

# **AN2051-UWB**

## **Multilayer Chip Antenna for 4~9 GHz Wireless Communication**

# AN2051-UWB Multilayer Chip Antenna

## ◆ Features

- Light weight and low profile 5.05mm(L)X2.0mm(W)X1.07mm(H)
- Omni-directional in azimuth
- Lead (Pb) Free

## ◆ Applications

- 4 ~9 GHz wireless communications
- Modules, chipsets

## Specifications

Center frequency	4 ~9 GHz
Peak gain	0.5dBi
Operation temperature	-40 ~ +85 °C
Storage temperature	-40 ~ +85 °C
VSWR	2.5 (max)
Input Impedance	50 Ohm
Power handling	2W (max)
Bandwidth	5000 MHz
Azimuth beamwidth	Omni-directional
Polarization <sup>77</sup>	Linear

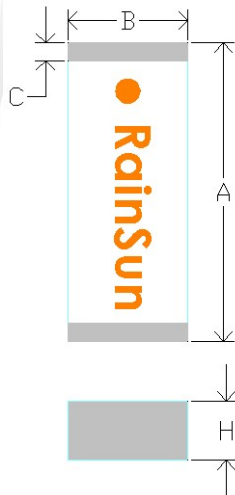
# Pin configuration

Top view



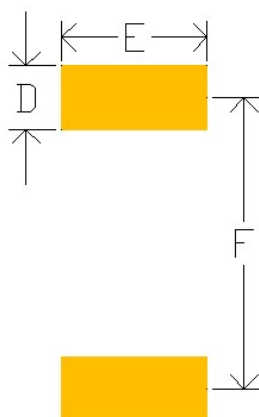
Pin No	Pin assignment
1	Feed termination
2	Feed point mark
3	Solder termination

## Dimensions



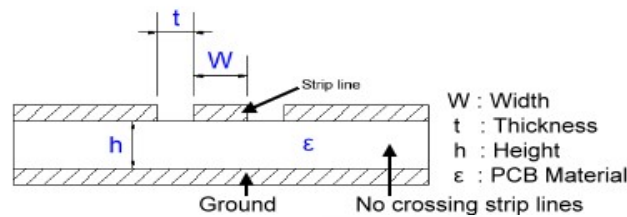
Symbol	Dimensions (mm)
A	$5.05 \pm 0.10$
B	$2.00 \pm 0.10$
C	$0.50 \pm 0.05$
H	$1.07 \pm 0.20$

## PCB Foot Print

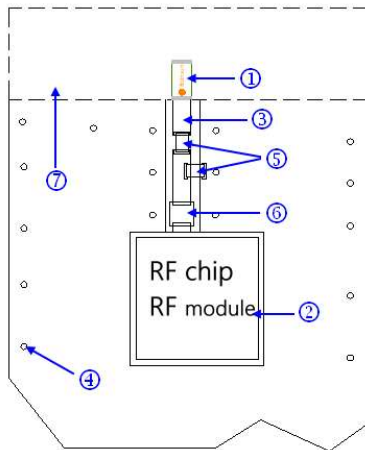


Symbol	Dimensions(mm)
D	1.5
E	2.0
F	5.1

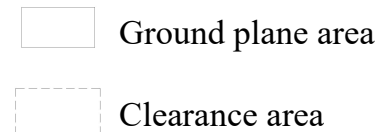
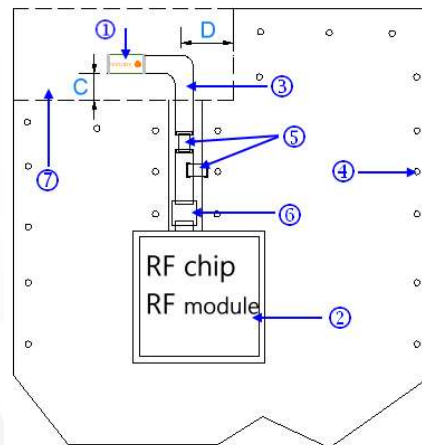
## Application design guide



### 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.

**DO NOT place 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 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 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.

