

# AN-3216

## Ceramic Chip Antenna for 2.4GHz Wireless Communication



## PF-3216 Chip Antenna

### ◆ Features

- Light weight and low profile 3.2mm(L)X1.6mm(W)X1.3mm(H)
- Omni-directional in azimuth
- Lead (Pb) Free

### ◆ Applications

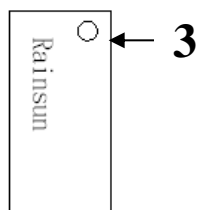
- 2.4GHz wireless communications
- 2.4GHz Modules
- Bluetooth System
- 802.11b/g/n Wireless LAN System

## Specifications

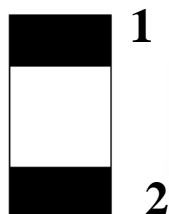
Center frequency	2.45GHz
Peak gain	0.5dBi (typ.)
Operation temperature	-40 ~ +85 °C
Storage temperature	-40 ~ +85 °C
VSWR	2.5 (Max)
Input Impedance	50 Ohm
Power handling	1W (Max)
Bandwidth	100MHz (typ.)
Azimuth beamwidth	Omni-directional
Polarization	Linear
Soldering pad	Natural tin

# Pin configuration

Top view

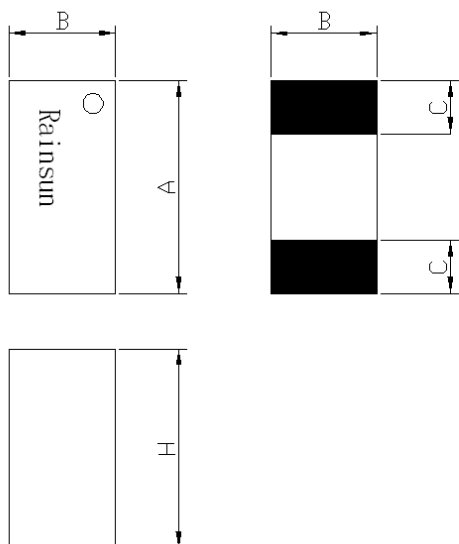


bottom view



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

## Dimensions

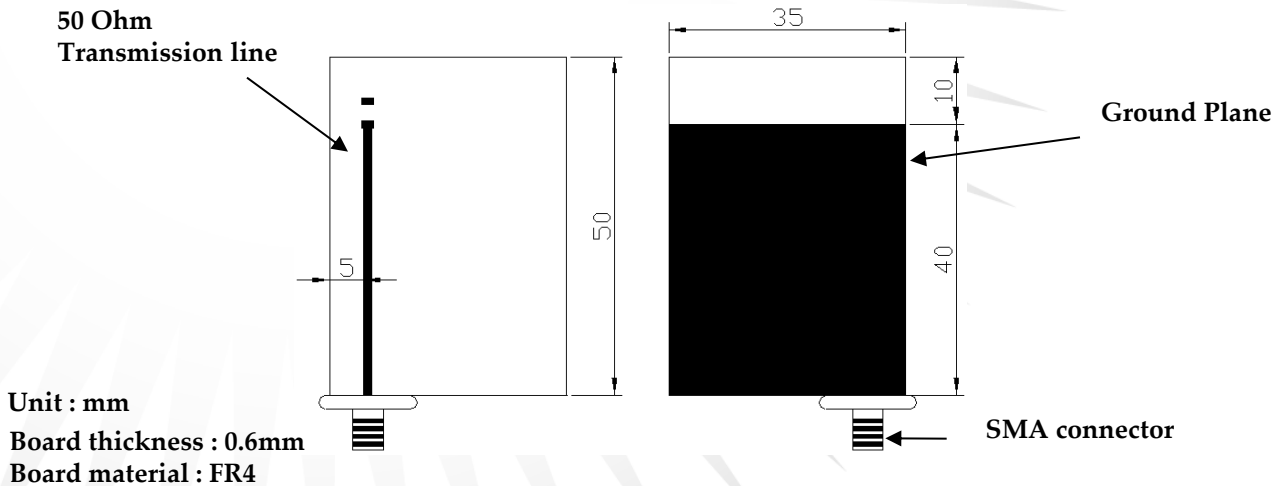


Symbol	Dimensions(mm)
A	$3.2 \pm 0.2$
B	$1.6 \pm 0.2$
C	$0.8 \pm 0.2$
D	$0.4 \pm 0.2$
H	$1.3 \pm 0.2$

