APPLICATION NOTES

AN-PT-MB-2427-060322

KYOCERA AVX

ISM or ISM & BT or GPS Stamp Metal Embedded SMT Antenna

868 MHz, 915 MHz, 1.575 GHz, 2.4 GHz

Applications:

- Cellular Handsets
- Wireless Headsets
- M2M
- Automotive
- Automatic Meter Reading
- Healthcare
- Point of Sale
- Tracking
- Smart Applications
- Tablets and Notebooks
- Other Wireless Devices
- PDAs Notebook PCs
- Industrial Devices
- Media Players
- Bluetooth

ISM or ISM & BT or GPS Stamp Metal Embedded SMT Antenna

Table of Contents

Purpose

This document provides information for incorporating KYOCERA AVX's 1002427 standard ISM or ISM & BT or GPS stamp metal embedded SMT antenna into wireless products. Specifications, design recommendations, board layout, packaging and manufacturing recommendations are included.

Overview

Product Selection Guide

Antenna PN	Application	Antenna PN Application Type Typical Deliverable	Typical Deliverable Size
1002427	• ISM 868-915 MHz • ISM & 915/2450 MHz • GPS 1560–1606 MHz	• Partial Ground • Flexible antenna placement	• SMT mountable antenna assembly • 32.1 x 3.9 x 2.28 mm
1002427-01	• ISM 868-915 MHz • ISM & BT 915/2450 MHz • GPS 1560-1606	• Demo Board	• Antenna Assembly on PCB board • 110 x 55 mm • SMA connector

Difference between On Ground application

Features	Advantage
Stamping metal structure with Small Form Factor & Ground clearance Requirements	• Flexibility in antenna placement • Ability to source antenna only for direct placement on customer PCB
High Performance Embedded Solution	• Greater than 60% average efficiency across all bands
Extensive design collateral and apps support	• Speeds development time
Standard “Off-the-Shelf” Product	• Speeds development time and reduces costs since reduces NRE and custom development time
Small Form Factor & Ground Clearance Requirements	• Can be used in a variety of custom form factors and applications
Cost Effective & Rugged Design	• SMT, Pick and Place, Tape & Reel Packaging, Enable lower manufacturing costs.

One antenna, multiple configuration

Across this document, you will find one particularity of the 1002427 antenna : be adjusting the antenna board layout, the 1002427 can operate as a Bluetooth/WIFI antenna, a GNSS antenna or a ISM antennas. The different layouts are available from page 8 to 10.

Design Guidelines

Introduction

The ISM or ISM & BT or GPS Stamp Metal Embedded SMT Antenna can be designed into many wireless product types. The following sections explain KYOCERA AVX's recommended layouts to help the designer integrate the 1002427 antenna into a product with optimum performance.

Electrical Specifications

Typical Characteristics Measurements taken with a matching circuit on a 110 x 55 mm ground plane.

Frequency	ISM Layout 868-915 MHz	ISM & BT Layout 915/2450 MHz		GPS Layout 1560-1606 MHz
Peak Gain	1.0 dBi	3.0 dBi	3.4 dBi	1.8 dBi
Average Efficiency	62%	67%	61%	65%
VSWR Match	2.0:1 max			
Feed Point Impedance	50 ohms unbalanced			
Polarization	Linear			
Power Handling	2 Watt CW			

Mechanical Specifications

Ordering Part Number	1002427
Size (mm)	32.1 x 3.9 x 2.28
Mounting	SMT, Through-Hole, (P&P)
Weight (grams)	0.16
Packaging	Tape & Reel, 5,400 pieces per box, 1,800 per reel
Demo Board	1002427-01

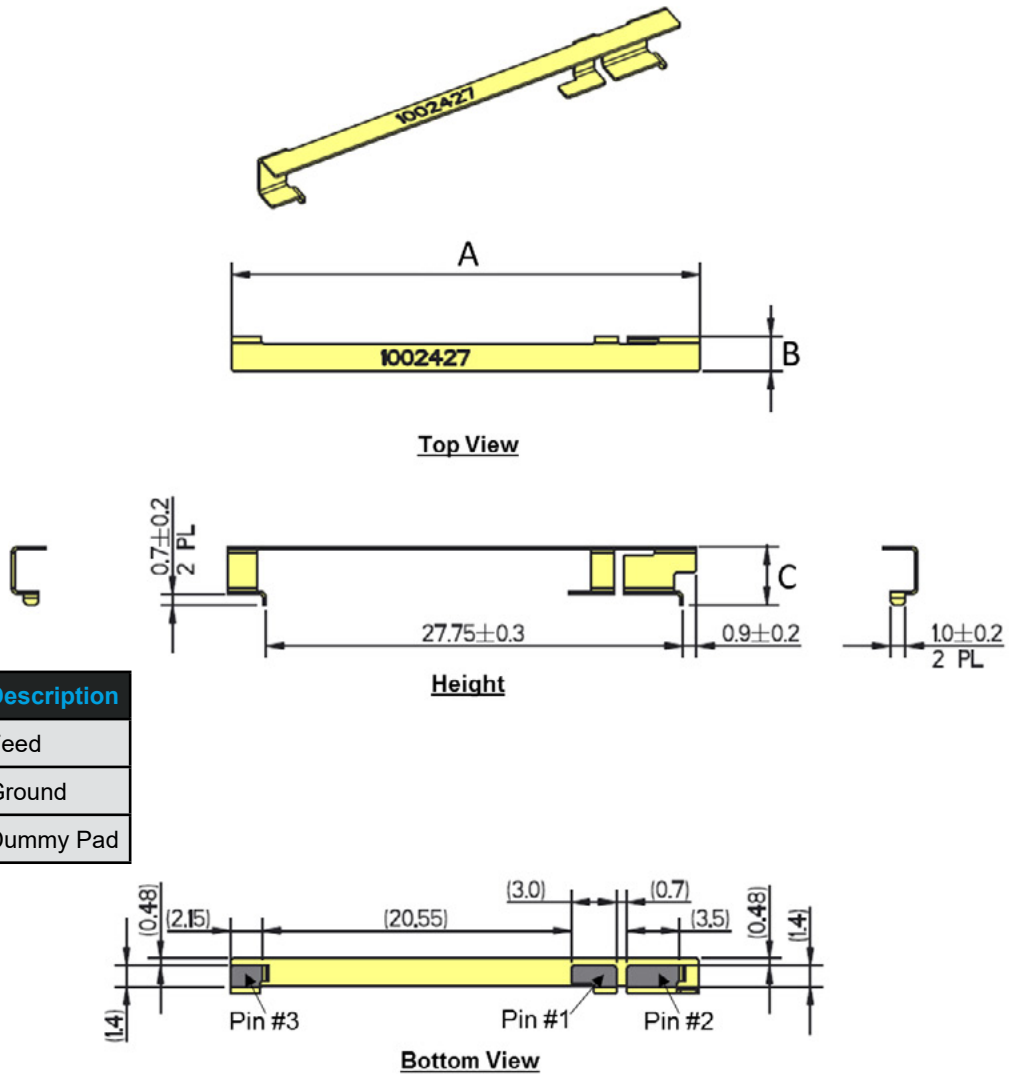
Antenna Dimension and Pad Layout

Figure 1 below shows the Antenna Dimensions and Pad Layout for 1002427

Antenna Dimensions

Typical antenna dimensions (mm)

Part Number	A (mm)	B (mm)	C (mm)
1002427	31.2 ± 0.3	2.28 ± 0.3	3.9 ± 0.4



Pin #	Description
1	Feed
2	Ground
3	Dummy Pad

Figure 1 Antenna Dimensions for the 1002427

Antenna Footprint Layout

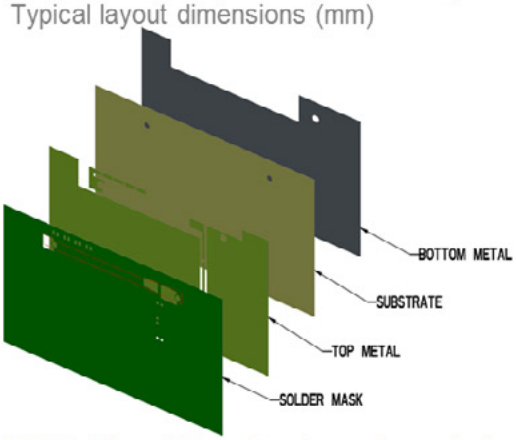
Figure 2 below shows the 1002427 Antenna Footprints for GPS ONLY

