APPLICATION NOTES

AN-GNSS-MP-20180522

KYOCERA AVX

GPS/GLONASS/Beidou/Galileo PCB Antenna (On/Off Ground) 1001011

1.561, 1.575, 1.603 GHz

Applications:

Tracking
Smart Home
Agriculture
Automotive
Healthcare
Digital Signage
Wearables
Industrial Devices

GPS/GLONASS / Beidou / Galileo PCB Antenna (On/Off Ground) 1001011

Table of Contents

GPS/GLONASS / Beidou / Galileo PCB Antenna (On/Off Ground) 1001011

1.561, 1.575, 1.603 GHz

Purpose

This document provides information for incorporating KYOCERA AVX' standard embedded GPS/GLONASS/Beidou/Galileo PCB antenna 1001011 into wireless products. Specifications, design recommendations, board layout, packaging and manufacturing recommendations are included.

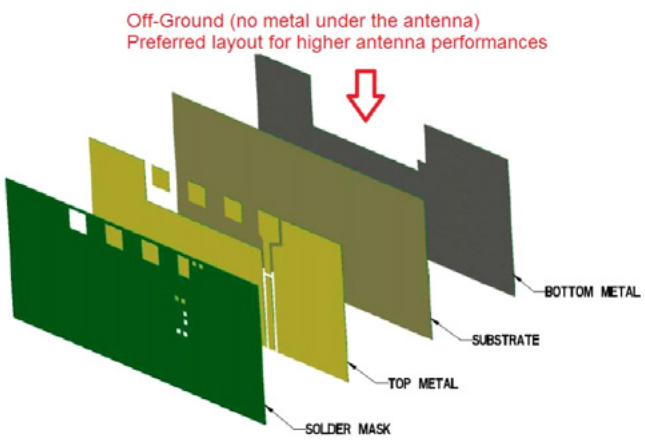
Overview

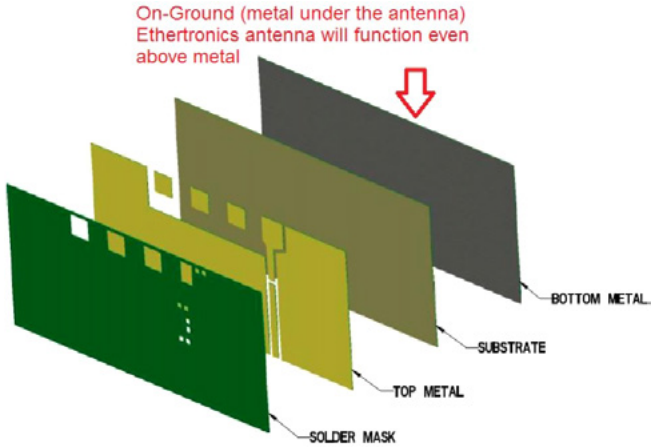
Product Selection Guide

Feature	Advantage	Benefits	Typical Deliverable Size
1001011	GNSS Bands: • Beidou: 1.559-1.563 GHz • GPS: 1.575 GHz • Galileo: 1.559-1.591 GHz • Glonass: 1.593-1.610 GHz	• Capable for Off-Ground and On-Ground (over metal) environments. • 1.561,1.575 & 1.603 GHz • Small dimension and High Performance • Flexible antenna placement	• SMT mountable antenna assembly • 22.0 x 3.2 x 3.3 mm
1001011-1	• Demo board tuned for GNSS band	• Off ground with 5.70mm ground clearance	• 72 x 50 mm PCB with SMA connector

Difference between On Ground application

Across this document, the terms "Off-Ground" and "On-Ground" will be mentioned. The Off-Ground designation is referring to an antenna layout that as all the metal removed from the different PCB layers (or even on the device stack up) under the antenna. Most on board antennas will be designed with this requirement as it is the preferred layout for optimum antenna performances. The On-Ground designation means that the antenna layout maintain ground under the antenna. It might be a ground layers on the PCB, or any other metallic element placed under the antenna area on the PCB.