## Recommended Test Board Pattern

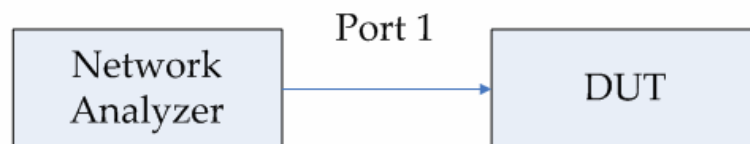
Top view

Bottom view



**Fig-1**

## Testing Setup



## Measurement



### Testing Instrument:

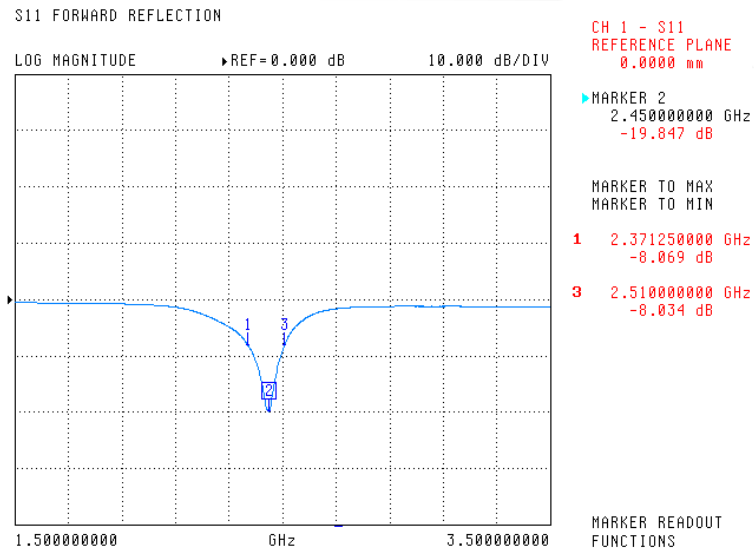
Anritsu 37369C VNA(Vector Network Analyzer)

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

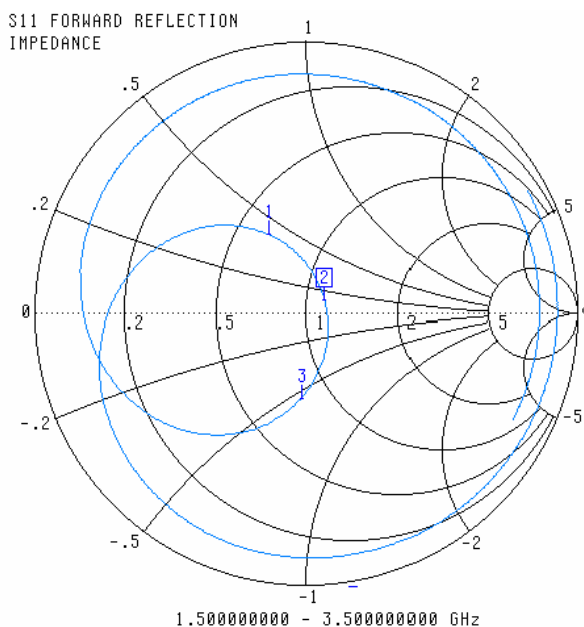
The test board dimension and it's layout is the same as Fig-1.