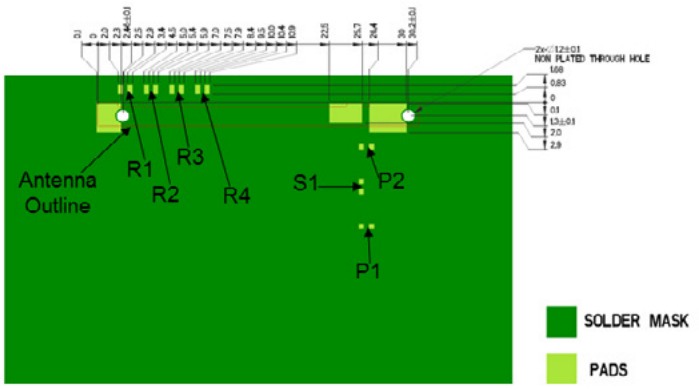
Figure 3 below shows the 1002427 Antenna Footprints for ISM ONLY

Figure 4 below shows the 1002427 Antenna Footprints for ISM & BT Dual Band

Antenna Layout (GPS Layout ONLY)

Typical layout dimensions (mm)

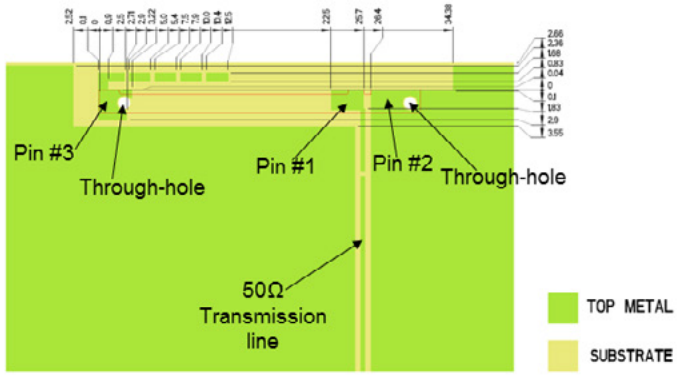




* VIAS: Diam. 0.2mm, (no vias on transmission lines).
Via holes must be covered by solder mask

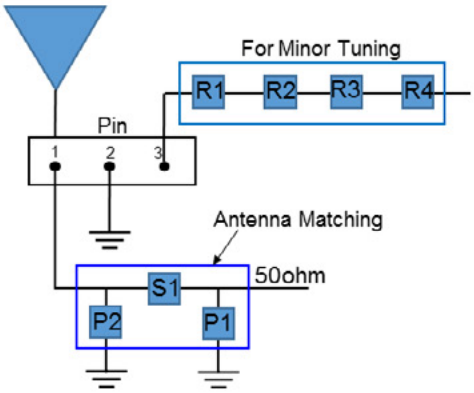
Pin Descriptions

Pin#	Description
1	Feed
2	Ground
3	Dummy Pad



Matching Pi Network

Component	Value	Tolerance
P1	DNI	N/A
S1	0Ω	N/A
P2	3.0pF	±0.05pF
R1-R4	DNI	N/A



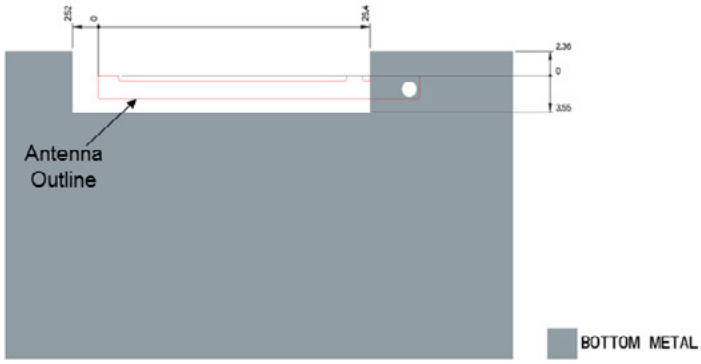
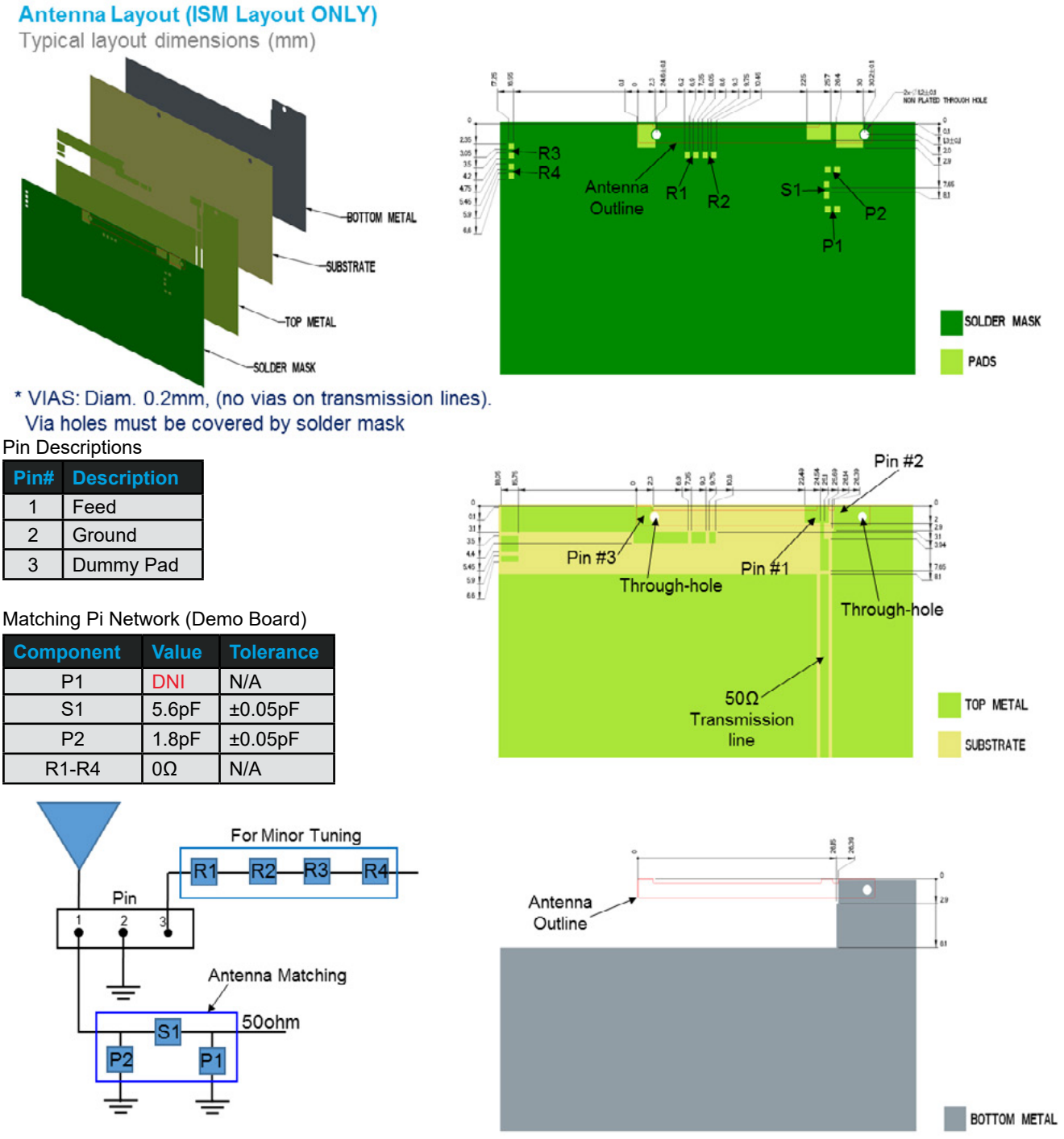


