Applicant:Shenzhen Renwei Digital Technology Co., Ltd

Address:Room 403, Building B3, hongwan Maker Center, Xixiang Street,Shenzhen City, Guangdong Province,China

Model:KS05

※ Product Features:

- The whole is surface mounting device

It has the characteristics of small volume, low shape and light weight

- Wide frequency band
- Be certified lead-free
- The dimensions are :3.2x1.6x0.4mm

※ Application:

- Bluetooth/WLAN/home RF technology
- ISM band 2.4GHz applications

Technical indicators:

Center frequency	2.50 GHz
Band width	100MHz(typ.)
Maximum gain	2.71dBi(typ.) (XZ-V)
Average gain	0.5dBi(typ.) (XZ-V)
Voltage standing wave ratio	< 2
Impedance	50 Ω
Power capacity	3 W(max)
Operating temperature	-40 ~ +85 ℃
Storage temperature	-40 ~ +85 ℃

Part number information:

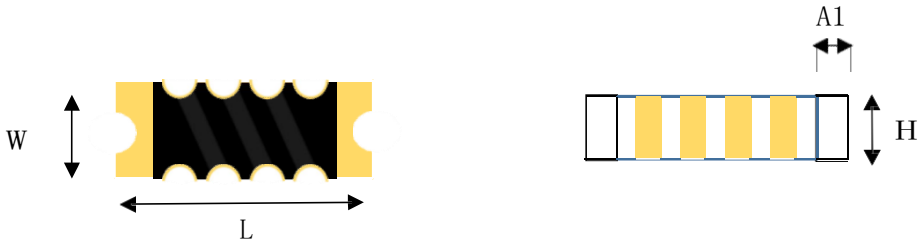
A 3216 H 2G50 M200 - 03

A	Antenna	2G50	The center frequency is 2.5G
3216	Dimensions 3.2x1.6	M200	Band width 200M
H	electrosensibility	03	Product model number

Description of structure:

Pin No.	1	2
Pin allocation	Feed end	Fixed end
Note: One of these two pads is connected to the RF signal and the other is used to hold without receiving any signal. The left and right sides of these two pads are perfectly symmetrical and do not distinguish between positive and negative and left and right.		

Size description:



Units (mm)	L	W	H	A1
Overall dimensions	3.2 + / - 0.05	1.6 ± 0.05	0.4 ± 0.05	0.35 ± 0.05

Antenna placement and clearance processing and matching network diagram

Matching device values	Parallel device Shunt 1	1.7 nH
	Series Device Series 1	2.4 nH
	Shunt device 2	NC

Impedance matching is required for any type of antenna to ensure that the antenna performance meets the requirements of the impedance specification,

The A3216H2G50M200-03 patch antenna also needs to add a matching network to ensure the antenna performance meets the standards.

Three matching devices together form π-type matching network for the impedance matching of A3216H2G50M200-03 patch antenna.

The specific device values of these matching devices need to be obtained after antenna impedance matching and debugging. The device values shown above are the values of our test circuit board and can be used as reference values. If you do not have high requirements for antenna performance, you can also directly use the above reference values.

The line width should match the 50 ohm characteristic impedance according to the PCB material and thickness design.

It is recommended to place the antenna on the edge or corner of the board, do not place the antenna in the middle of the board and do not allow the antenna to be surrounded by conductors.

Gaps are needed near the antenna area of the board. As shown above, the blank area on the board (the white area) is the gap area of the antenna. The so-called

clearance area refers to the area that cannot be paved and wired except for the antenna pad and antenna signal wiring. The clearance treatment of this area should be for all layers of the PCB board, not just the surface layer.

The headroom area of the antenna should be as large as possible, and the antenna should be placed as close as possible to the edge of the board, so that the antenna body is far away from the board, and the larger the headroom means the better the antenna efficiency and gain performance.

In the structure of the whole machine, it is recommended that there should be no conductor in the overhead area above or below the direction of overlooking the PCB, otherwise the antenna performance will be affected.

The non-clearance area needs to be paved, and the ground should be increased as much as possible through the hole connection between the layers.

