APPLICATION NOTES
AN-WF-0146-20211012

WLAN/BT/Zigbee/Wi-Fi/Embedded Stamp Metal Antenna 1000146

2.4/4.9/5.2/5.8 GHz (802.11 a/b/g/n/c + Japan)

Applications:

Embedded Design	Gateway	M2M	Smart Grid
Handheld	Access Point	Healthcare	OBD-II
Wireless Headsets	Telematics	Industrial	Media Players
Tablets	Tracking	Devices	Bluetooth

WLAN/BT/Zigbee/Wi-Fi/Embedded Stamp Metal Antenna 1000146

Table of Contents

WLAN/BT/Zigbee/Wi-Fi/Embedded Stamp Metal Antenna 1000146

2.4 / 4.9 / 5.2 / 5.8 GHz (802.11 a/b/g/n/c + Japan)

Purpose

This document provides information for incorporating KYOCERA AVX's Prestta standard WLAN / BT / ZigBee stamp metal embedded SMT antenna into wireless products. Specifications, design recommendations, board layout, packaging, and manufacturing recommendations are included.

This document is divided into two parts: a main section and appendices. The main section addresses points and issues common to all products. The appendices provide product-specific information.

Overview

The Prestta Product Line

The Prestta series of standard WLAN embedded antennas represents a new category of standard, internal antennas. KYOCERA AVX's antennas utilize proprietary and patented Isolated Magnetic Dipole (IMD) technology to meet the needs of device designers for higher functionality and performance in smaller/ thinner designs.

IMD Technology Advantages

Real-World Performance and Implementation

KYOCERA AVX continues to set the standard for antenna performance with its award-winning IMD technology, which uses patented design configurations to confine the current flow to the antenna element rather than exciting the main circuit board. Other antennas may contain simple PIFA or monopole designs that interact with their surroundings, complicating layout or changing performance with user position. KYOCERA AVX's antennas utilize patented IMD technology to deliver a unique size and performance combination.

IMD technology offers important real-world advantages over other approaches. Please see our white paper and Website www.kyocera-avx.com for a full explanation.

IMD Features, Advantages and Benefits Summary

Feature	Advantage	Benefits
High Performance	High efficiency	Meet and exceed design performance specs. Lower design risks. Enhance end-user satisfaction.
	High isolation	Less interaction with surrounding components. Smallest effective antenna size when component keep-out areas are included. Resists detuning due to orientation on circuit board. Lowers design risk and time to market. One antenna part number can serve multiple designs. Simplifies design and ordering.
	High selectivity	Eliminates need for additional band-pass filters and other circuitry. Saves cost and space.
Superior RF Field Containment	Virtually eliminates detuning	Better performance. Higher end-user satisfaction.

Product Selection Guide

Antenna PN	Application	Antenna PN Application Type Typical Deliverable	Typical Deliverable Size
1000146	WLAN 2.4, 4.9 GHz 5.2, 5.8 GHz	• Ground Cleared • 2.4 & 5 GHz • Flexible antenna placement • Antenna element from 1000418	• Antenna element only • SMT • 17.9 X 6.9 X 4.3 mm (Antenna only)

WLAN/BT/Zigbee/Wi-Fi/Embedded Stamp Metal Antenna 1000146

2.4 / 4.9 / 5.2 / 5.8 GHz (802.11 a/b/g/n/c + Japan)

Prestta Features and Benefits Summary

Features	Benefits
Stamped Metal Antennas with SMD capability	• Flexibility in antenna placement with direct placement on • Ease of manufacturing
Embedded Solutions for WLAN	• Eliminates external antennas • More desirable form factors • Can be used in Access Points, Routers, Gateways, Wireless Displays/TVs, and other consumer electronic devices
High Performance	• Better performance than external dipole in diversity antenna • Situation
Ground Cleared Solution	• Enables flexibility in antenna placement within end device • Can be used within Access Points, Routers, Handhelds, Displays
Extensive Design Collateral and Apps Support	• Speeds development time
Standard “Off the Shelf” Product	• Standard "Off the Shelf" Product • Speed development time and reduces costs by reducing NRE and custom development time

Design Guidelines

Introduction

The Prestta standard WLAN embedded antenna can be designed into many wireless product types. The following sections explain KYOCERA AVX's recommended layouts to help the designer integrate the 1000146 antenna element into a device with optimum performance.

Electrical Specifications

Typical Characteristics Measurements taken on a 120 x 180 mm PCB

Features	2400 – 2485 MHz	4900 – 5825 MHz
Peak Gain	1.5 dBi	2.6 dBi
Average Efficiency	80%	72%
VSWR Match	1.5:1 max	1.6:1 max
Feed Point Impedance	50 ohms unbalanced	
Polarization	Linear	
Power Handling	0.5 Watt CW	

Mechanical Specifications

Ordering Part Number	1000146
Size (mm)	17.85 x 6.9 x 4.3
Mounting	SMT
Weight (grams)	0.35
Packaging	Tape & Reel, 1000146 – 1,200 pieces per reel
Demo Board	1000418

WLAN/BT/Zigbee/Wi-Fi/Embedded Stamp Metal Antenna 1000146

2.4 / 4.9 / 5.2 / 5.8 GHz (802.11 a/b/g/n/c + Japan)

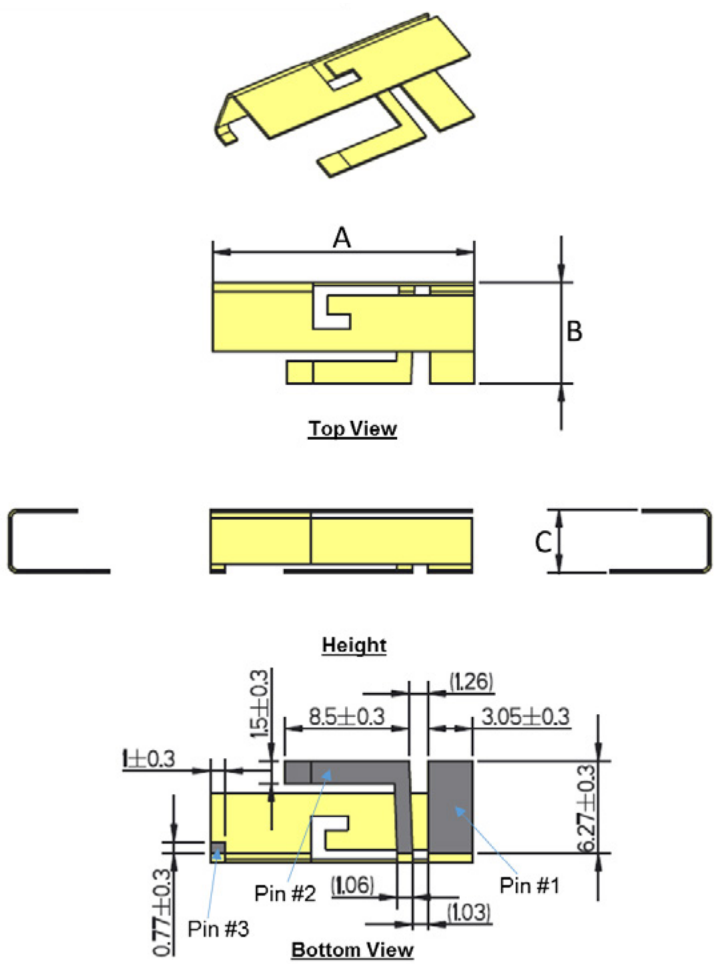
Antenna Dimension and Pad Layout

Figure 1 below shows the Antenna Dimensions and Pad Layout for 1000146

Antenna Dimensions

Typical antenna dimensions (mm)

Features	A (mm)	B (mm)	C (mm)
1000146	17.85 ± 0.3	6.9 ± 0.3	4.3 ± 0.4



Pin	Description
1	Feed
2	Ground
3	Dummy Pad

Figure 1 : Antenna Dimensions and Pad Layout for 1000146

Antenna Footprint Layout

Figure 2 below shows the 1000146 Minor Tuning Layout

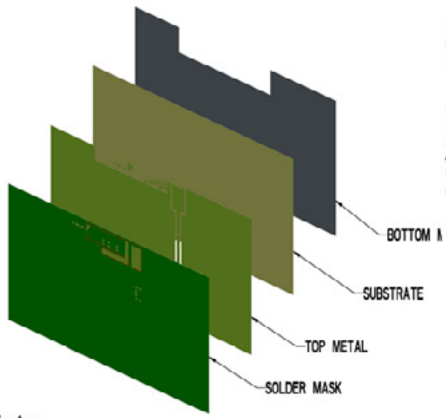
Figure 3 below shows the 1000146 Major Tuning Layout