Figure 2 : 1002427 Antenna Footprints for GPS ONLY

Figure 3 below shows the 1002427 Antenna Footprints for ISM ONLY



Pin Descriptions

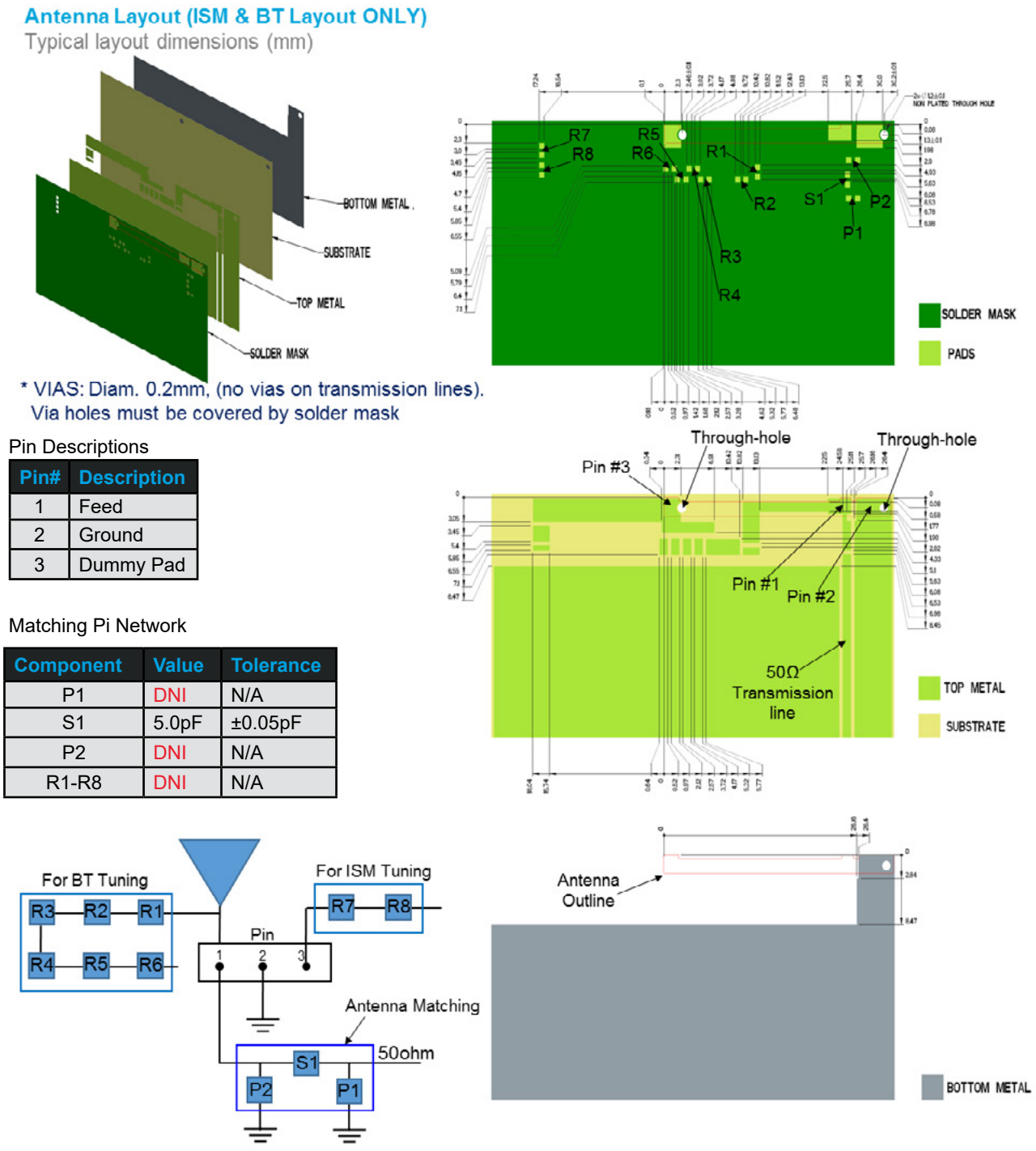
Pin#	Description
1	Feed
2	Ground
3	Dummy Pad

Matching Pi Network (Demo Board)

Component	Value	Tolerance
P1	DNI	N/A
S1	5.6pF	±0.05pF
P2	1.8pF	±0.05pF
R1-R4	0Ω	N/A

Figure 3 : 1002427 Antenna Footprints for ISM ONLY

Figure 4 below shows the 1002427 Antenna Footprints for ISM and BT



Pin Descriptions

Pin#	Description
1	Feed
2	Ground
3	Dummy Pad

Matching Pi Network

Component	Value	Tolerance
P1	DNI	N/A
S1	5.0pF	±0.05pF
P2	DNI	N/A
R1-R8	DNI	N/A

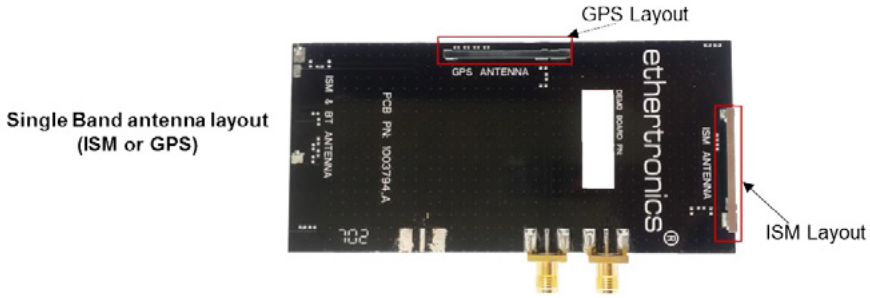
Figure 4 : 1002427 Antenna Footprints for ISM & BT Dual Band

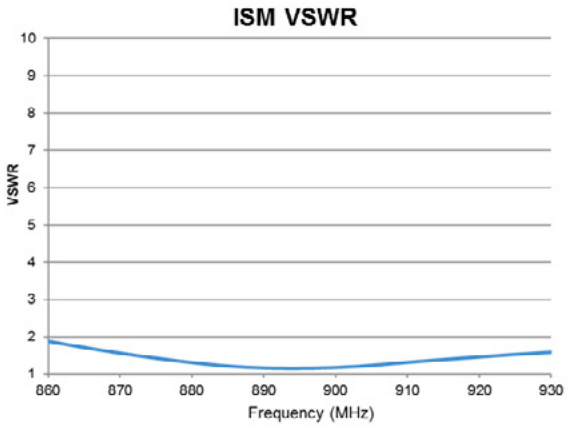
Typical Measured Data

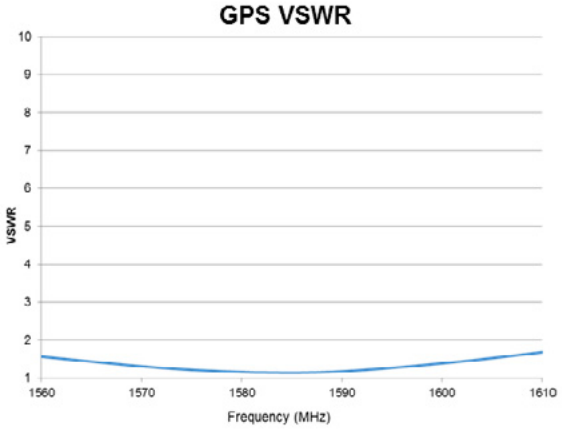
VSWR , Efficiency and Radiation Pattern

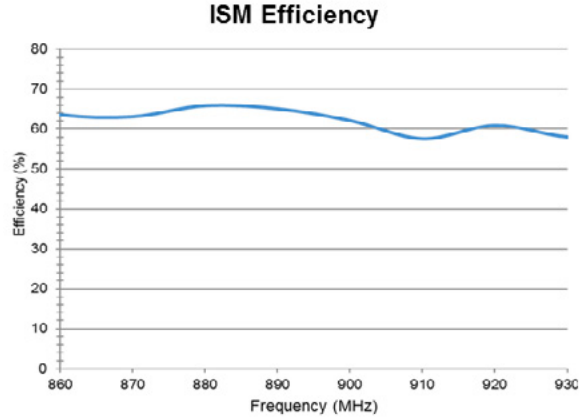
Figure 5 below shows the 1002427 Antenna Typical Performance for GPS ONLY and ISM ONLY

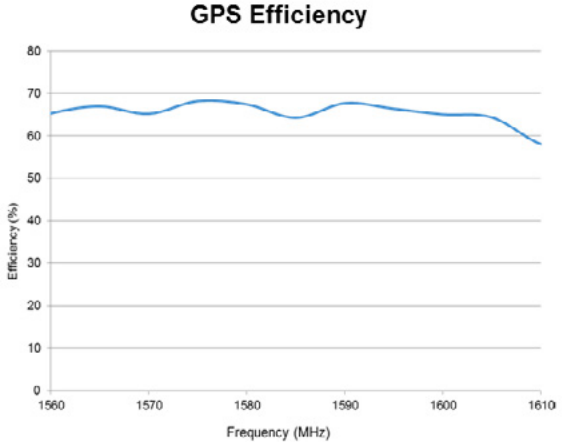
Figure 6 below shows the 1002427 Antenna Typical Performance for ISM & BT Dual Band