# Typical Electrical Characteristics

## Return loss



## Smith Chart

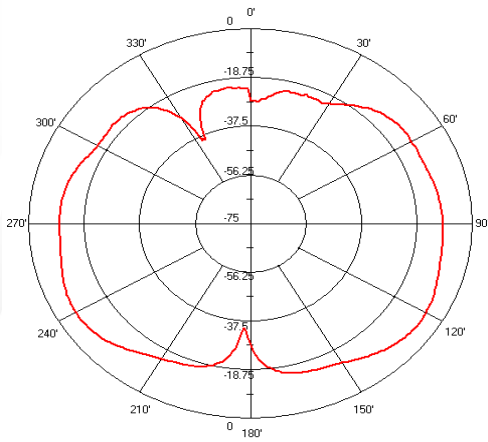


**Marker data:**

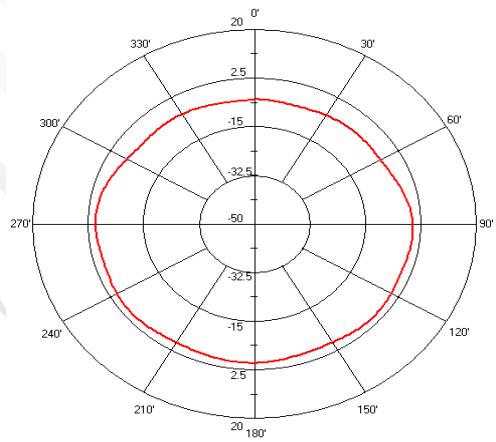
- 1 : f=2.411 GHz
- 2 : f=2.450 GHz
- 3 : f=2.492 GHz