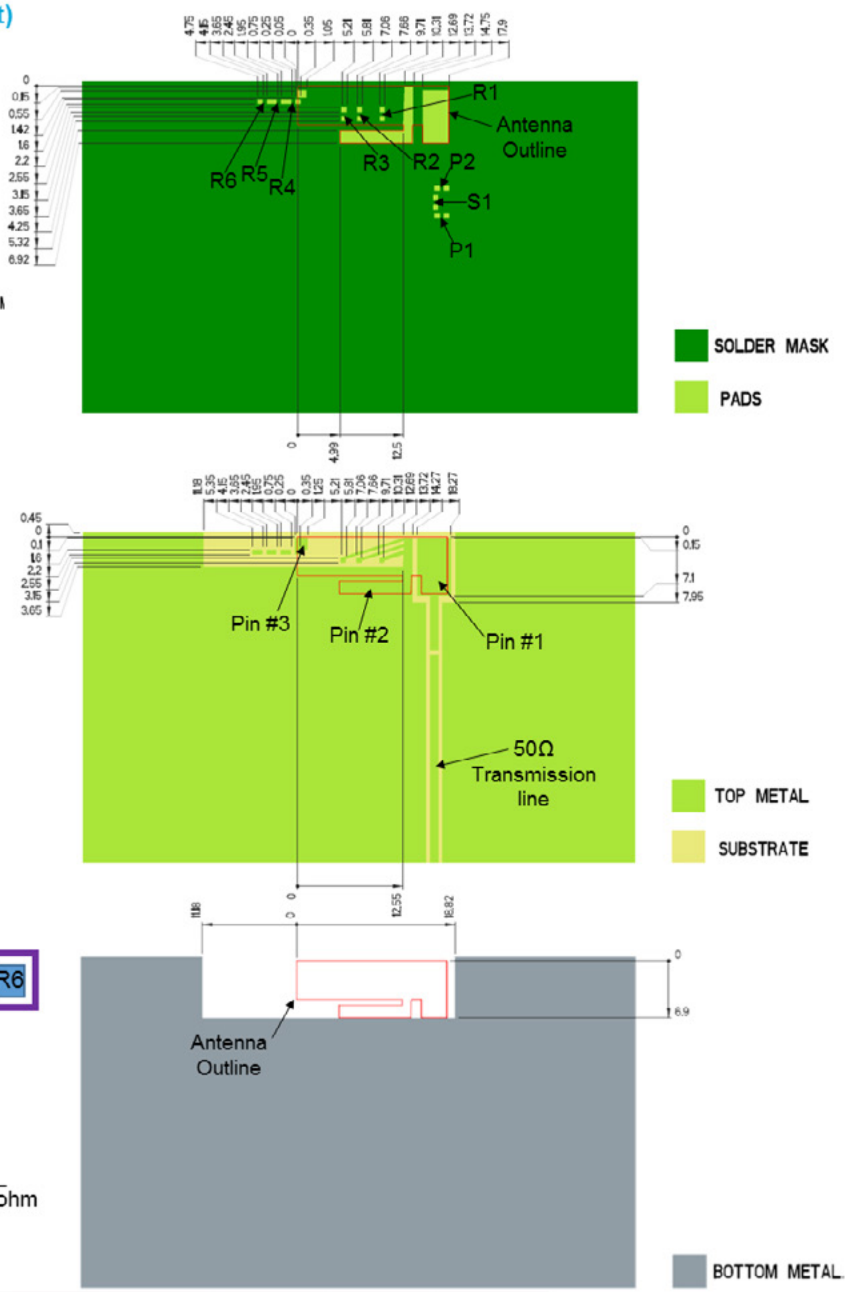
Figure 4 below shows the 1000146 Antenna Matching Structure (Major Tuning Structure)

Figure 2 below shows the 1000146 Minor Tuning Layout

Antenna Layout (Minor Tuning Layout)

Typical layout dimensions (mm)

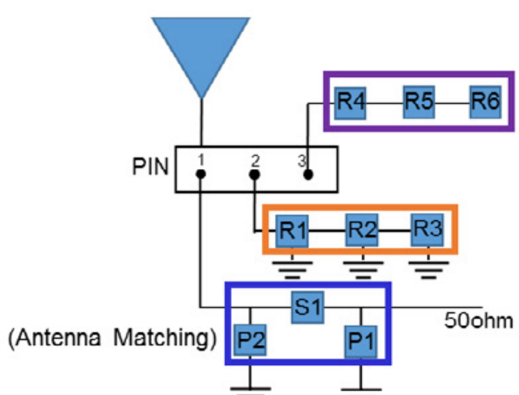




Note:
Layout has minor tuning capabilities to allow for small antenna footprint.

Pin Descriptions

Pin#	Description
1	Feed
2	Ground
3	Dummy Pad



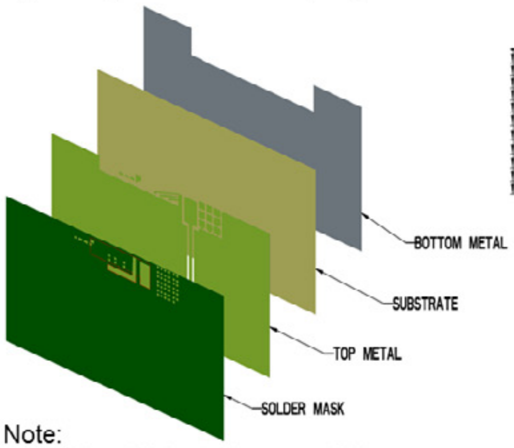
	P1	S1	P2	R1 – R3	R4 – R6
Default Matching	DNI	0Ω	DNI	DNI	DNI
Tolerance	N/A	N/A	N/A	N/A	N/A

Figure 2: 1000146 Minor Tuning Layout

Figure 2 below shows the 1000146 Minor Tuning Layout

Antenna Layout (Major Tuning Layout)

Typical layout dimensions (mm)



Note:
Layout has Major tuning capabilities to allow for robust tuning after board spin, instructions on **Antenna Matching Structure** page.