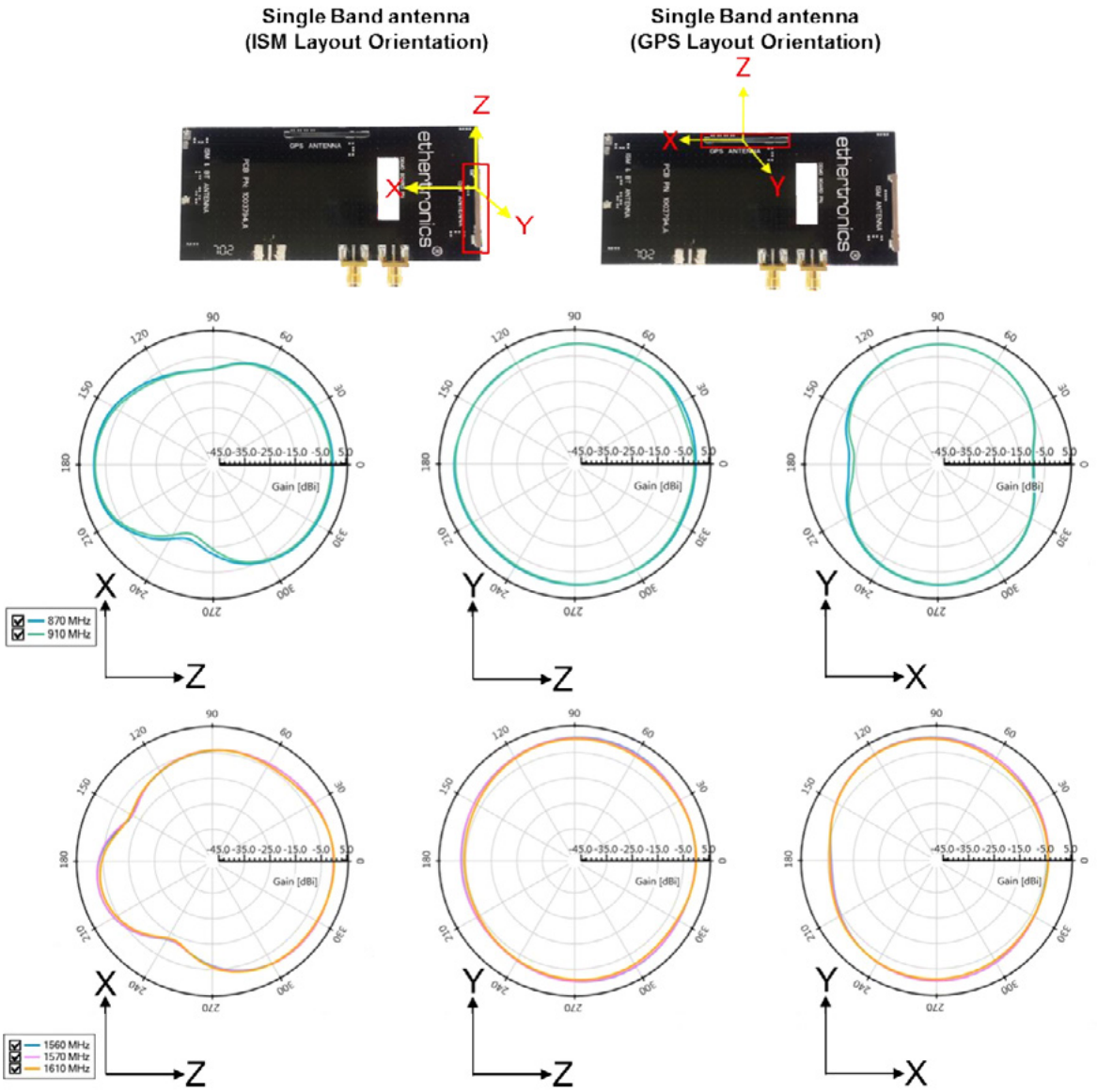
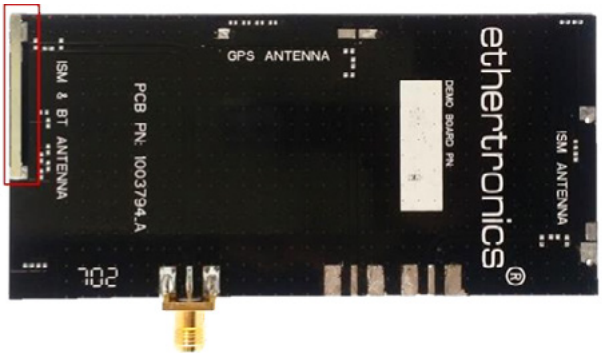
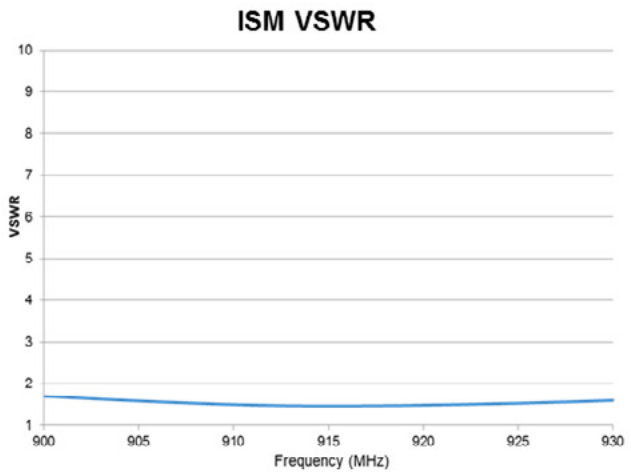


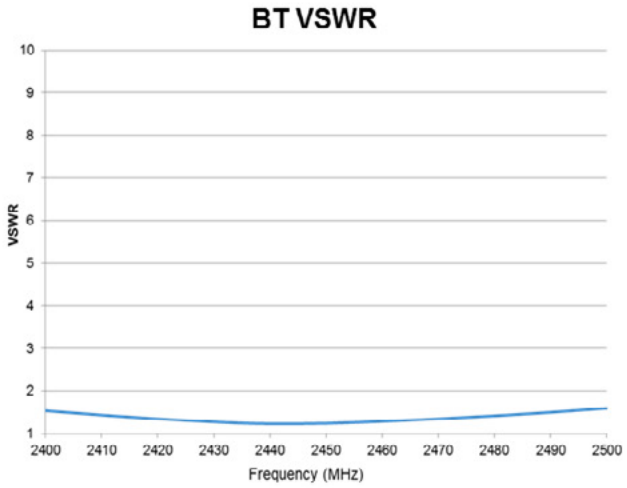
Figure 5 : 1002427 Antenna Typical Performance for GPS ONLY and ISM ONLY

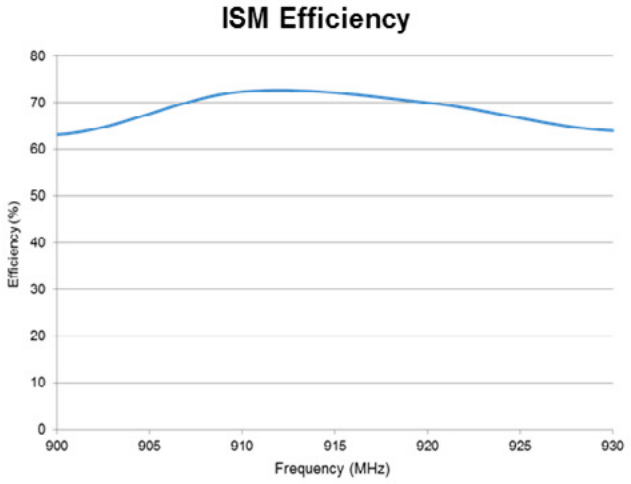
Dual Band antenna layout (ISM and BT)