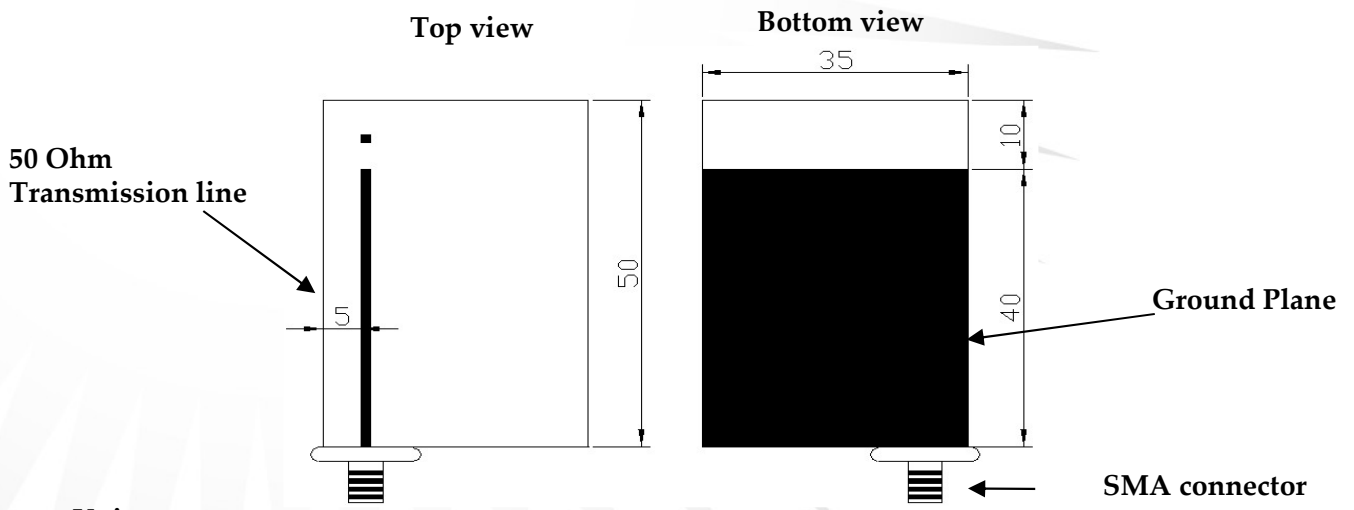
#### 6. DC Block

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

#### 7. Clearance

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