GPS/GLONASS / Beidou / Galileo PCB Antenna (On/Off Ground) 1001011

1.561, 1.575, 1.603 GHz

GNSS PCB Embedded Antenna Features and Benefits Summary

Features	Benefits
PCB Embedded structure with Small Form Factor & Ground clearance Requirements	• Flexibility in antenna placement • Robust 3.2mm thickness PCB material • Ability to source antenna only for direct placement on customer PCB
High Performance Embedded Solution	• Greater than 65% average efficiency cross all bands for off-Ground solution • Greater than 45% average efficiency cross all bands for On-Ground solution
Off Ground and On Ground solutions	• Enables flexibility in antenna placement within end device • Can be used within Tracking, Handhelds, Automotive, Wearable Devices.
Ground Cleared Solution	• Speed development time
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
Cost Effective & Rugged Design	• SMT, Pick and Place, Tape & Reel Packaging, Enable lower manufacturing costs.

Design Guidelines

Introduction

The Standard GNSS PCB embedded antenna 1001011 can be designed into many wireless product types. The following sections explain KYOCERA AVX’ recommended layouts to help the designer integrate the 1001011 antenna element into a product with optimum

Electrical Specifications

Typical Characteristics Measurements taken using the standard matching circuit on a 72 x 50 mm ground plane.

Frequency (GHz)	1.559 – 1.563	1.575	1.559 – 1.591	1.593 – 1.610
Mounting	Off Ground / On Ground			
GNSS Bands	Beidou	GPS	Galileo	Glonass
Peak Gain (dBi)	0.96 / -0.26	0.87 / -0.22	0.96 / -0.18	1.00 / -0.35
Efficiency (%)	72 / 47	71 / 46	70 / 45	69 / 41
Center Frequency f_o (GHz)	1.561	1.575	1.575	1.603
VSWR	1.5:1 / 2.5:1			
Feed Point Impedance	50 ohms unbalanced			

Mechanical Specifications

Ordering Part Number	1001011
Size (mm)	22.0 x 3.2 x 3.3
Mounting	Surface mounted to the PCB
Weight (grams)	0.45
Packaging	Tape & Reel
Demo Board	1001011-02

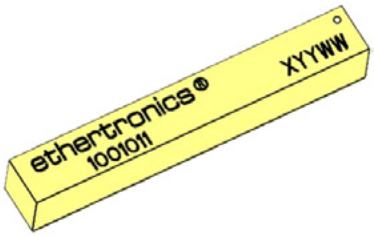
Antenna Dimension and Pad Layout

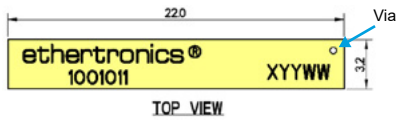
Figure 1 below shows the GNSS PCB Antenna Dimensions and Pad Layout for 1001011

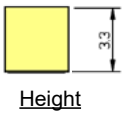
Antenna Dimensions

Typical antenna dimensions (mm)