Pin Descriptions

Pin#	Description
1	Feed
2	Ground
3	Dummy Pad

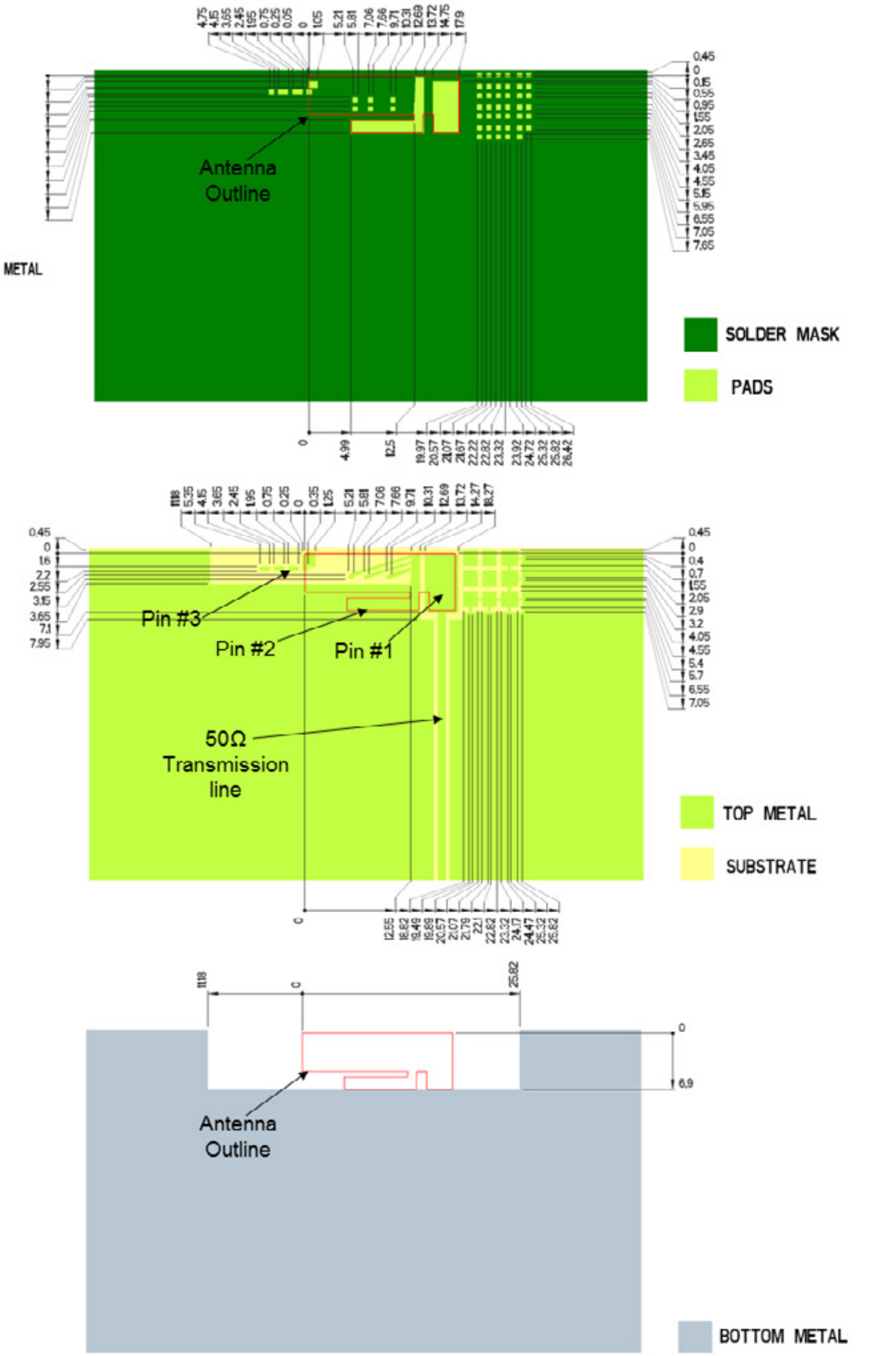
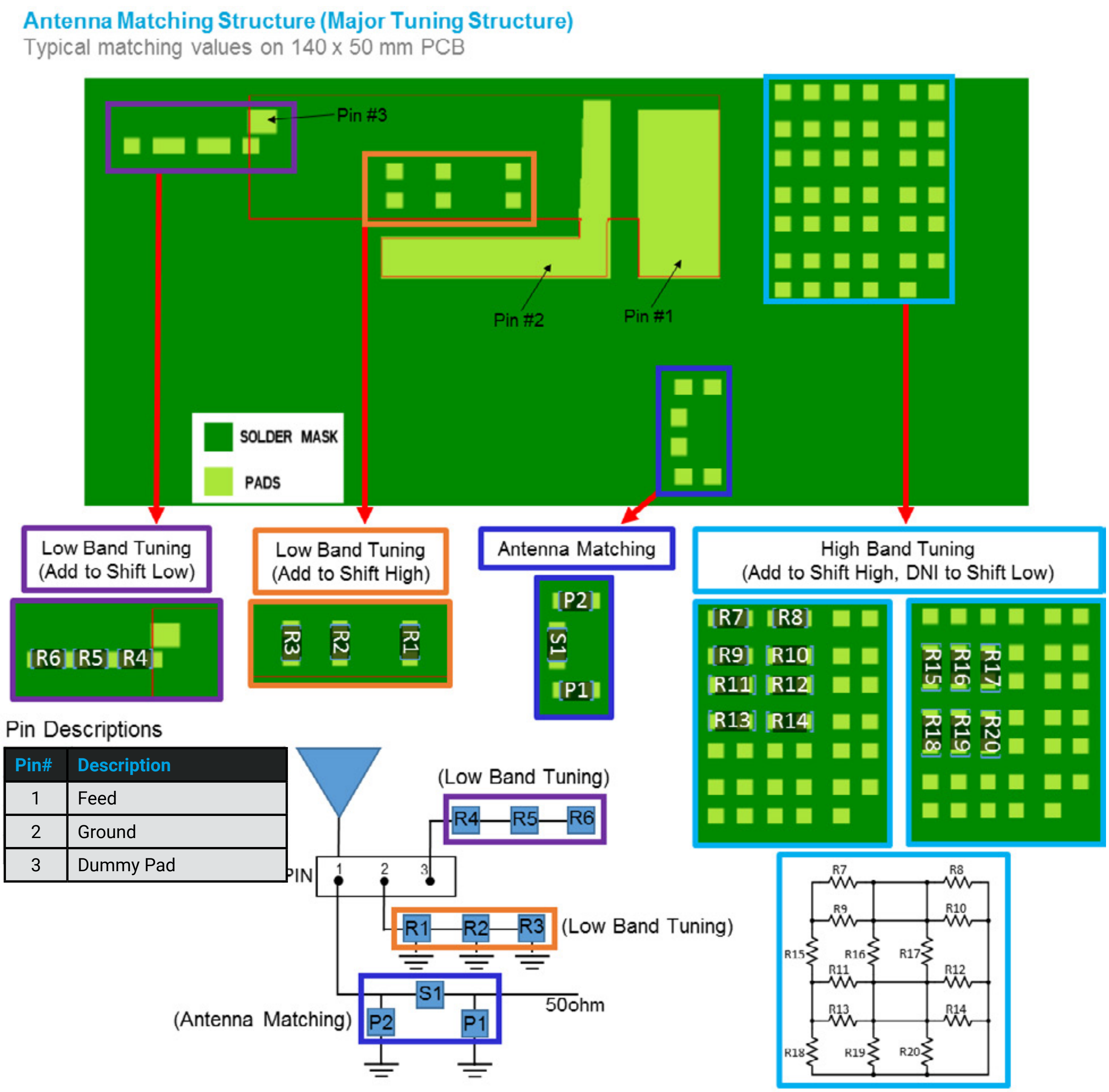


Figure 3: 1000146 Major Tuning Layout

Figure 3 below shows the matching structure



Pin#	Description
1	Feed
2	Ground
3	Dummy Pad

	P1	S1	P2	R1 – R3	R4 – R6	R7 – R14	R15 – R20
Default Matching	DNI	0Ω	DNI	DNI	DNI	DNI	DNI
Tolerance	N/A	N/A	N/A	N/A	N/A	N/A	N/A

Figure 4: 1000146 Antenna Matching Structure (Major Tuning Structure)

WLAN/BT/Zigbee/Wi-Fi/Embedded Stamp Metal Antenna 1000146

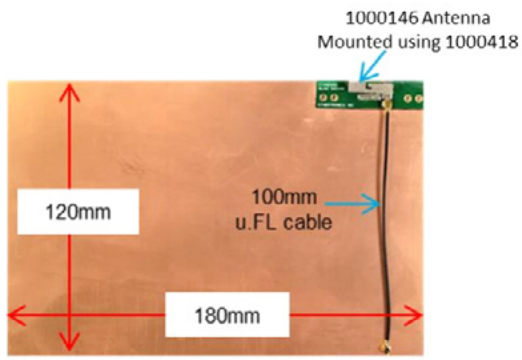
2.4 / 4.9 / 5.2 / 5.8 GHz (802.11 a/b/g/n/c + Japan)

Typical Measured Data

VSWR, Efficiency and Radiation Pattern

Figure 5 below shows the 1000146 Antenna Typical VSWR & Efficiency Plots on 120 x 180mm PCB

Figure 6 below shows the 1000146 Antenna Typical Radiation Pattern Plots on 120 x 180mm PCB



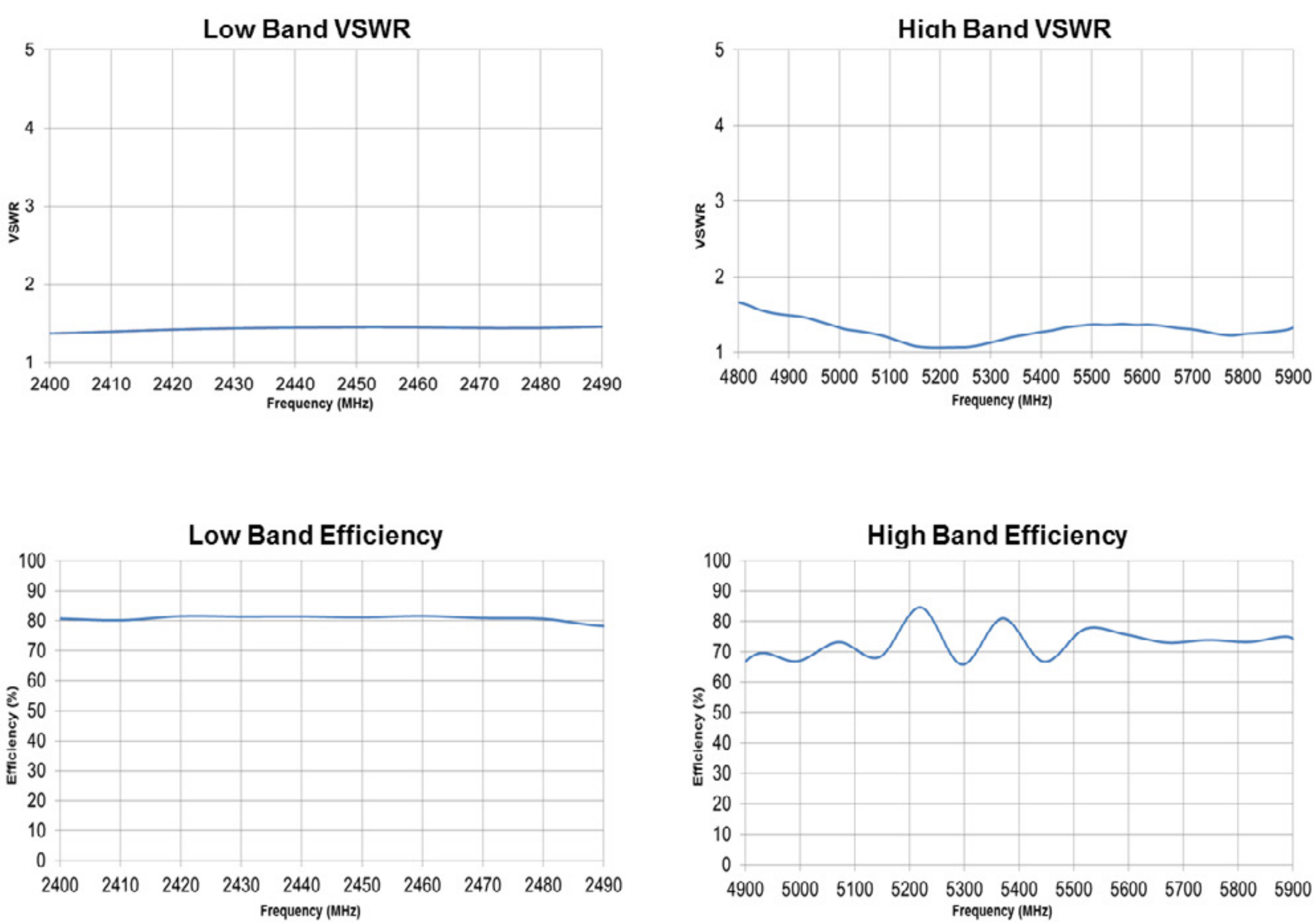
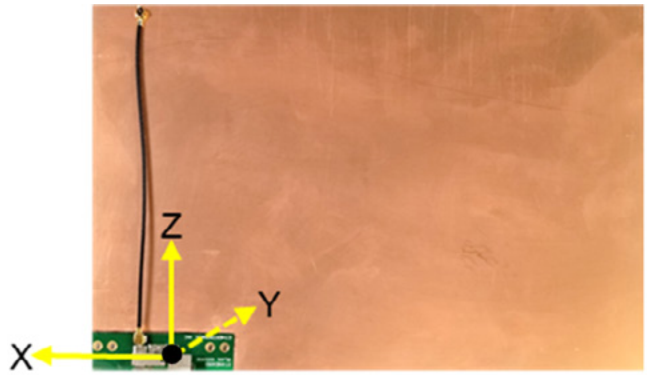


