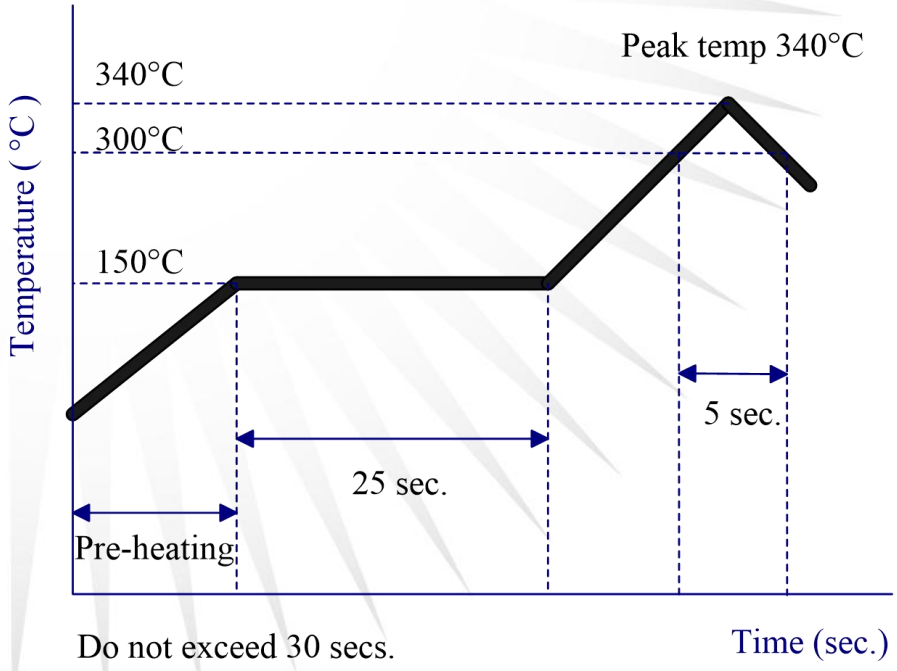
5G



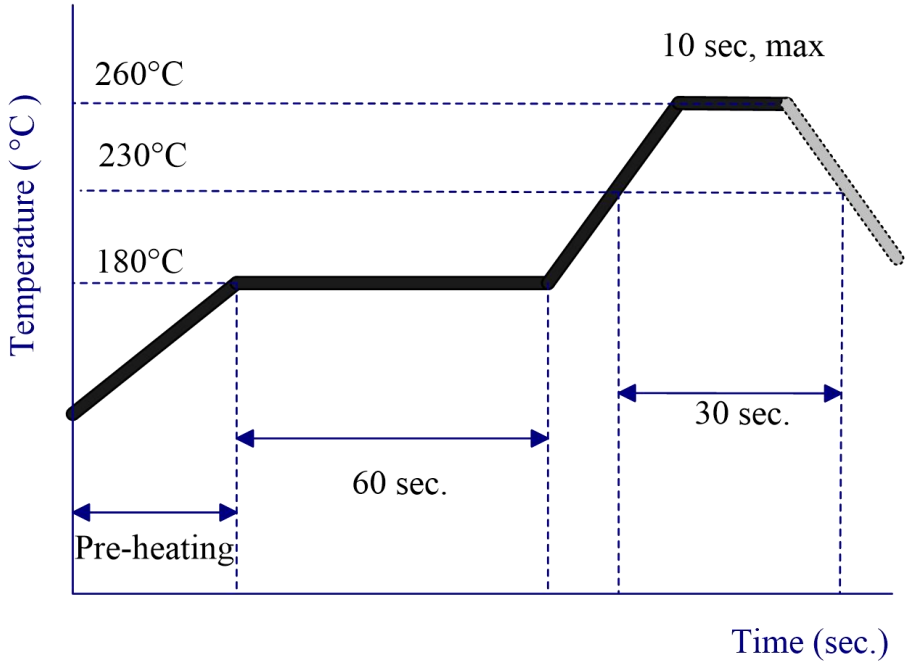
Frequency	Gain (dBi)	Efficiency
2400MHz	3.16	47%
2410MHz	3.08	49%
2420MHz	3.10	50%
2430MHz	3.45	53%
2440MHz	3.99	55%
2450MHz	4.39	56%
2460MHz	4.38	57%
2470MHz	4.21	59%
2480MHz	4.08	62%
2490MHz	4.01	61%
2500MHz	4.12	58%

Frequency	Gain (dBi)	Efficiency( )
5000MHz	-6.23	6%
5100MHz	-3.52	11%
5200MHz	-3.03	12%
5300MHz	-1.21	18%
5400MHz	1.71	33%
5500MHz	2.32	36%
5600MHz	-0.61	18%
5700MHz	-5.29	6%
5800MHz	-11.13	1%