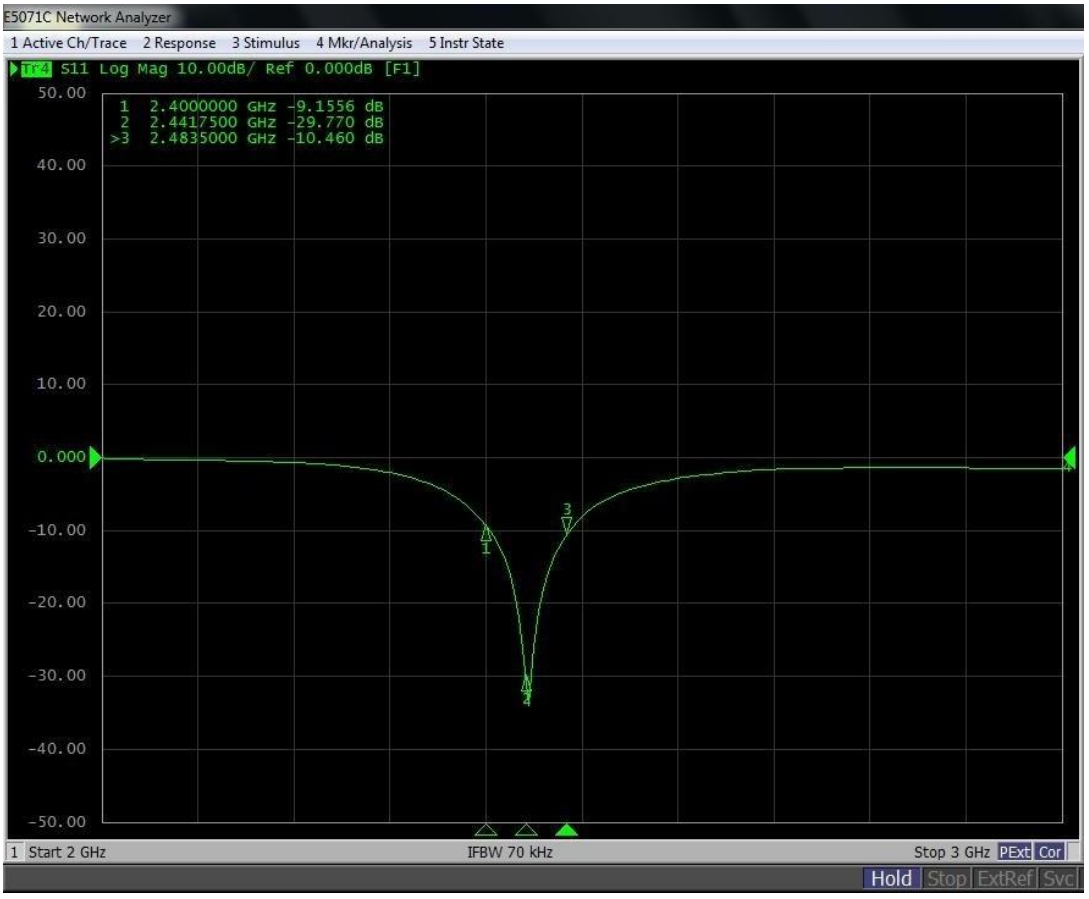
The equipment used for antenna matching debugging is the network analyzer. If you are an antenna professional technician, you can perform antenna matching debugging with the network analyzer that comes with you. If you do not have the relevant technology, please contact us and we can provide it for your product

Professional antenna impedance debugging services. Antenna matching debugging needs to provide the entire product (no need to open it).