Figure 6: 1000146 Antenna Typical VSWR & Efficiency Plots on 120 x 180mm PCB



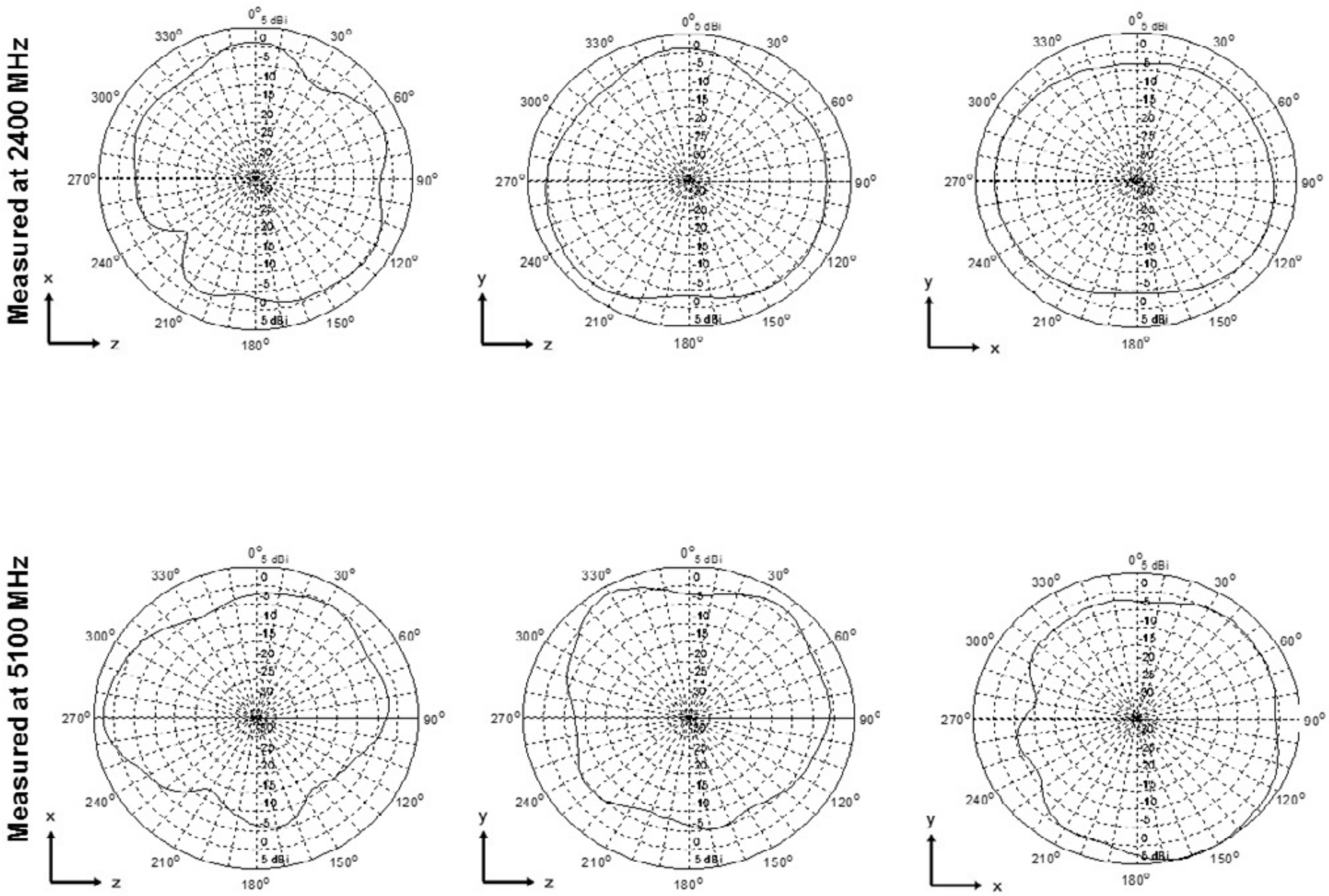


Figure 6: 1000146 Antenna Typical VSWR & Efficiency Plots on 120 x 180mm PCB

Antenna Placement Guidelines on PCB

The 1000146 antenna is a metal element which can be mounted onto any PCB using KYOCERA AVX's recommended footprint layout and ground layout with proper PCB size. Based on 1000146 antenna element, 1000418 is an antenna module assembly with PCB, 1000146 metal parts, U.FL connector and coax cable. The 1000418 antenna module assembly can be easily used as an off board antenna directly connecting to the RF module board through proper coax cable.

- Antenna should always be placed along the edge of the board unless there are special conditions preventing this.
- The antenna can be placed on either the top or bottom side of the PCB. The recommended antenna location, when you are looking at the board, lies close to upper and right edge with a minimum 15mm ground distance from Feed side of antenna to PCB edge shown in (Figure 7).

Figure 7 shows the optimal single antenna placement for 1000146

Figure 8 shows the optimal antennas placement for 1000146 on a large PCB

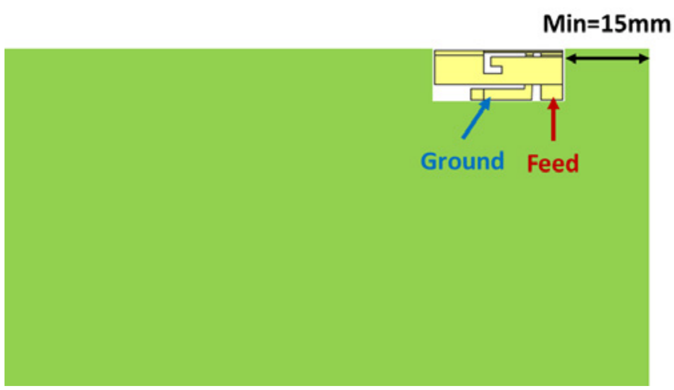


Figure 7: optimal single antenna placement for 1000146

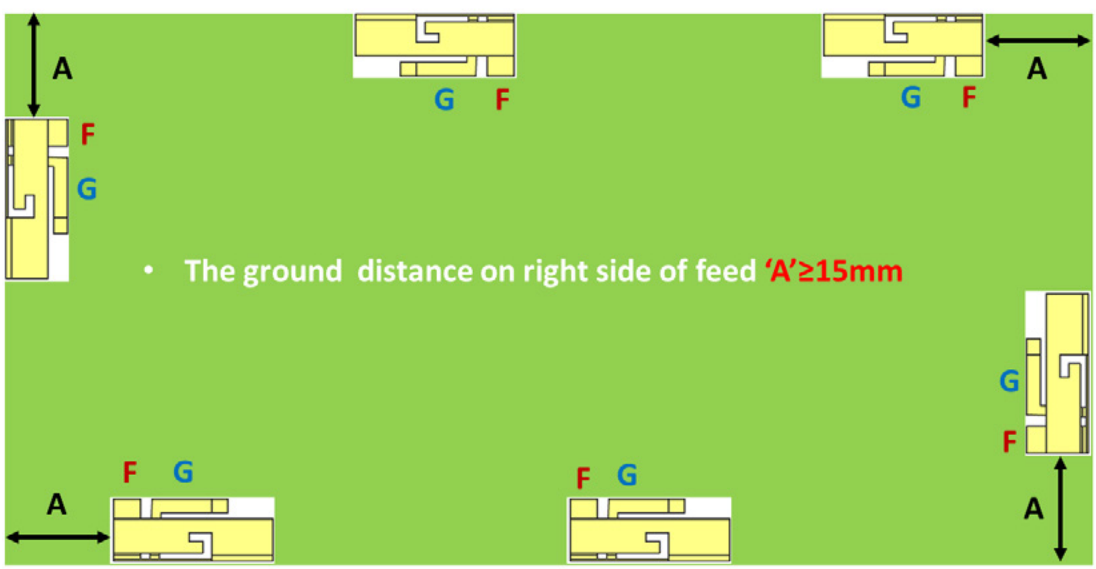


Figure 8: optimal antennas placement for 1000146 on a large PCB

Antenna Tuning Guidelines

In real application environments, variation of the antenna resonating frequency may be caused by the following: Different antenna locations, PCB board variations (including PCB size and PCB thickness), Component(s) and shield cans located close to the antenna, Outside Cover and metal element from inside or outside of device, etc.