Typical Soldering Profile for Lead-free Process

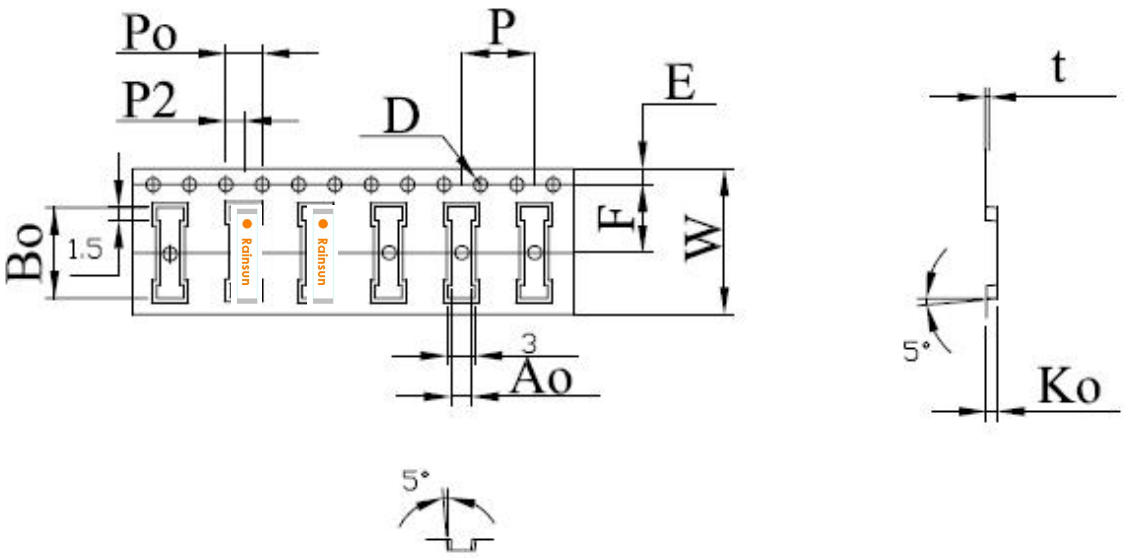


Reflow Soldering



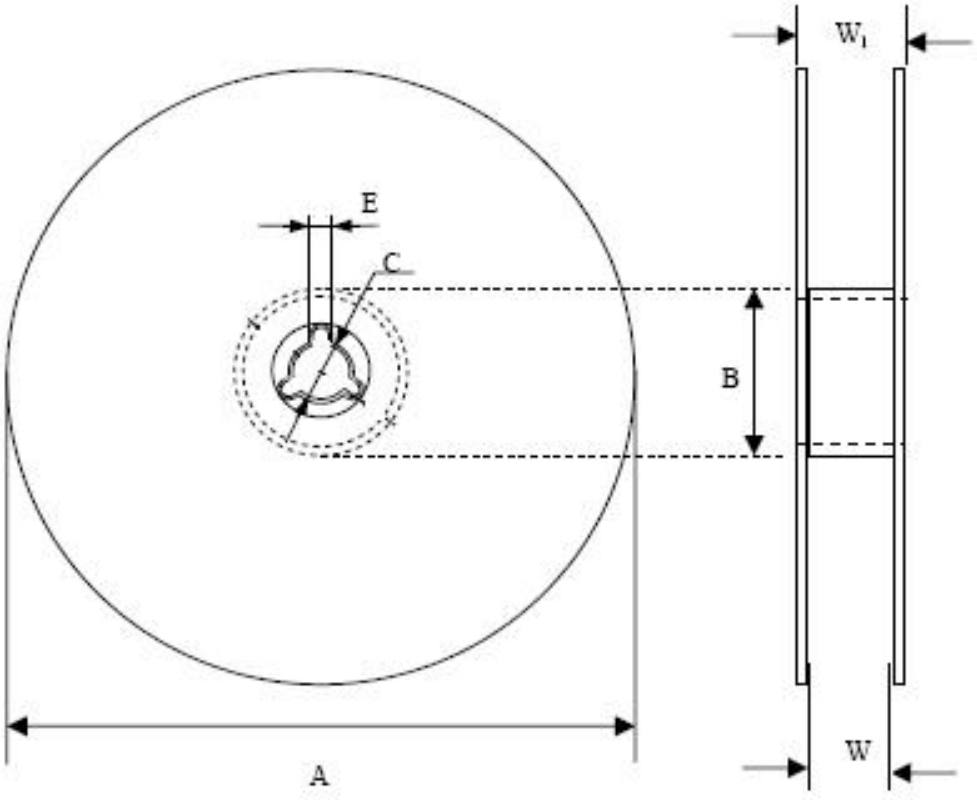
Packing

Blister Tape Specifications



Symbol	Dimension	Tolerance	Unit
W	16.00	± 0.30	mm
E	1.75	± 0.10	mm
F	7.50	± 0.10	mm
D	1.50	+ 0.10 - 0.00	mm
D_1	1.50	+ 0.25 - 0.00	mm
P_0	4.00	± 0.10	mm
P	8.00	± 0.10	mm
P_2	2.00	± 0.10	mm
A_0	2.20	± 0.10	mm
B_0	10.00	± 0.10	mm
K_0	1.20	± 0.10	mm
t	0.30	± 0.05	mm

Reel Specifications



Quantity Per Reel	Tape Width (mm)	A (mm)	C (mm)	B (mm)	E (mm)	W (mm)	W_1 (mm)
1,500	16	180±1	13.0±0.5	62±0.5	2.2±0.5	16±0.5	20±0.2