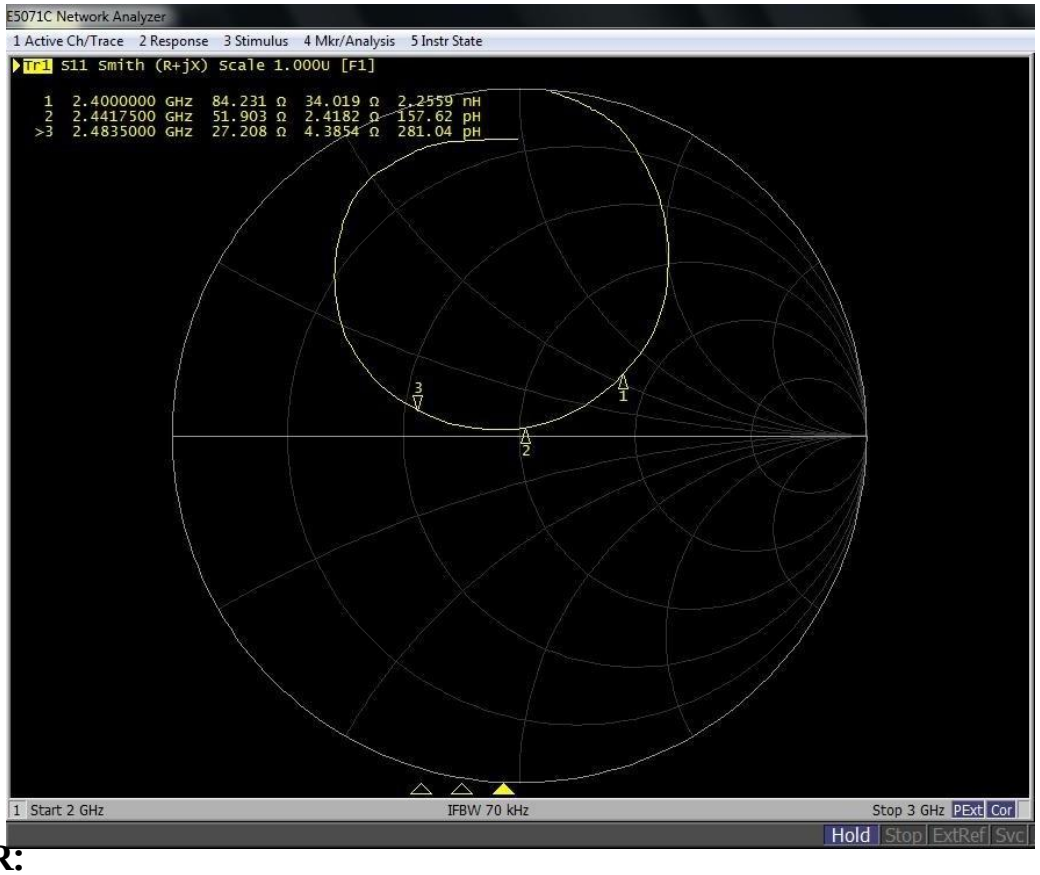
Note that antenna impedance matching debugging mentioned here has nothing to do with RF line impedance control. Rf line impedance control is only for RF line wiring in paved areas, where the matched antenna impedance is the antenna. Please do not confuse the two.