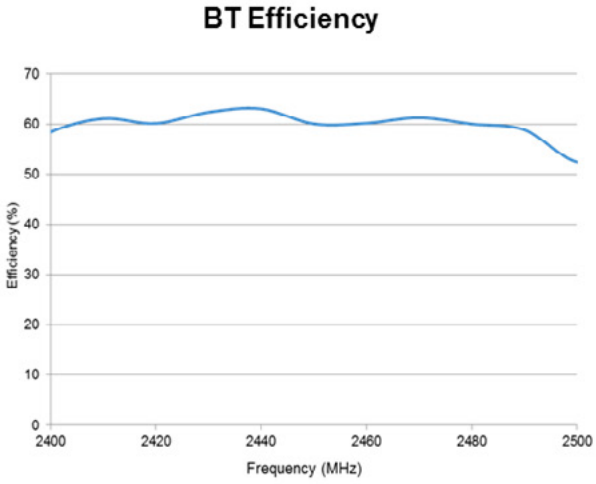
ISM & BT Layout











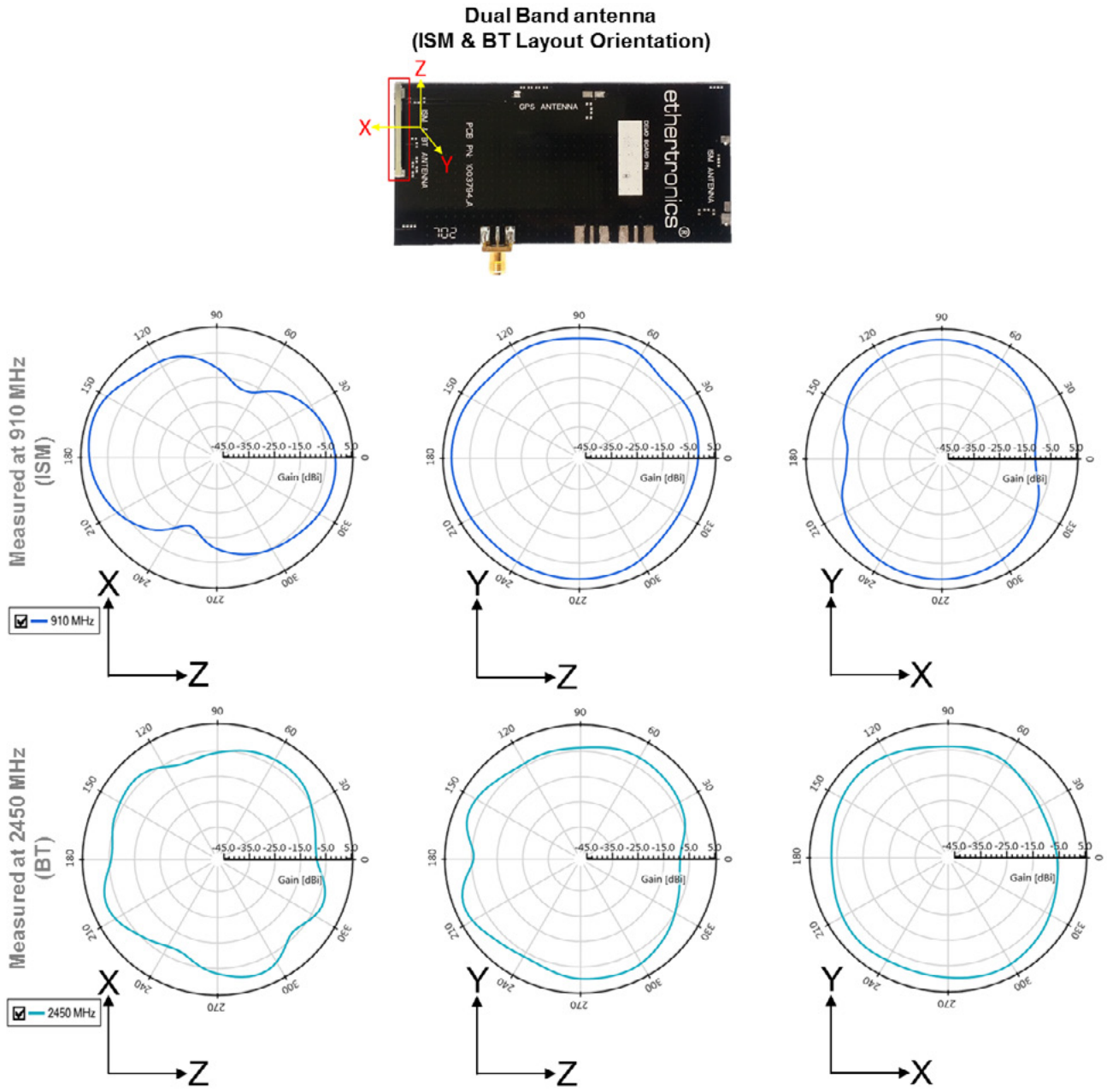


Figure 6 : 1002427 Antenna Typical Performance for ISM & BT Dual Band

Antenna Placement Guidelines on PCB

The 1002427 antenna can be mounted onto any PCB using KYOCERA AVX's recommended footprint layout and ground layout with proper PCB size. For the purposes of the Design Guidelines section 3 of this document, the 1002427 has been mounted KYOCERA AVX's PCB demo board (1002427-01) which represent the approximated size of a standard end device PCB.

- The antenna should always be placed along the edge the board unless this location is not available on your design.
- The 1002427 requires a ground clearance on the left side of the tuning pad element. It is recommended to keep a ground cleared area greater than 8mm beside the tuning pad.
- The ideal long edge length “L” should be defined by 90mm ≤ L ≤ 130mm, and the recommended antenna location for each antenna layout are described below :
    - GPS
        - Center of long edge
    - ISM Single Band
        - Short edge of board
    - ISM & BT Dual Band
        - Short edge of board

Figure 7 shows 1002427 typical landing location for each antenna layout.

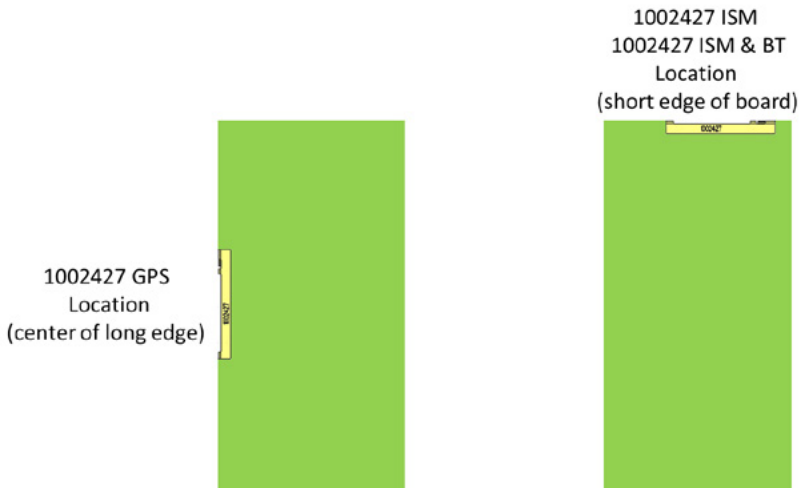


Figure 7 : 1002427 typical landing location for each antenna layout

Antenna Tuning Guidelines

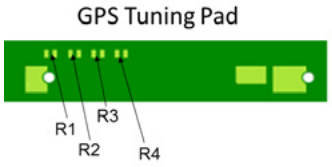
In real application environment, variation of the antenna resonating frequency may be caused by a the following effects: Different antenna locations, PCB board variations (including PCB size and PCB thickness), components and shield cans located close to the antenna, and outside cover… To offset the detuning effect, There are four methods can be applied to the board To solve the above effects, there are four methods can be applied :

- Major tuning through the tuning pad printed on the PCB
- Minor tuning using the matching circuit
- Change of the antenna location
- Change of the PCB length or width

Major tuning through the tuning pad printed on the PCB

The antenna tuning pads can be considered as an extended portion of the antenna that allow the antenna frequency resonance to be shifted lower or higher by adding or removing 0ohm resistors.. Basically, adding 0 ohm resistors is equivalent to increasing the antenna physical length and removing 0ohm resistors is equivalent to reducing the antenna physical length. The advantage of using tuning pads is to be able to adjust the antenna performances using components instead of re-spinning the PCB. In mass production, the 0 ohm resistors can be replaced by a trace.

Figure 8 shows 1002427 GPS ONLY Tuning Pad Configurations
Figure 9 shows 1002427 ISM ONLY Tuning Pad Configurations
Figure 10 shows 1002427 ISM & BT Dual Band Tuning Pad Configurations



GPS Tuning Pad Length	R1	R2	R3	R4
GPS Pad Length 1 (Default)	DNI	DNI	DNI	DNI
GPS Pad Length 2	0Ω	DNI	DNI	DNI
GPS Pad Length 3	0Ω	0Ω	DNI	DNI
GPS Pad Length 4	0Ω	0Ω	0Ω	DNI
GPS Pad Length 5	0Ω	0Ω	0Ω	0Ω

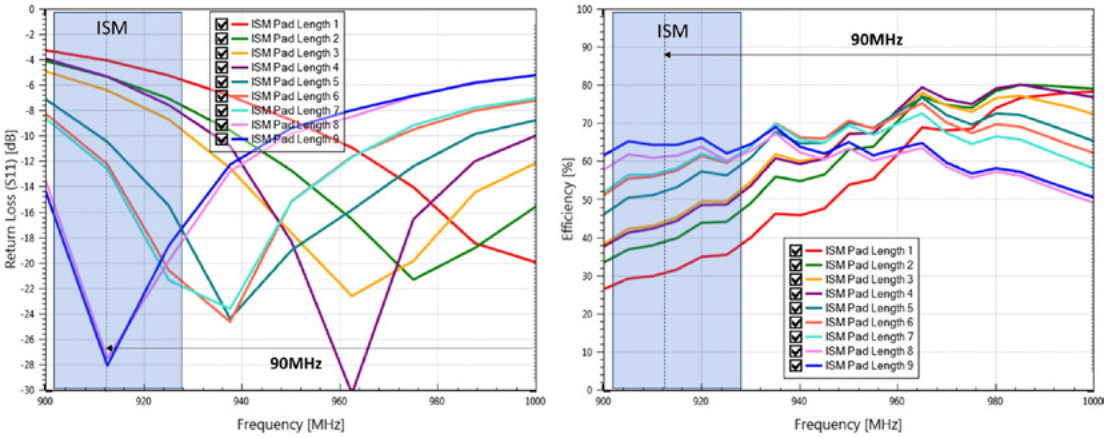
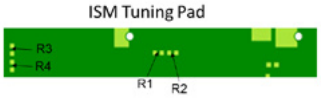