Currently for 1000146, there are two types of antenna footprint layouts

- Minor Tuning Layout: layout has minor tuning capabilities to allow for small antenna footprint, and incorporates tuning pads for low band tuning.
- Major Tuning Layout: layout has major tuning capabilities to allow for robust tuning after board spin, and this layout requires slightly larger footprint space on the right side of antenna feed. This tuning layout is including the Minor Tuning Layout and will work on both low band and high band tuning.

Based on the Major Tuning Layout, the following methods can be applied to solve the above effects

- Major Tuning Through the Tuning Pad Printed on the PCB
- Minor Tuning Through Matching Circuit Guidelines

Major Tuning Through the Tuning Pad Printed on the PCB

Antenna Tuning Pad can be considered as a part of antenna which allow shifting the antenna frequency resonance lower or higher by adding/removing 0 ohm resistors on the tuning pad layout. Adding 0 ohm resistors to connect two isolated metal pads is equivalent to increasing the antenna physical length. In opposite, removing 0 ohm resistors is to isolate two metal pads which is equivalent to reducing the antenna physical length. The advantages of using tuning pads enables antenna tuning directly on board to avoid or reduce the re-spin times of the customer PCB.

- Low Band (2.4GHz Band) Tuning Through Low Band Tuning Pad.
- High Band (5GHz Band) Tuning Through High Band Tuning Pad.

Figure 9 shows 1000146 Major Tuning Layout Structure
Figure 10 shows 1000146 Low Band Tuning Pad Configurations
Figure 11 shows 1000146 High Band Tuning Pad Configurations

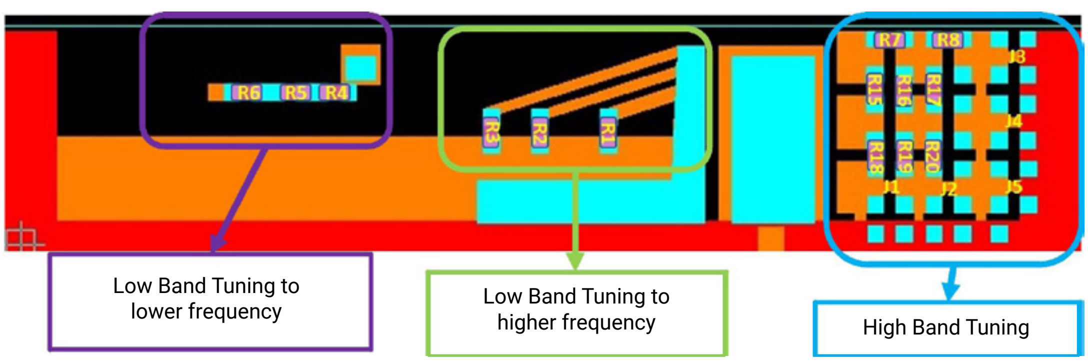
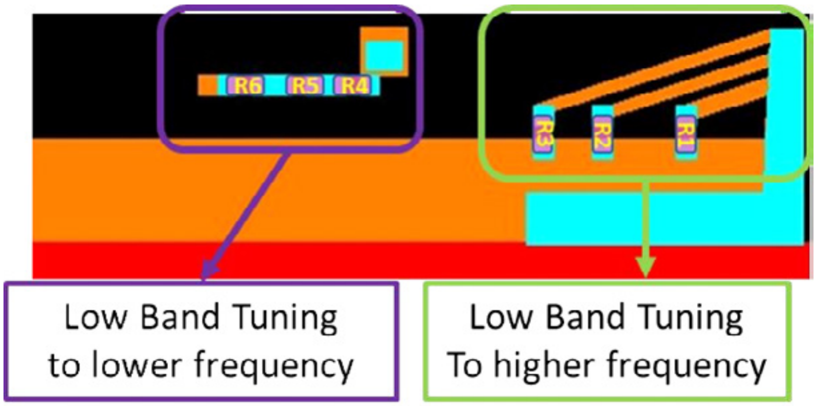


Figure 9: 1000146 Major Tuning Layout Structure



Low Band Tuning Pad Length	R1	R2	R3	R4	R5	R6
Low Band Pad Length 1 (Default)	DNI	DNI	DNI	DNI	DNI	DNI
Low Band Pad Length 2	0Ω	DNI	DNI	DNI	DNI	DNI
Low Band Pad Length 3	0Ω	0Ω	DNI	DNI	DNI	DNI
Low Band Pad Length 4	0Ω	0Ω	0Ω	DNI	DNI	DNI
Low Band Pad Length 5	DNI	DNI	DNI	0Ω	DNI	DNI
Low Band Pad Length 6	DNI	DNI	DNI	0Ω	0Ω	DNI
Low Band Pad Length 7	DNI	DNI	DNI	0Ω	0Ω	0Ω

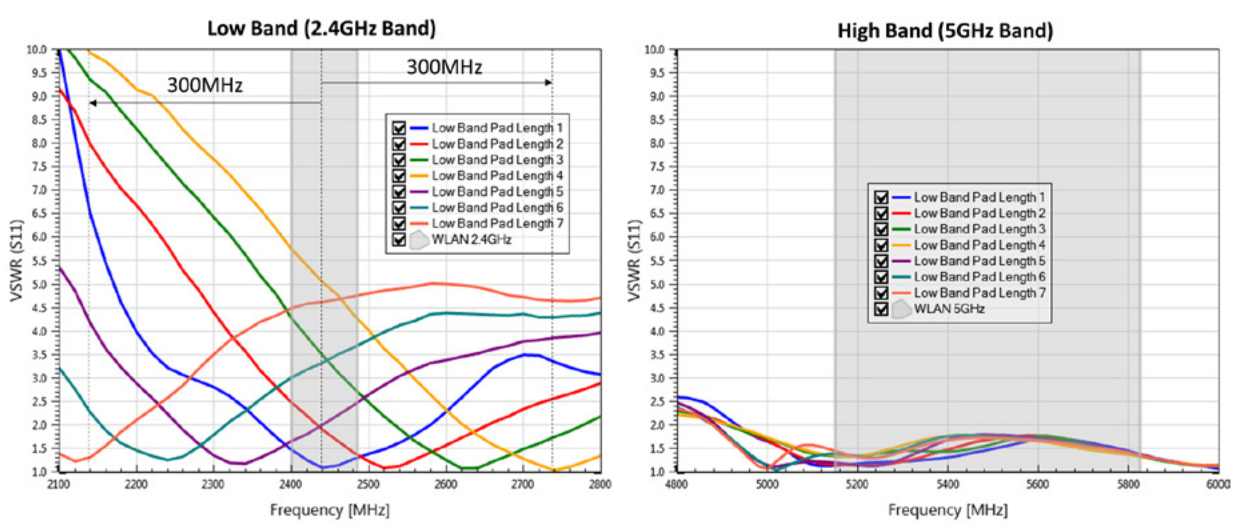
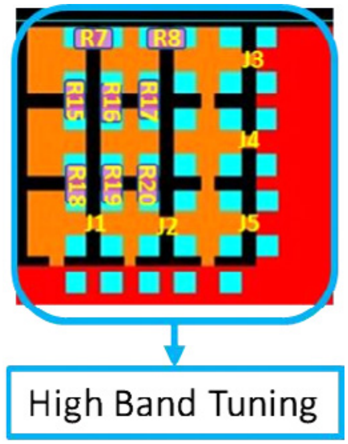


Figure 10: 1000146 Low Band Pad Configurations



Low Band Tuning Pad Length	R7	R8	R15	R16	R17	R18	R19	R20
Low Band Pad Length 1 (Default)	DNI	DNI	DNI	DNI	DNI	DNI	DNI	DNI
Low Band Pad Length 2	DNI	0Ω	0Ω	0Ω	DNI	0Ω	0Ω	DNI
Low Band Pad Length 3	0Ω	DNI	0Ω	DNI	0Ω	0Ω	DNI	0Ω
Low Band Pad Length 4	0Ω	0Ω	0Ω	DNI	DNI	0Ω	DNI	DNI
Low Band Pad Length 5	DNI	0Ω	DNI	0Ω	DNI	DNI	0Ω	DNI
Low Band Pad Length 6	DNI	0Ω	DNI	DNI	DNI	DNI	DNI	DNI
Low Band Pad Length 7	0Ω	0Ω	DNI	DNI	DNI	DNI	DNI	DNI