# Recommended Test Board Pattern



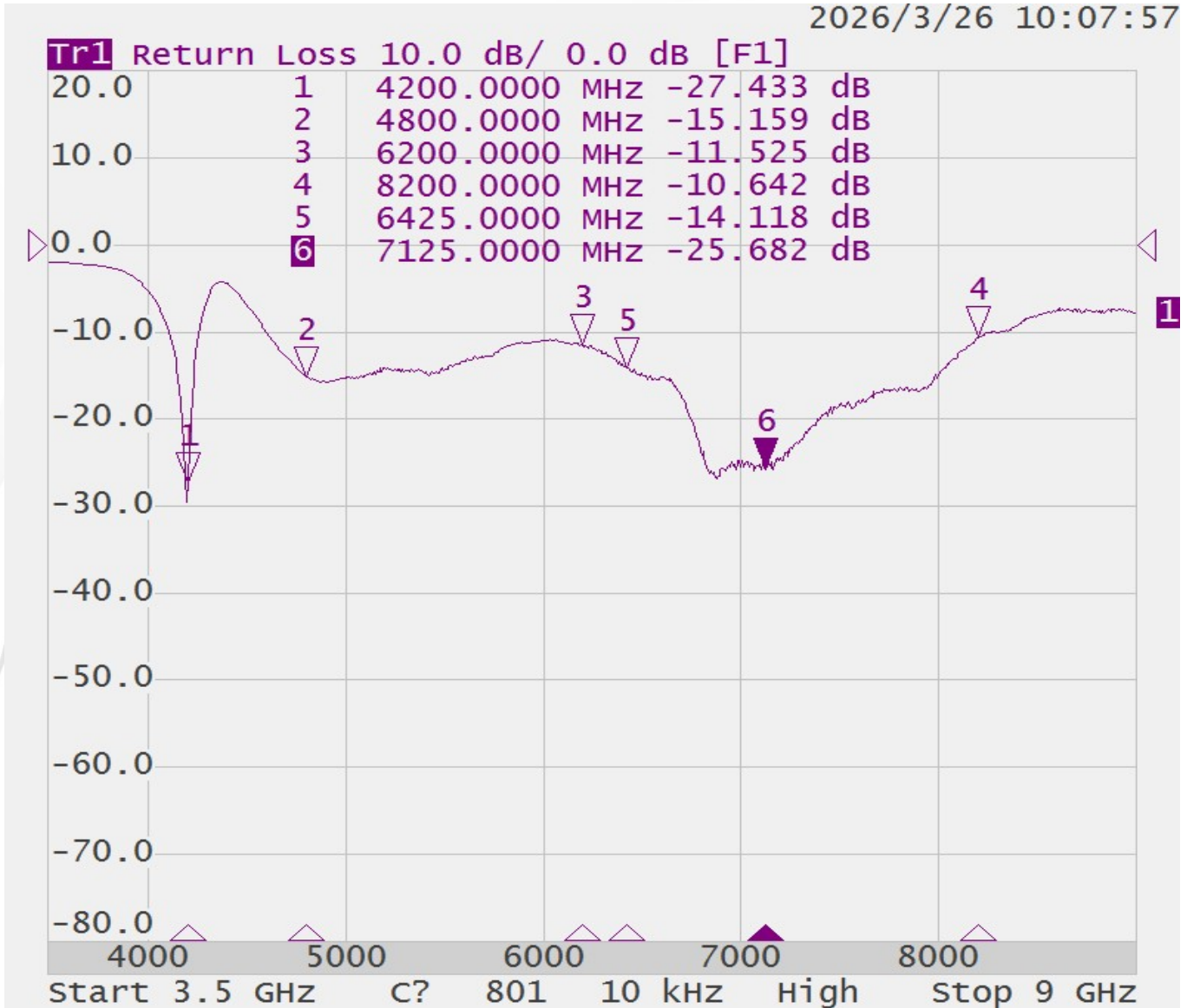
Unit : mm  
 Board thickness : 0.6mm  
 Board material : FR4