2. The performance parameters of antenna matching after debugging are shown in the figure below:

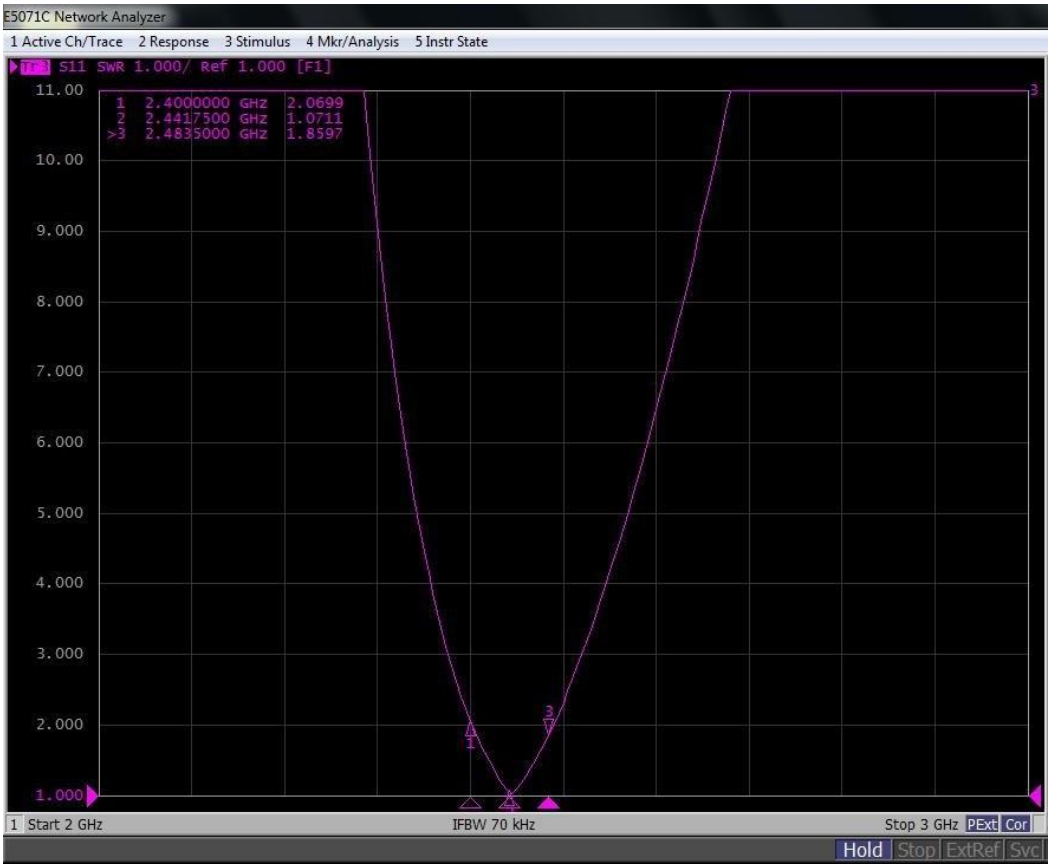
S11 Log Mag:

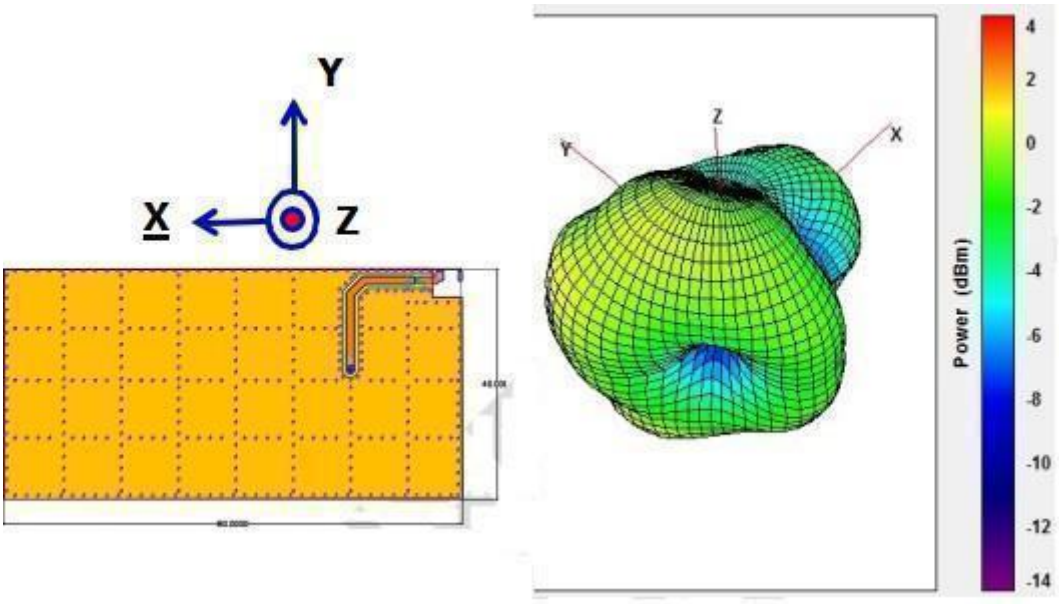


S11 Smith R+jx:



S11SWR:





	Efficiency	Peak Gain	Directivity
2400MHz	55.21%	1.45 dBi	5.32 dBi
2450MHz	66.45%	2.71 dBi	5.21 dBi
2500MHz	57.53%	1.98 dBi	5.29 dBi