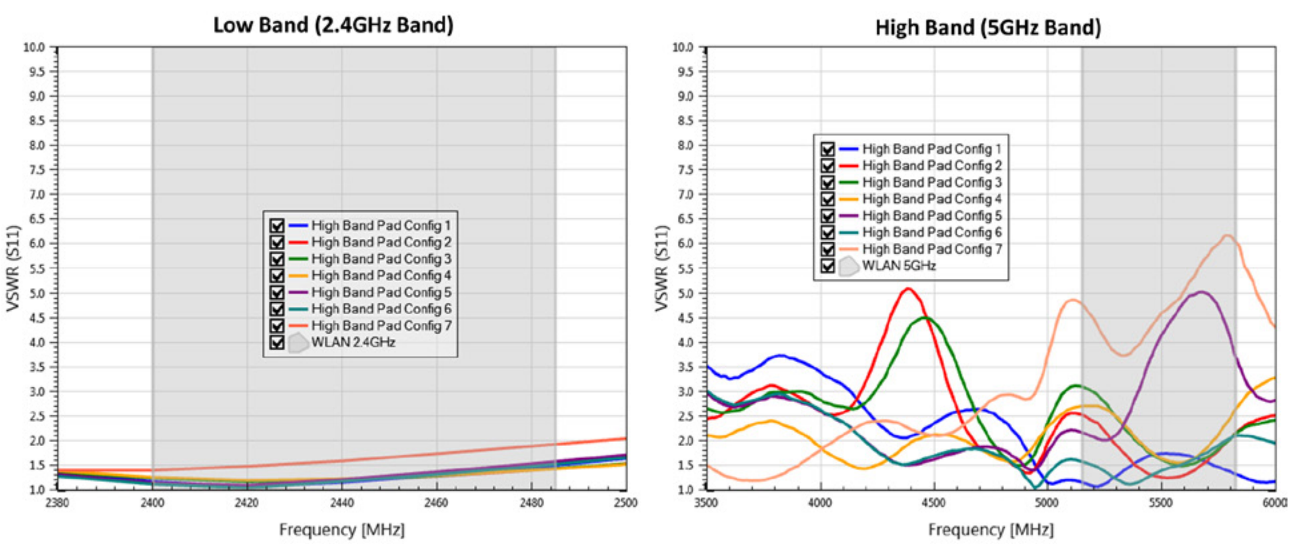
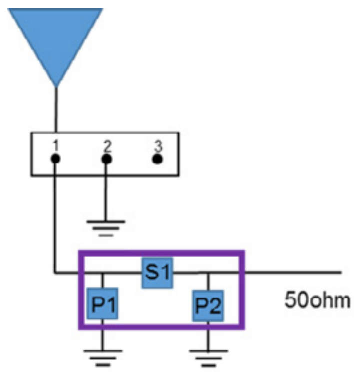


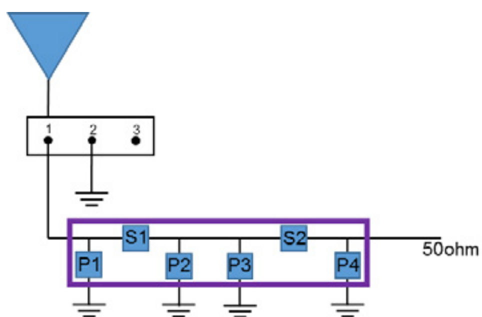
Figure 11: 1000146 High Band Tuning Pad Configurations

Minor Tuning Through Matching Circuit Guidelines

Performance can also be improved by tuning the matching circuit. Optimum matching values may vary based on the boards transmission line design, the antenna location, the PCB size and the antenna working environment. Nevertheless, the antenna performance can be improved by modifying the tuning pad as mentioned in the previous section, and optimizing the matching components accordingly. For the single-band application (WiFi 2.4GHz single band or Bluetooth 2.4GHz), if the frequency is slightly off the required band, one "pi" type of matching network is enough to tune frequency back. In general, two matching components are enough. (Using P1 & S1 or S1 & P2 from network below)



For the dual-band design (WiFi 2.4GHz & 5GHz dual band), if the frequency are slightly out of the required bands, a double "pi" type of matching is preferred, one "pi" network will be for low band tuning and another "pi" network will be for high band tuning. In many cases, there is only one "pi" network available on the board. If this is the case, use the tuning pads to perform band tuning for the first band and obtain a good impedance, and then optimize the other band using the matching components and tuning pad configuration accordingly.



Shield Can Tuning Guidelines

A 60x45x5 mm shield can is placed close to the antenna to show its effect. The shield can causes frequency shifting of the antenna but the peak efficiency is not affected. Therefore, when a shield can is placed close to the antenna, the detuning effect can be compensated by frequency tuning through methods mentioned earlier.

- For VSWR, The shield can causes the low band center frequency to shift lower, The closer the shield can is (smaller "d"), the more the frequency shift. But less effect on high band.
- For Efficiency, the peak efficiency of low band appears to shift due to the shield can effect, however, the same peak value is always achieved. Not too much effect on high band.

Figure 12 shows a Shield Can Demonstration with 1000146 Antenna.
Figure 13 shows the Shield Can affect on VSWR & Efficiency of 1000146 antenna with varied distances.

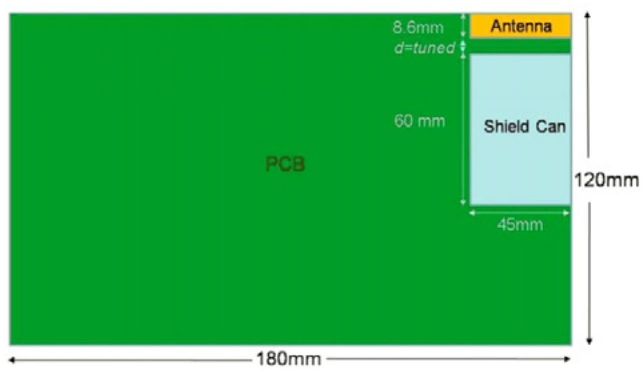


Figure 12: Shield Can Demonstration with 1000146

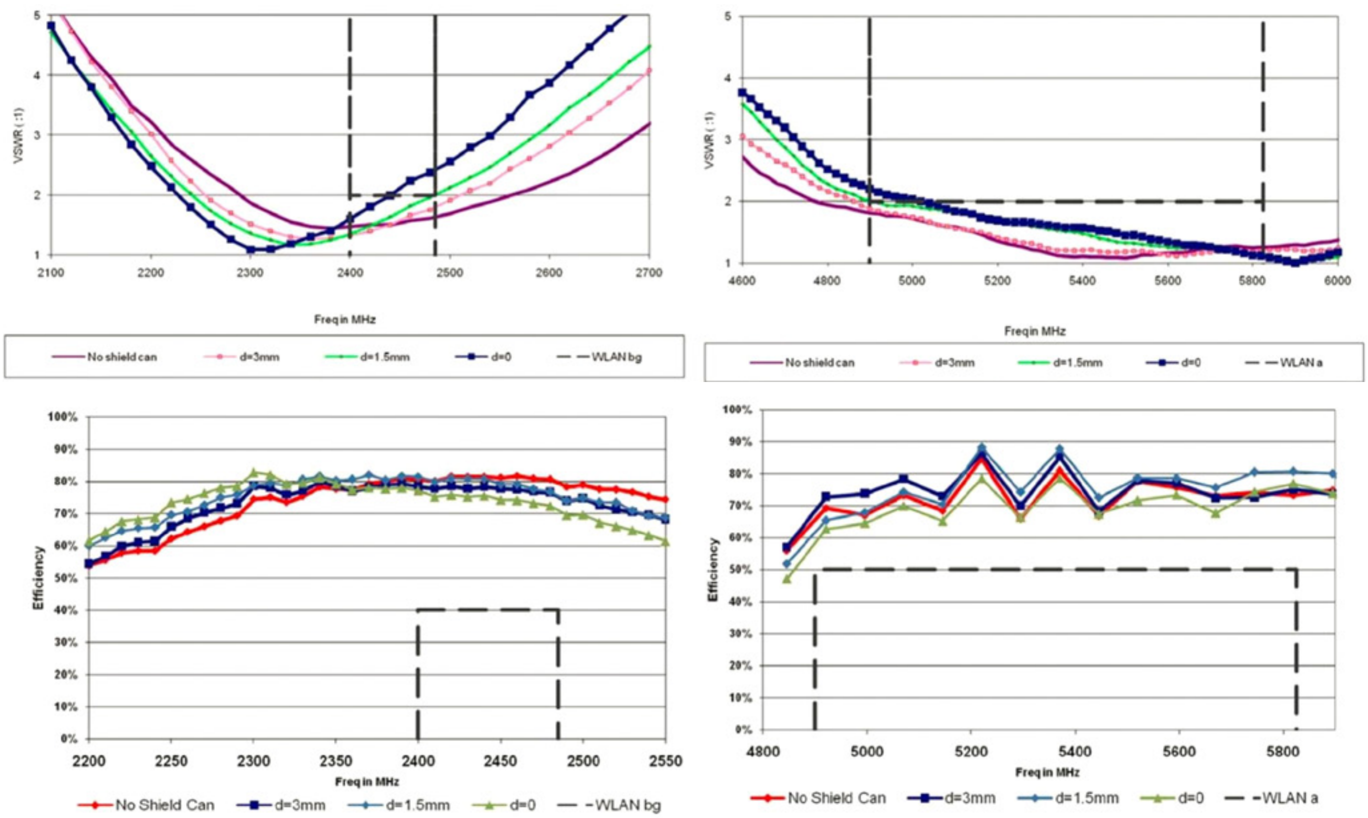


Figure 13: Shield can effect on VSWR & Efficiency of 1000146 antenna based on the distance

Space Saving Configuration–PCB Land Pattern

With the Space Saving Configuration, the antenna layout is the most compact layout design for the 1000146 antenna. Proper evaluation and tuning during prototype development stage will allow for this configuration layout compared to other shown previously.