**Fig-1**

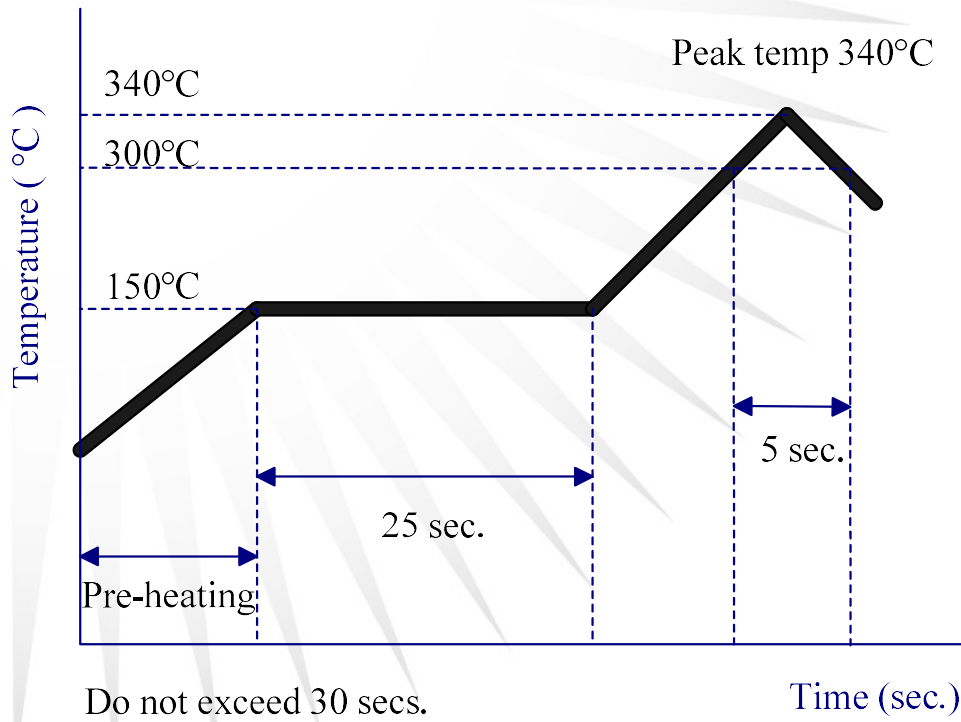
# Typical Electrical Characteristics

## Return loss

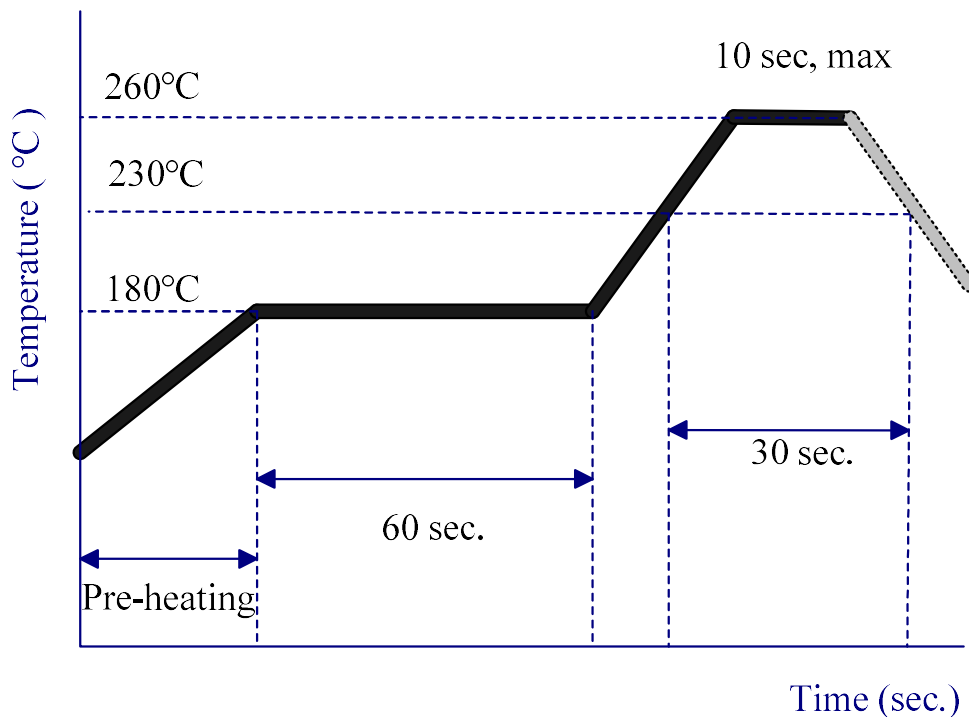
2026/3/26 10:07:57



## Typical Soldering Profile for Lead-free Process

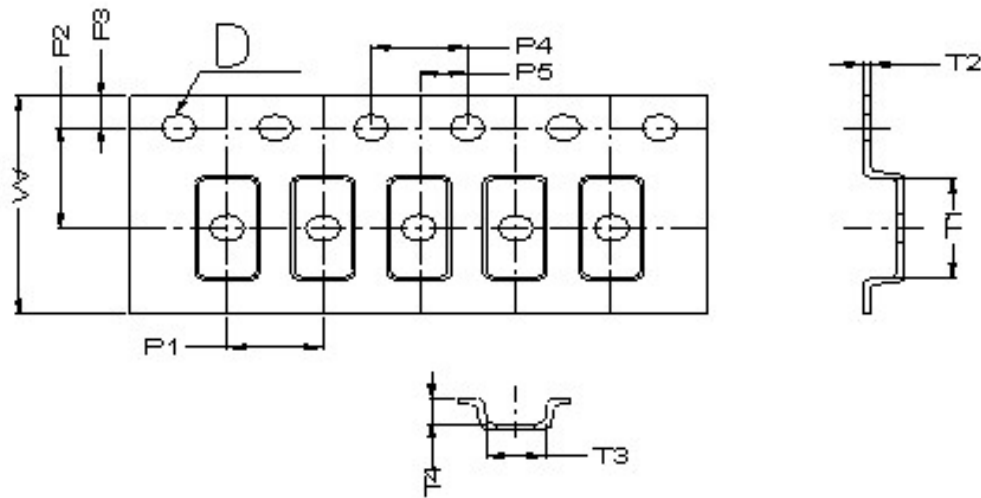


### Reflow Soldering



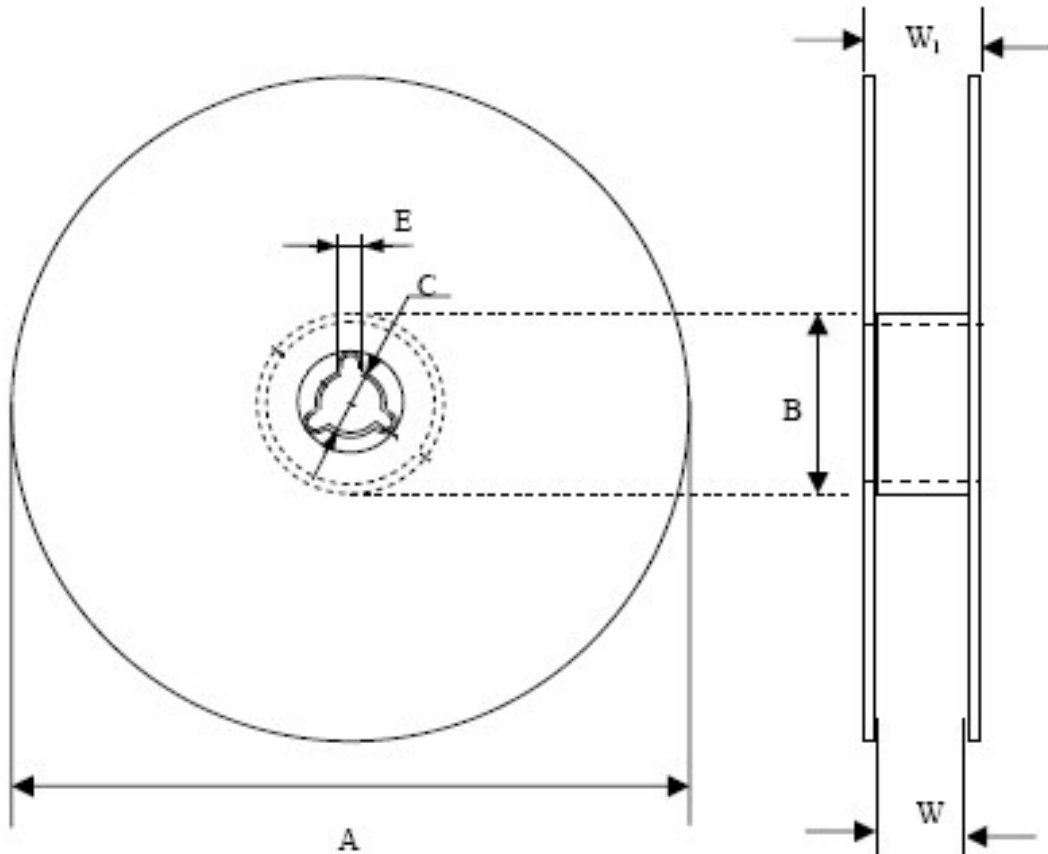
# Packing

## Blister Tape Specifications



Symbol	Dimension	Tolerance	Unit
W	12.00	± 0.30	mm
P1	4.00	± 0.10	mm
P2	5.50	± 0.10	mm
P3	1.75	± 0.10	mm
P4	4.00	± 0.10	mm
P5	2.00	± 0.10	mm
D	1.50	± 0.10	mm
T1	5.40	± 0.10	mm
T2	0.30	± 0.05	mm
T3	2.40	± 0.10	mm
T4	1.40	± 0.10	mm

# Reel Specifications



Quantity Per Reel	Tape Width (mm)	A (mm)	C (mm)	B (mm)	E (mm)	W (mm)	W <sub>1</sub> (mm)
3,000	12	180±1	13.0±0.2	62±0.5	2.2±0.5	12±0.5	16±0.2