# Typical Radiation Patterns

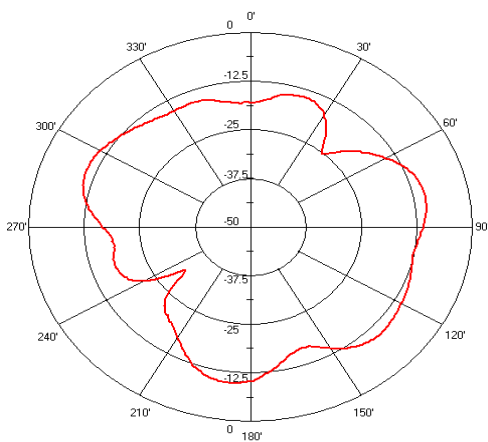
**2.45 GHz H-Plane  
vertical**



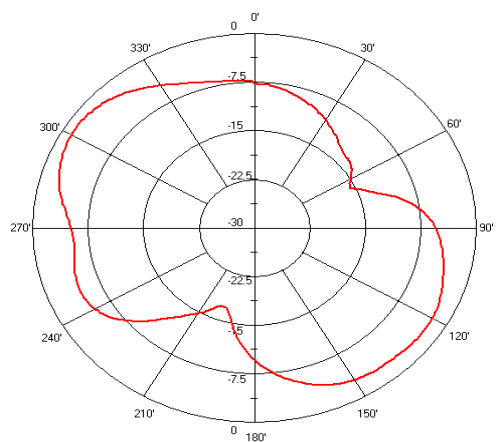
**2.45 GHz H-Plane  
horizontal**



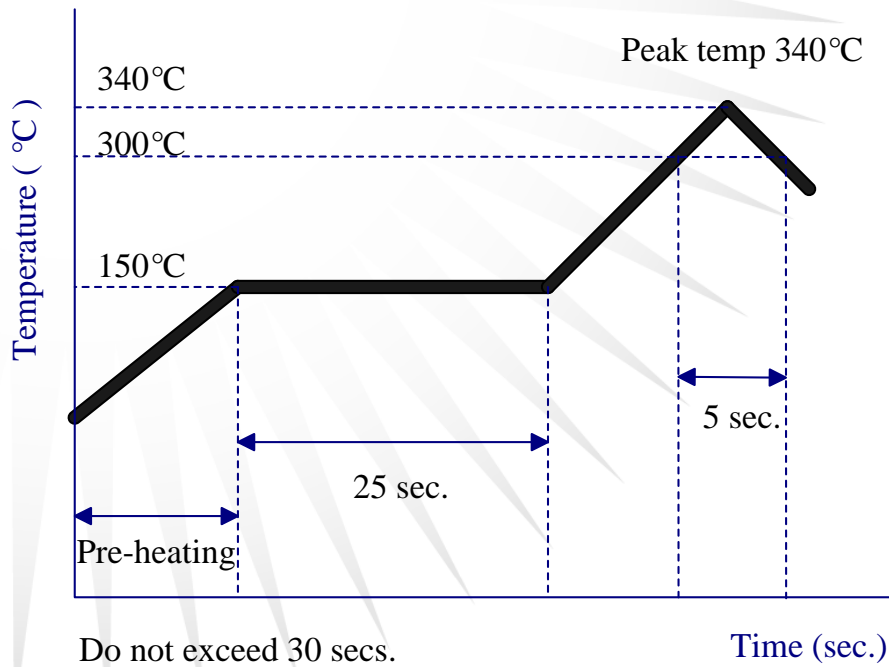
**2.45 GHz E-Plane  
vertical**



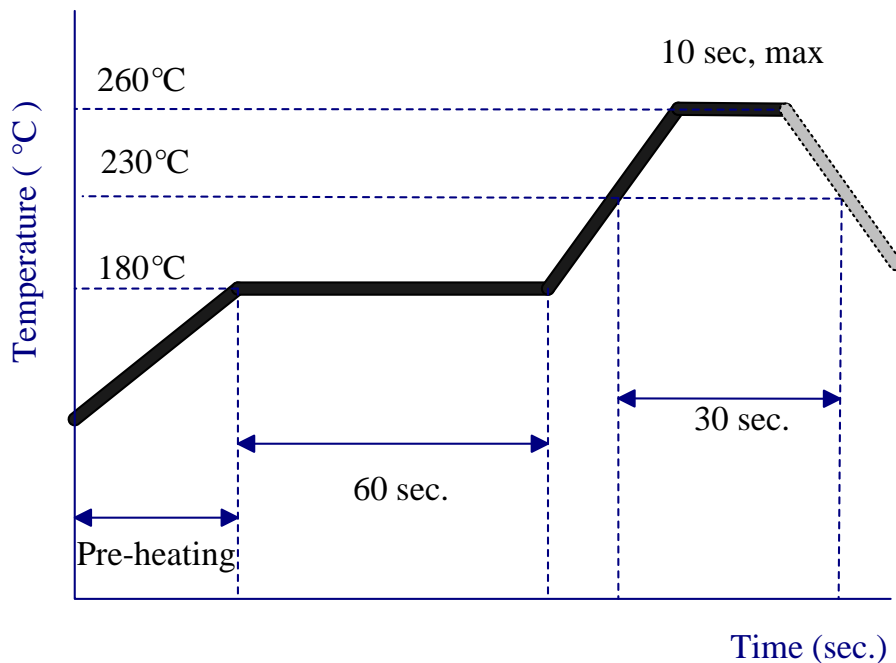
**2.45 GHz E-Plane  
horizontal**



## Typical Soldering Profile for Lead-free Process



### Reflow Soldering



## Application example

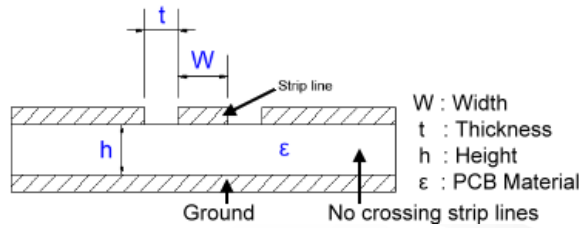
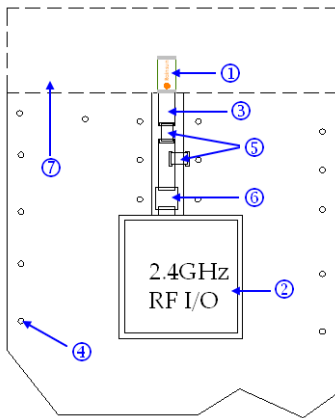
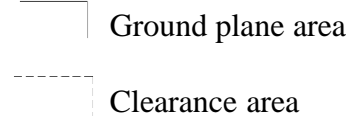
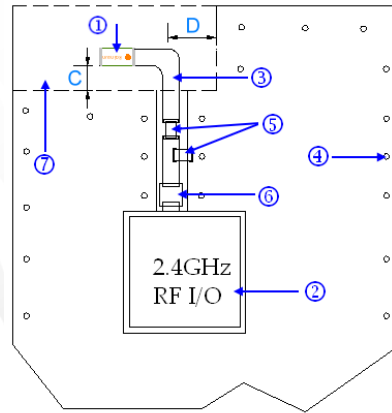


Fig.2

### 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.