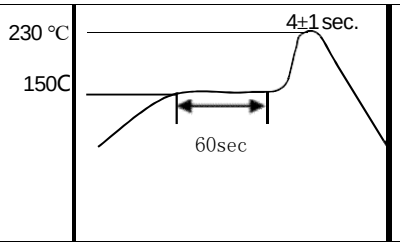
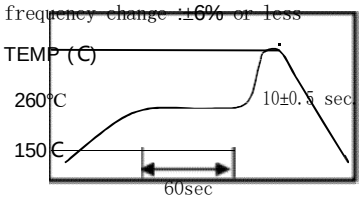
3, reliability and test conditions

<table>
<tr><th>Items</th><th>requirement</th><th>TEST CONDITION</th></tr>
<tr><td rowspan="2">Weldability test</td><td>1.Wettability should exceed 90% coverage
2. No visible mechanical damage temperature</td><td rowspan="2">Heating temperature :150 c / 60 seconds.
Welding temperature :230±5 c

Duration :4±1

SEC. Welding :Sn-

Ag3.0-Cu0.5

Lead-free flux: rosin</td></tr>
<tr><td> 
</td></tr>
<tr><td>Heat resistance of solder</td><td>1. No obvious mechanical damage
2. Center frequency change :±6% or less
 
</td><td>Heating temperature :150 c / 60 SEC.
Welding temperature :260±5 c

Duration :10±0.5

seconds.

Welding :Sn-Ag3.0-

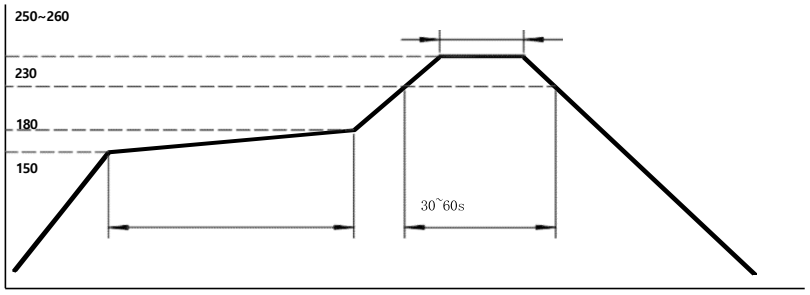
Cu0.5

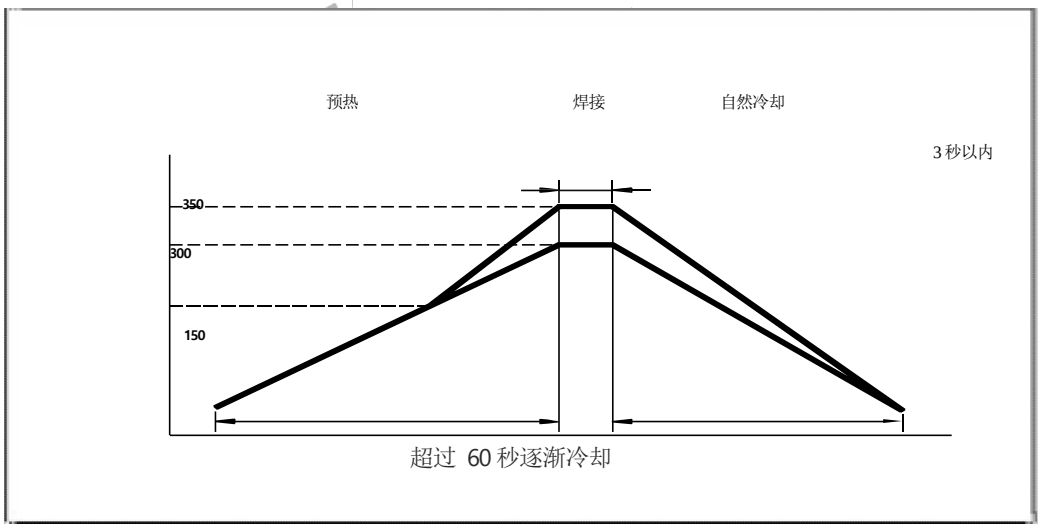
Lead-free flux: rosin</td></tr>
<tr><td>Component adhesion (Push-push test)</td><td>1. No obvious mechanical damage</td><td>The unit should be reflow soldered (230±5C for 10 seconds) to the tinned copper substrate, applying a dynamometer to the side of the element. The device must have -st-f 0.5 kg without terminating the connection to the component o</td></tr>
<tr><td>Component adhesion (tensile test)</td><td>1. No obvious mechanical damage</td><td>Insert 10cm of wire into the remaining open eye bend, wrapping the ends of the wire evenly upward together.
There should be no visible damage to the end.</td></tr>
<tr><td>Heat Shock</td><td>1. No obvious mechanical damage
2. Center frequency change: within ±6%
<table><tr><th>Phases</th><th>Temperature (C)</th><th>Time (min)</th></tr><tr><td>1</td><td>+110 ± 5C</td><td>30 + 3</td></tr><tr><td>2</td><td>Room temperature</td><td>In 3 seconds</td></tr><tr><td>3</td><td>-40 ± 2C</td><td>30 + 3</td></tr><tr><td>4</td><td>Room temperature</td><td>In 3 seconds</td></tr></table></td><td>+ 110 c = > 30±3 minutes
-40 c = > 30±3 minutes
Test cycle :10 cycles
The chip is stable for 2 to 3 hours under normal conditions before being measured</td></tr>
<tr><td>High temperature resistance</td><td>1. No obvious mechanical damage
2. Center frequency change :±6% or less
3. No disconnection or short circuit</td><td>Temperature :+110±5C Duration :1000±12h measurement
The front chip should be stable in normal condition for 2 to 3 hours.</td></tr>
</table>