- Maximum VSWR of low band will increased from 1.7:1 to 2:1
- Average efficiency of low band will be decreased from 81% to 69%

Figure 14 shows the PCB Land Pattern of Space Saving Configuration for 1000146
Figure 15 shows the Space Saving Configuration effect on VSWR & Efficiency for 1000146

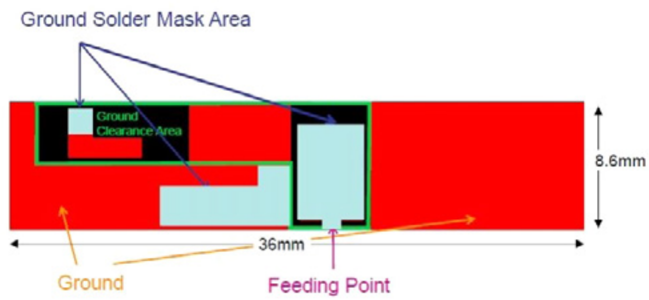


Figure 14: PCB Land Pattern of Space Saving Configuration for 1000146

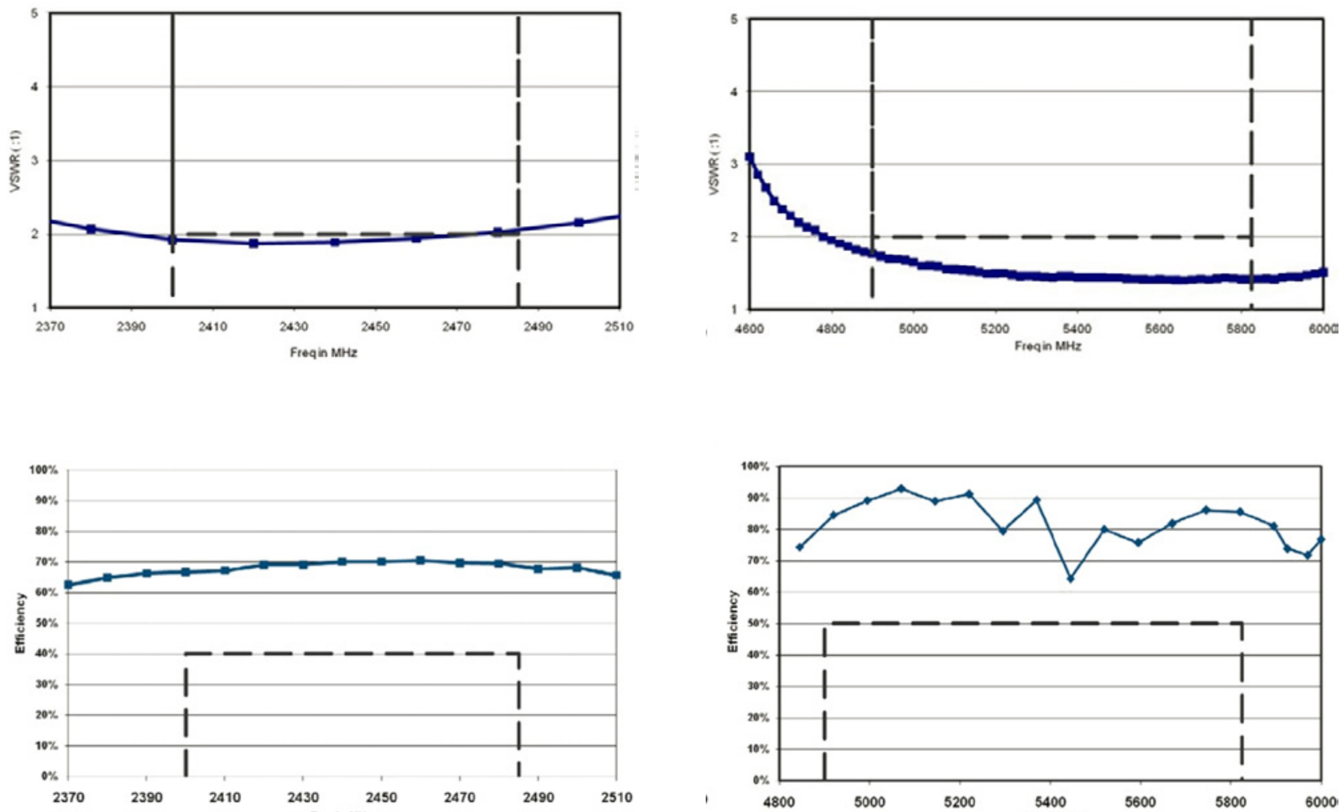


Figure 15: Space saving configuration effect on VSWR & Efficiency for 1000146

MIMO Application Guidelines

Figure 16 below shows the Recommended Layout for MIMO Applications
Place two antennas on two perpendicular edges of a board.

- The recommended antenna edge-to-edge distance is 85mm or larger for in-band isolation of 20dB or greater. (Note, this will depend on the environment in which the two antennas are located)
- To improve isolation in limited space, KYOCERA AVX has developed a special isolator approach that need to be fully customized to the end device board. As an example, minimum 30dB isolation can be achieved with an antenna edge-to-edge distance of 45mm. Please contact KYOCERA AVX directly for more information

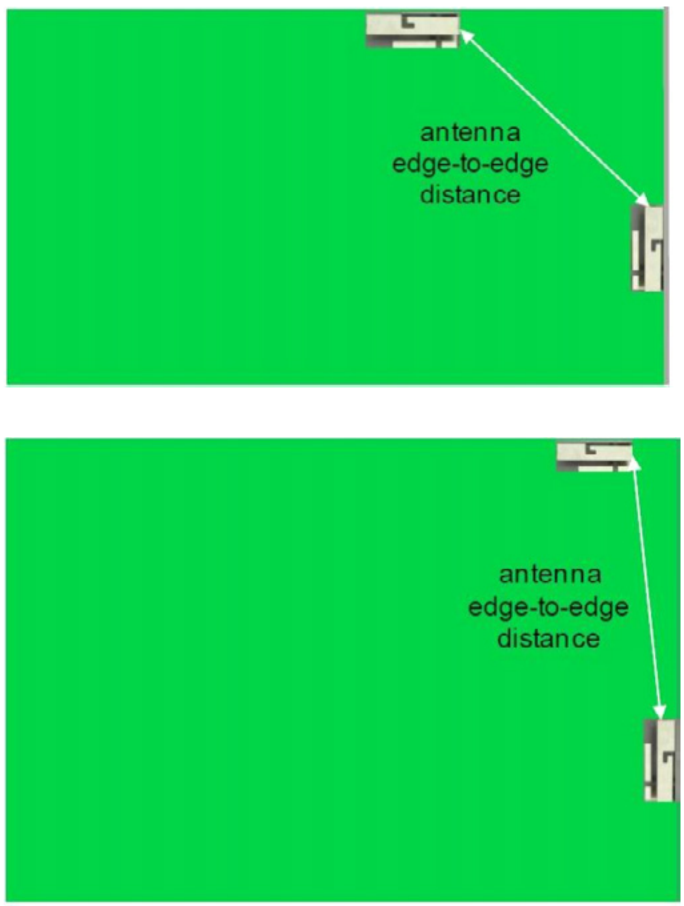


Figure 16: Recommended Layout for MIMO Applications

Figure 17 below shows the recommended guidelines when two antennas must be placed on the same edge of a PCB:

- The edge-to-edge distance should be 80mm or greater for in-band isolation of 20dB or greater.

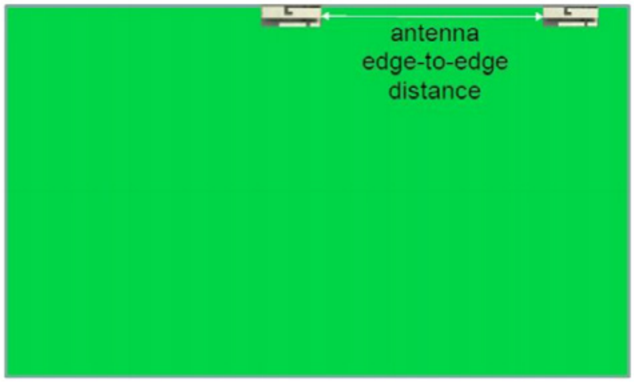


Figure 17 : Recommended guidelines for two antennas placed on the same edge of PCB

Figure 18 below shows the recommended guidelines when two antennas must be placed on opposite edges of a PCB.

- The edge-to-edge distance should be 100mm or greater.