Figure 8 : 1002427 GPS ONLY Tuning Pad Configurations



ISM Tuning Pad Length	R1	R2	R3	R4
GPS Pad Length 1 (Default)	DNI	DNI	DNI	DNI
GPS Pad Length 2	0Ω	DNI	DNI	DNI
GPS Pad Length 3	DNI	DNI	0Ω	DNI
GPS Pad Length 4	0Ω	0Ω	DNI	DNI
GPS Pad Length 5	0Ω	DNI	0Ω	DNI
GPS Pad Length 6	0Ω	0Ω	0Ω	DNI
GPS Pad Length 7	DNI	DNI	0Ω	0Ω
GPS Pad Length 8	0Ω	DNI	0Ω	0Ω
GPS Pad Length 9	0Ω	0Ω	0Ω	0Ω

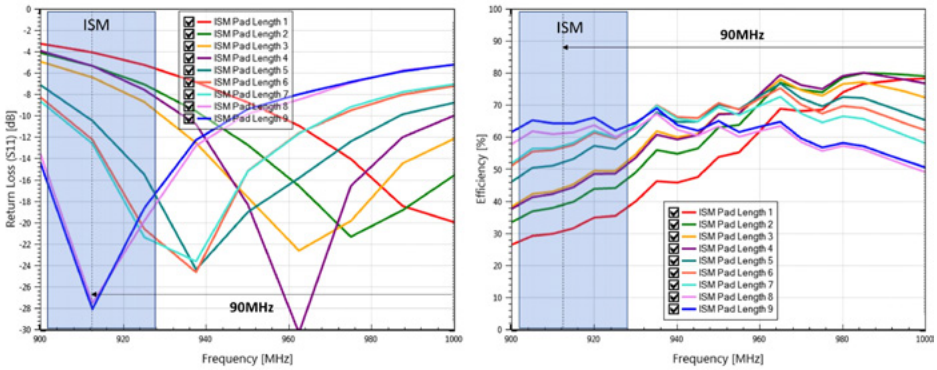
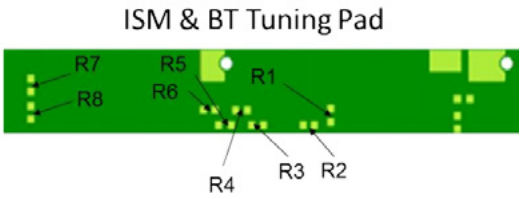


Figure 9 : 1002427 ISM ONLY Tuning Pad Configurations



ISM & BT Tuning Pad Length	R1 (BT Tuning)	R1 (BT Tuning)	R1 (BT Tuning)	R1 (BT Tuning)
GPS Pad Length 1 (Default)	DNI	DNI	DNI	DNI
GPS Pad Length 2	DNI	DNI	0Ω	DNI
GPS Pad Length 3	DNI	DNI	0Ω	0Ω
GPS Pad Length 4	DNI	0Ω	DNI	DNI
GPS Pad Length 5	0Ω	DNI	DNI	DNI

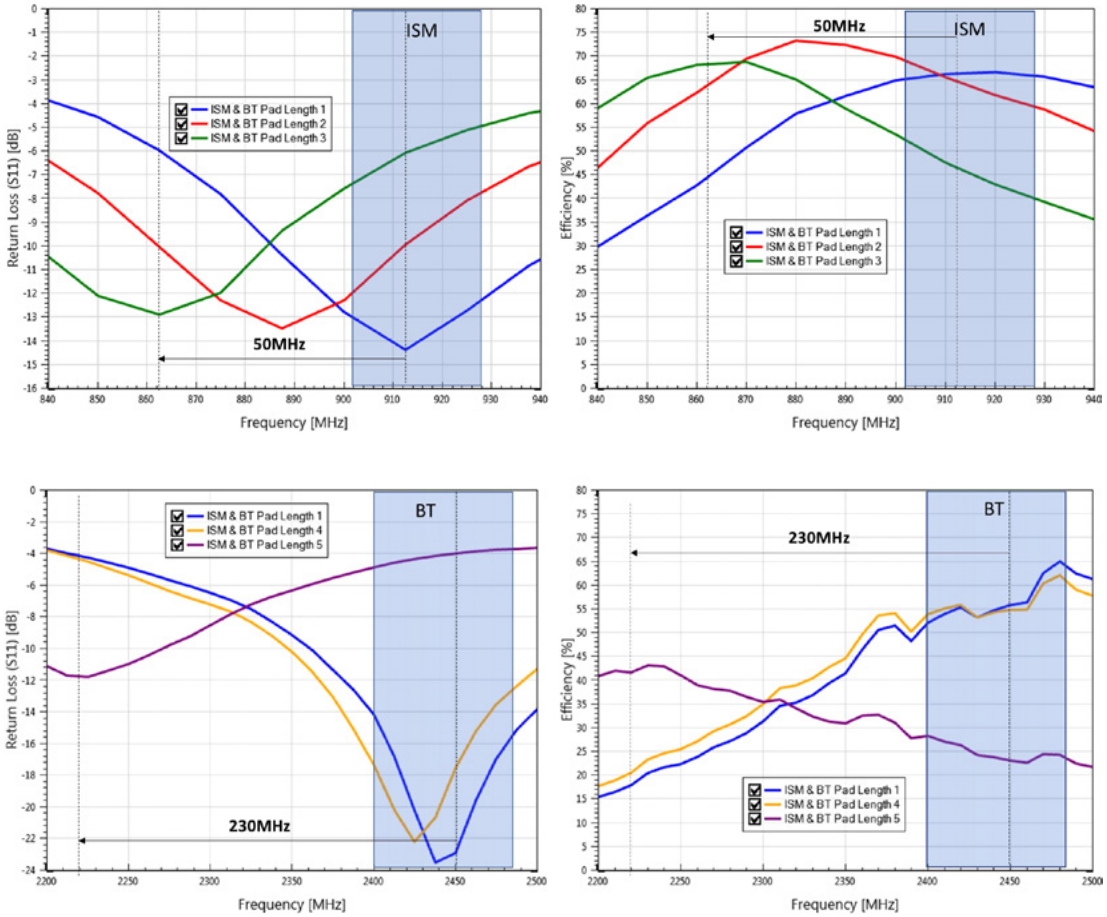
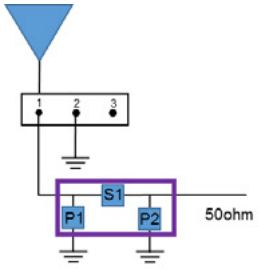


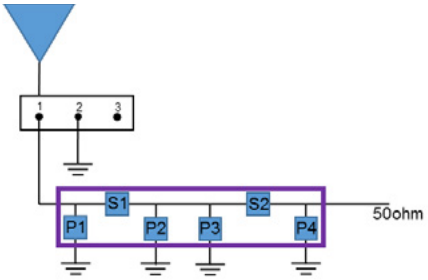
Figure 10 : 1002427 ISM & BT Dual Band Tuning Pad Configurations

Minor tuning through matching circuit

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 design (GPS ONLY layout and ISM ONLY layout), if the frequency is slightly off the required band, one “pi” type of matching is enough to tuned it back, in general, two matching components are enough (Using P1&S1 or S1&P2 from network below).



For the dual-band design (ISM & BT Dual Band layout), 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.



Change of the antenna location

The antenna location is one of the most important factor that will impact the antenna performances. Antenna performance will immediately be changed when the antenna location is changed. Here are some studies to look each antenna layout performance in four different locations.