Low temperature resistance	1. No obvious mechanical damage 2. Center frequency change :±6% or less 3. No disconnection or short circuit	Temperature :-40±5 c Duration :1000±12 hours The chip is normally stable for 2-3 hours before being measured.
Moisture test	1. No obvious mechanical damage 2. Center frequency change :±6% or less 3.No disconnection or short circuit	Temperature :40±2 c Humidity :90% to 95% RH Duration :1000±12h Before measuring, the chip should be stable in normal state for 2 to 3 hours.

4. Mounting and welding process

Lightly activated rosin flux is preferred. Due to the coefficient of expansion between solder, chip and substrate do not





The recommended temperature curve for reflow soldering is shown in Figure 1.

The use of soldering irons to join products is discouraged due to inherent process control limitations. If a soldering iron must be used, the following precautions are recommended.
· Preheat the circuit and product to 150 °C
· Never touch the ceramic with the iron tip. Use a 20 watt soldering iron with a diameter of 1.0 mm
· Tip temperature (Max.)280℃
· Tip diameter (Max. 1.0mm) limits welding time to 3 seconds.

5. A3216H2G50M200-03 ceramic chip antenna user reminder

1. Chip antennas are made of ceramic materials, which are harder and more fragile than printed circuit board materials. A bend in the circuit board where the chip antenna is located can cause the solder joints or the antenna itself to crack.

2. The antenna should be placed in the corner of the PCB with sufficient clearance from other circuits, and no components, planes, mounting screws, or wiring should be placed within the antenna exclusion zone of each layer, the actual forbidden area depends on the antenna being used. 3 Ceramic antenna as a built-in antenna, should try to avoid the impact of the circuit board metal and shell, so direct use often appear performance problems, can not be used directly, must be adjusted for their own products.

4. When ultrasonic welding is carried out near the chip antenna location, caution should be exercised. Strong ultrasonic vibrations may cause the chip antenna solder to crack.

5. The above data is measured on the reference PCB(ground) shown in this specification. When the antenna position or size of the PCB changes, the

values for antenna performance and matching elements may differ from the data shown here.

6. The information provided in this reference is believed to be correct as of the date of publication. Guangdong Dongyou Technology Co., Ltd. reserves the right to modify the reference specifications without prior notice due to technical improvements and other reasons. Please consult the company's engineering team for the latest information before using this product. Depending on the customer's requirements, we can advise and assist with the installation of this antenna on the customer's equipment by conducting simulated or actual measurements of the equipment of interest in our test facilities.