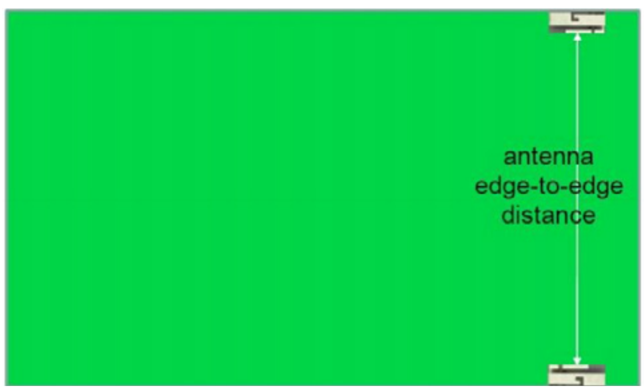


Figure 18 : Recommended guidelines for two antennas placed on opposite edges of PCB

MIMO Application Example

Figure 19 below shows a typical MIMO configuration where two 1000146 antennas and a shield can are placed close together in the corner of a PCB.

- Both antennas have typical VSWR and efficiency performances described above.
- The measured In-band isolation is less than –30dB as shown in Figure 20 below.

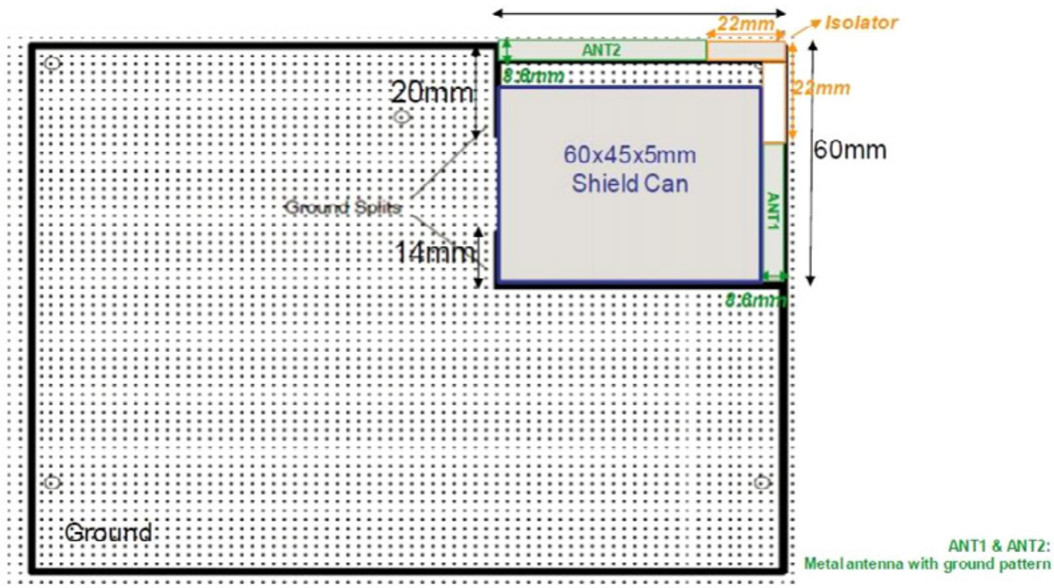


Figure 19: A Typical MIMO Configuration Example

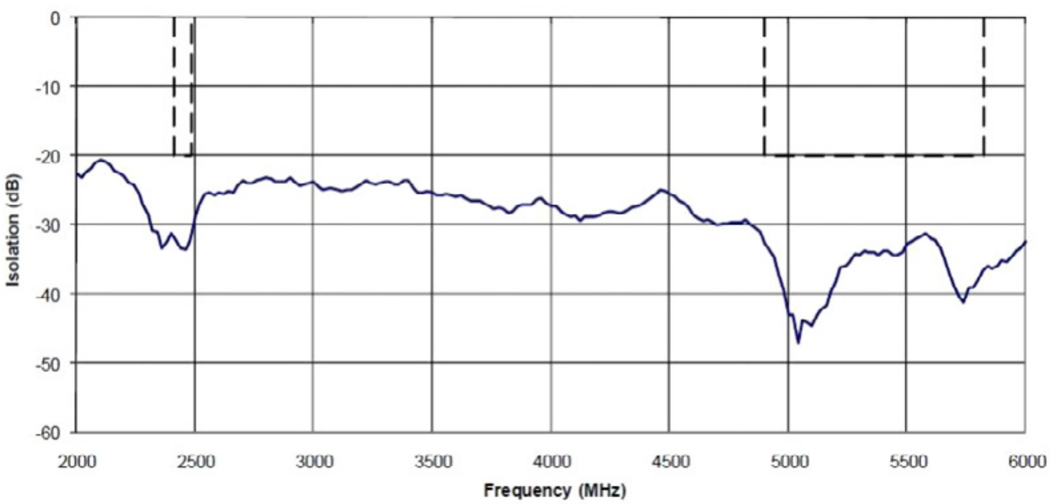


Figure 20: Measured Isolation Between Two 1000146 MIMO Antennas

WLAN/BT/Zigbee/Wi-Fi/Embedded Stamp Metal Antenna 1000146

2.4 / 4.9 / 5.2 / 5.8 GHz (802.11 a/b/g/n/c + Japan)

Material Specifications

Item	Material
Metal Element	C5210
Contact Finish	Ni and selective Au standard

Manufacturing and Assembly Guidelines

KYOCERA AVX's Prestta Standard WLAN antennas are designed for high volume board assembly. Because different product designs use different numbers and types of devices, solder paste, and circuit boards, no single manufacturing process is best for all PCBs. The following recommendations have been deter- mined by KYOCERA AVX, based on successful manufacturing processes.

The metal antenna only and metal antenna with carrier solutions are designed for automated pick and place surface mounting. However, as with any SMT device, KYOCERA AVX antennas can be damaged by the use of excessive force during the handling or mounting operation.

Component Handling Recommendations

The following are some recommendations for component handling and automated mounting:

- For manual mounting and handling, vacuum pens should be used to pick-up, transfer and mount the antennas.
- Take care not to deform the metal antenna the following are some recommendations for component handling and automated mounting:

KYOCERA AVX's metal antennas are not moisture sensitive and the antennas meet the requirements for a Level 1 classification of J-STD-020A (moisture/ reflow sensitivity classification for non-hermetic solid state surface mount devices from the Institute for Interconnecting and Packaging Electronic Circuits). Nevertheless, as a precaution to maintain the highest level of solder ability, KYOCERA AVX antennas are dry-packed.

(NOTE: Normal oxidation may result in a slight discoloration of the gold nickel surface. This has no effect on the performance of the antenna.)

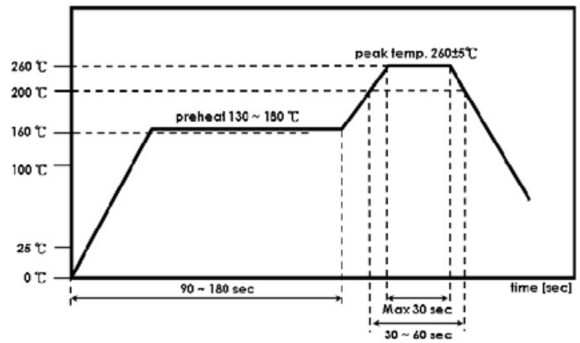
Paste Stencil Recommendation

KYOCERA AVX recommends application of paste stencil to a thickness of 0.1mm, applied to within 0.125 mm of the solder mask surrounding each exposed metal pad on the PCB. PCB layouts for each antenna are provided in earlier section of this document

Soldering Recommendations

The recommended method for soldering the antenna to the board is forced convection reflow soldering.
The following suggestions provide information on how to optimize the reflow process for the antenna:

Adjust the reflow duration to create good solder joints without raising the antenna temperature beyond the allowed maximum of 260° C.



Additional Manufacturing Recommendations

Care should be taken during certain customer-specific manufacturing processes including PCB separation and Ultrasonic Welding to ensure these processes don’t create damage to the components.

Cleaning Recommendations

After the soldering process, a simple wash with de-ionized water sufficiently removes most residues from the PCB. Most board assembly manufacturers use either water-soluble fluxes with water wash, or “no clean” fluxes that do not require cleaning after reflow.

Acceptable cleaning solvents are CFC alternatives, Isopropyl Alcohol (IPA), and water. If the application uses other types of solvents, please consult with KYOCERA AVX.

Cleaning processes that should be avoided are ultrasonic cleaning and any abrasive techniques, such as scrubbing with a cotton swab or with an abrasive material.

Rework & Removal Recommendations

There may be a need to rework or remove the antenna from the PCB. Although KYOCERA AVX's antennas are designed for ease-of-use, use care when separating them from the PCBs. Careless heating or removal of the antenna can cause thermal, mechanical or lead damage. These degradations may render the antenna useless, impeding any failure analysis and preventing the reuse of the device. Therefore it is recommended to observe the following precautions:

- The component can be reworked and soldered by hand using a soldering iron. However care should be used so the temperature does not exceed 260°. The soldering iron should not touch the composite material while soldering the leads of the antenna.
- The component can be reworked and soldered using a hot air rework station. However, care should be taken to ensure that the temperature does not exceed 260° C.
- Once the solder on the PCB is sufficiently heated, use a vacuum pen to lift the antenna straight up off the PCB. Avoid twisting or rotating the device while removing it.

Tape & Reel Specifications

Product will be shipped in Tape and Reel packaging

FOLLOW US:

VISIT US AT WWW.KYOCERA-AVX.COM

+1 (864) 967-2150
ETH.INFO@KYOCERA-AVX.COM