- Location 1 : Antenna is placed at the right side of the short edge
- Location 2 : Antenna is placed in the middle of the long edge
- Location 3 : Antenna is placed at the left side of the long edge
- Location 4 : Antenna is placed at the right side of the long edge

For each location, the impact on the antenna performances have been measured given each available layout

Figure 11 shows the four different antenna test locations
Figure 12 shows 1002427 GPS ONLY with Four Locations
Figure 13 shows 1002427 ISM ONLY with Four Locations
Figure 14 shows 1002427 ISM & BT Dual Band with Four Locations

Figure 11 : 1002427 Four Test Locations

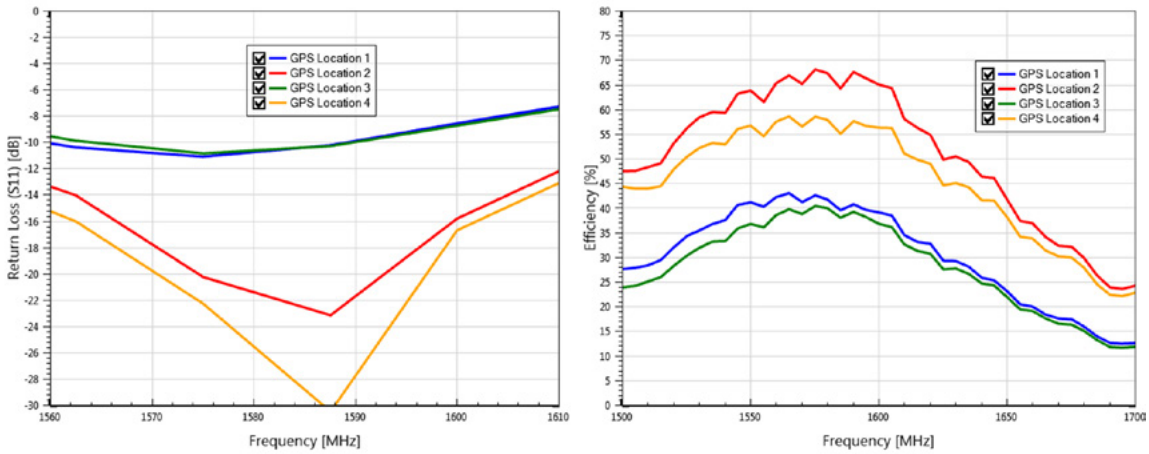


Figure 12 : 1002427 GPS ONLY with Four Locations

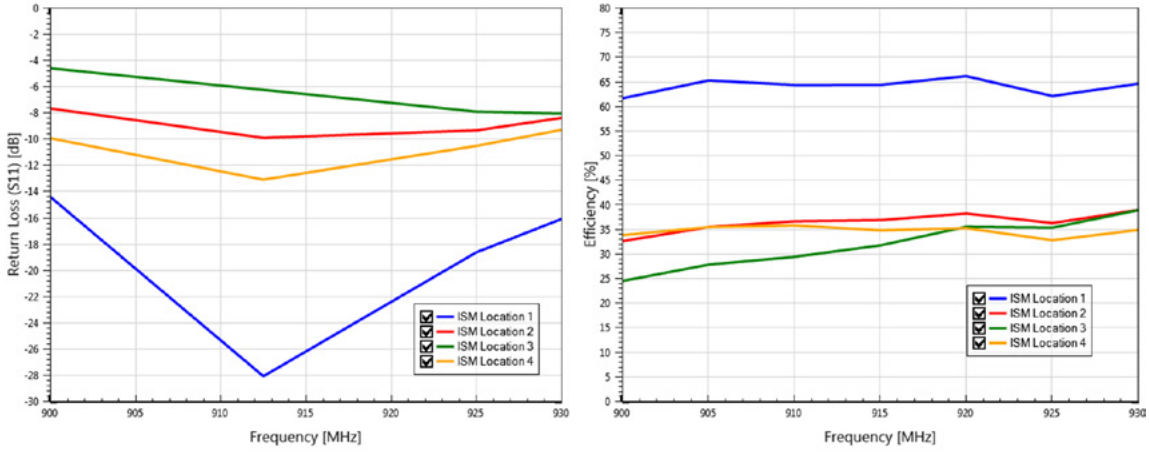


Figure 13 : 1002427 ISM ONLY with Four Locations

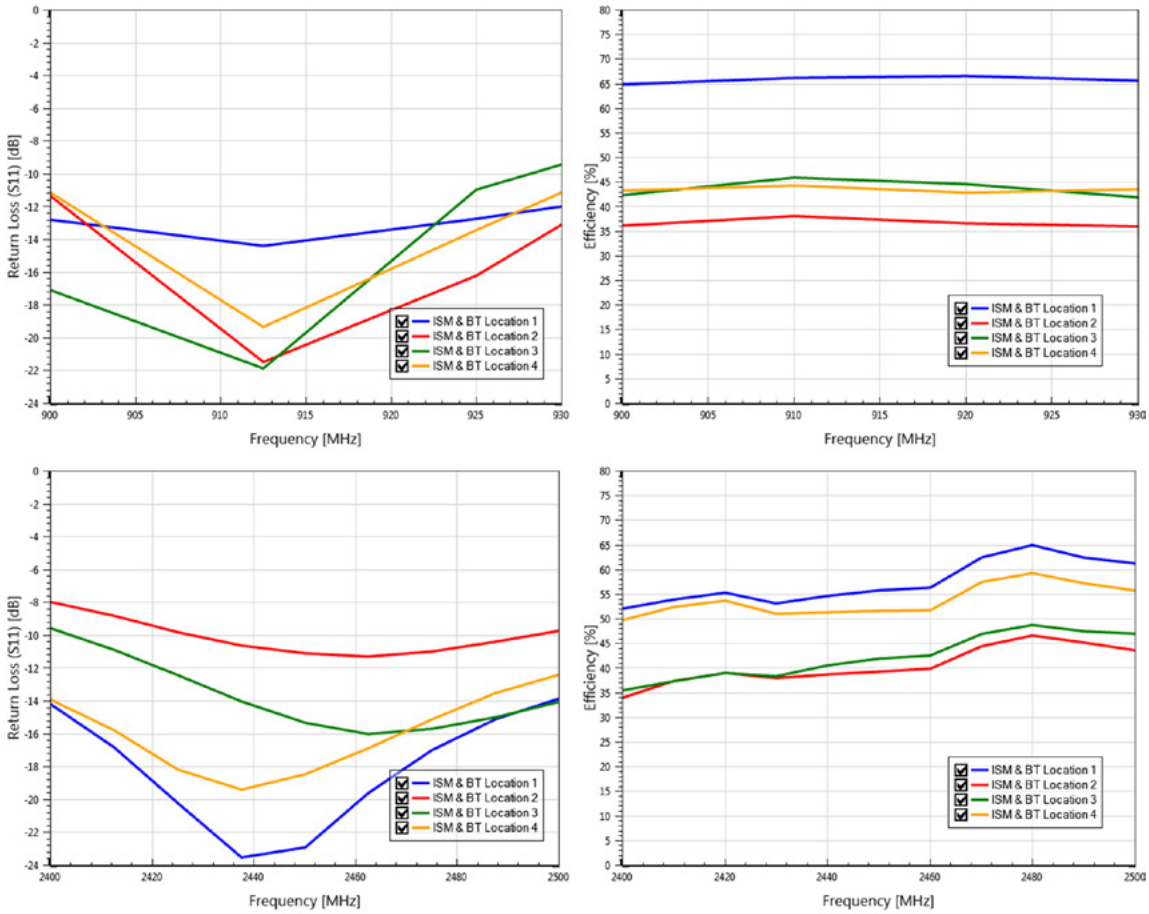


Figure 14 : 1002427 ISM & BT Dual Band with Four Locations

Change of the PCB length or width

The board size is another critical factor that impact antenna performances. The ideal PCB width for the 1002427 GPS antenna layout (placed on the long edge), is around 110mm . The ideal PCB length for the ISM and ISM & BT layout, where the antenna is placed on the short edge is around 110mm as well. Depending of the applications, any PCB length and width modification may degrade performances. Nevertheless, the antenna performance can be improved by modifying the tuning pad and optimizing the matching components accordingly as explained in previous sections. Below are studies to show the antenna performance variation based on the PCB width or PCB length change.

Figure 15 shows the PCB Width or Length Change
Figure 16 shows 1002427 GPS ONLY performance varies with different PCB width
Figure 17 shows 1002427 ISM ONLY performance varies with different PCB length
Figure 18 shows 1002427 ISM & BT Dual Band performance varies with different PCB length

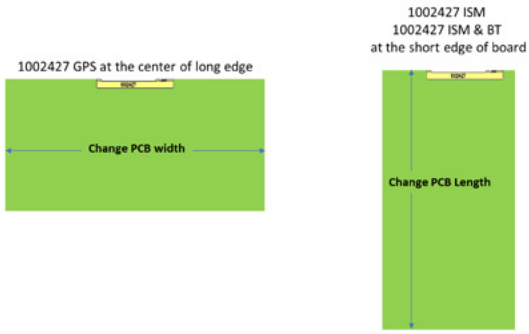


Figure 15 : PCB Width or Length Change

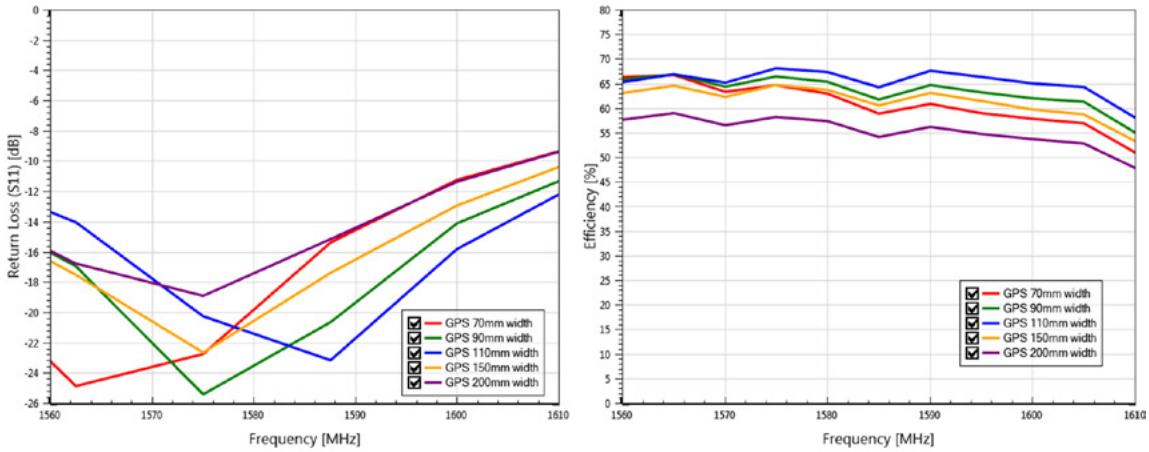


Figure 16 : 1002427 GPS ONLY performance varies with different PCB width

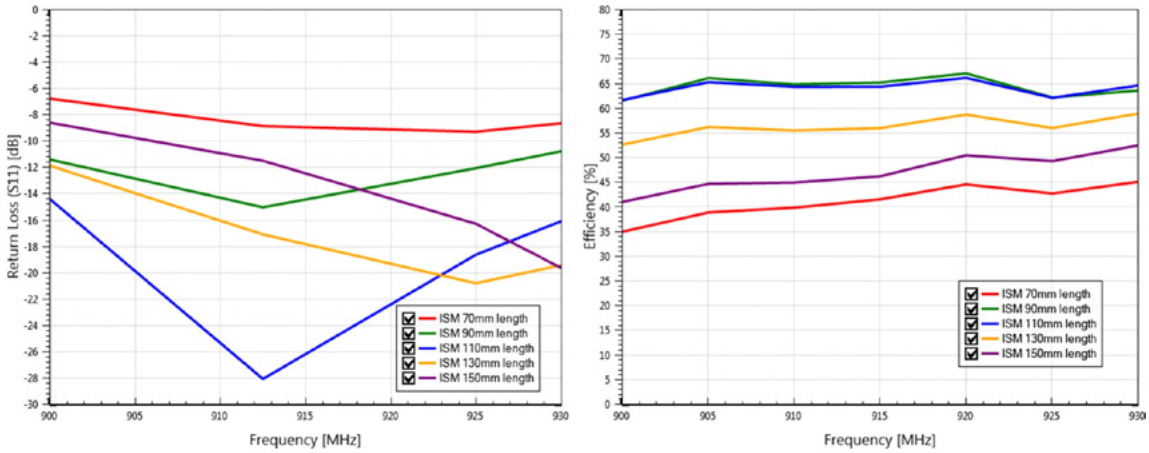
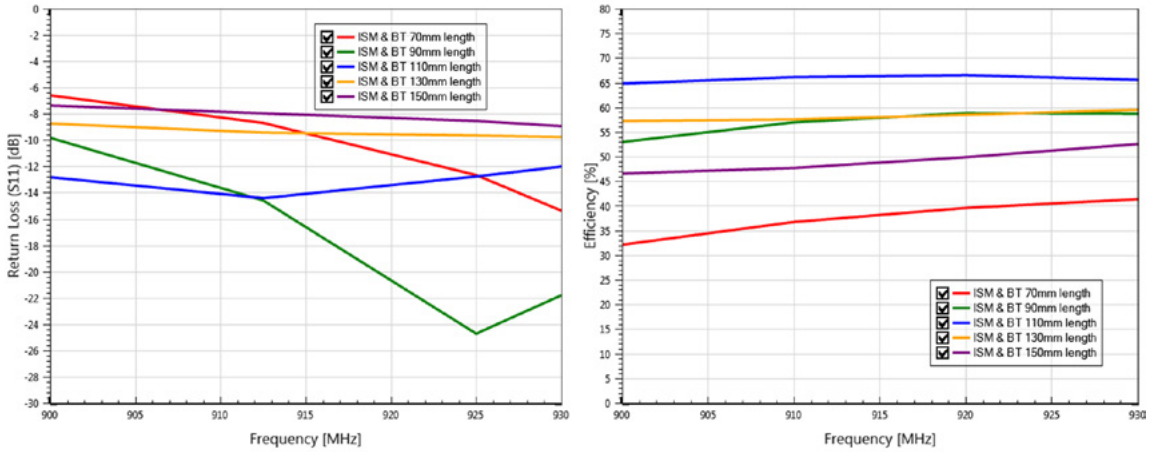


Figure 17 : 1002427 ISM ONLY performance varies with different PCB length



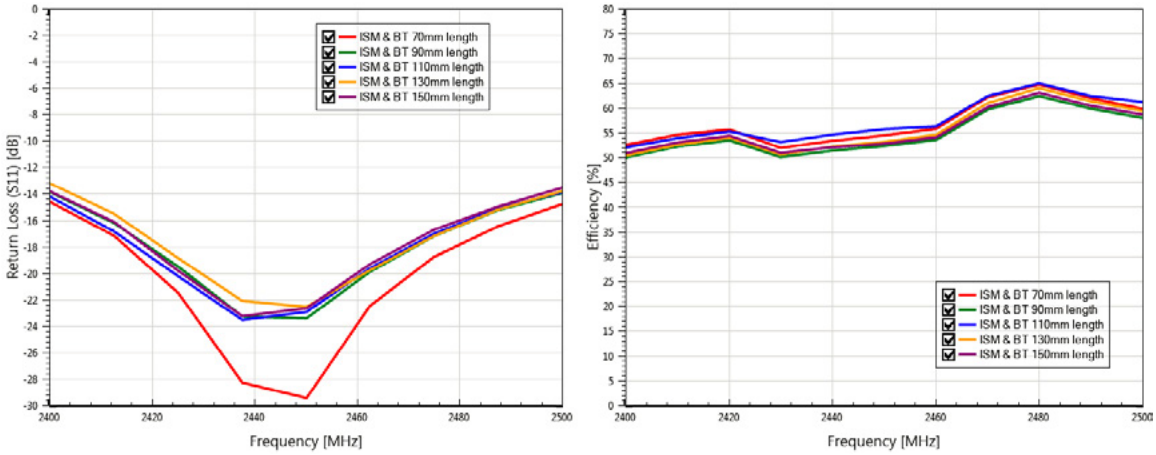


Figure 18 : 1002427 ISM & BT performance varies with different PCB length

Material Specifications

Item	Material
Metal Element	SUS 304
Contact Finish	Ni and Sn Plating

Manufacturing and Assembly Guidelines

KYOCERA AVX's 1002427 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 determined by KYOCERA AVX, based on successful manufacturing processes.

These antennas 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:

- KYOCERA AVX Standard 1002427 antennas ship in tape and reel.
- 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 antennas are not moisture sensitive and the ceramic 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 solderability, KYOCERA AVX antennas are dry-packed.

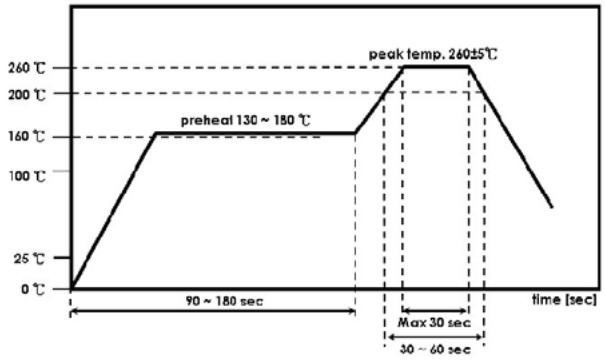
Paste Stencil Recommendation

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 deionized 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