Features	A (mm)	B (mm)	C (mm)
1001011	22.0 ± 0.3	3.2 ± 0.2	3.3 ± 0.3







Pin Descriptions

Pin	Description
1	Feed
2	Dummy Pad
3	Dummy Pad
4	Dummy Pad

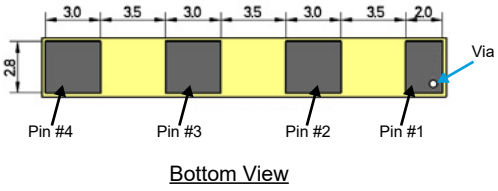


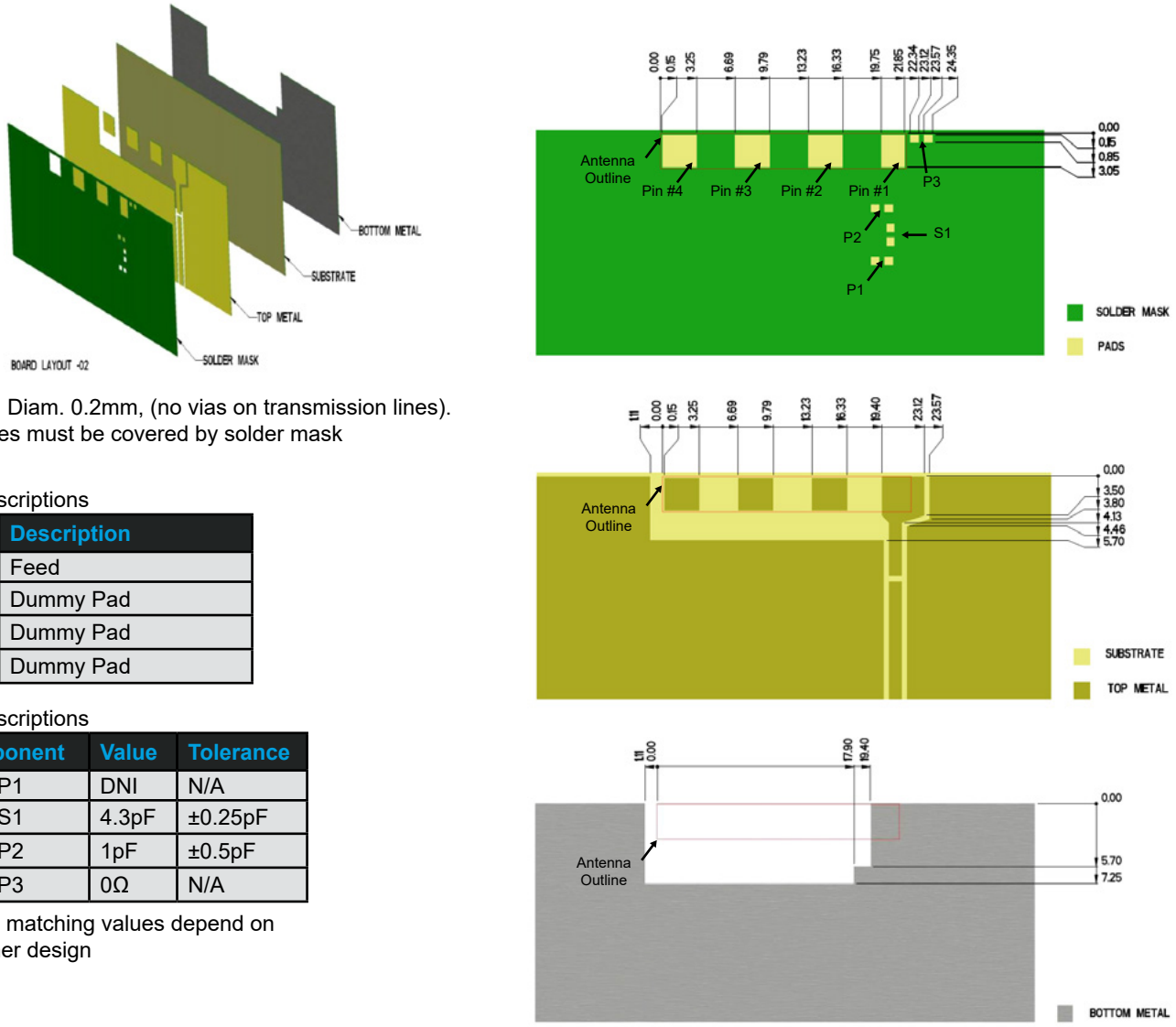
Figure 1 : Antenna Dimensions and Pad Layout for 1001011

Antenna Footprint Layout

Figure 2 below shows the GNSS PCB Antenna Layout for 1001011_Off-Ground

Figure 3 below shows the GNSS PCB Antenna Layout for 1001011_On-Ground

Figure 2 below shows the GNSS PCB Antenna Layout for 1001011_Off-Ground



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

Pin Descriptions

Pin#	Description
1	Feed
2	Dummy Pad
3	Dummy Pad
4	Dummy Pad

Pin Descriptions

Component	Value	Tolerance
P1	DNI	N/A
S1	4.3pF	±0.25pF
P2	1pF	±0.5pF
P3	0Ω	N/A

*Actual matching values depend on customer design

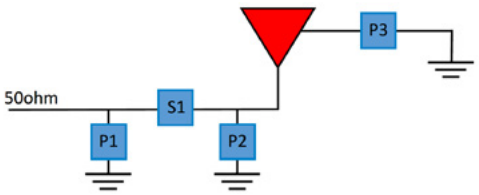
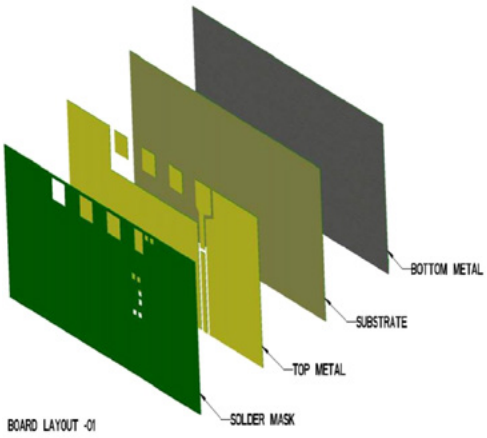
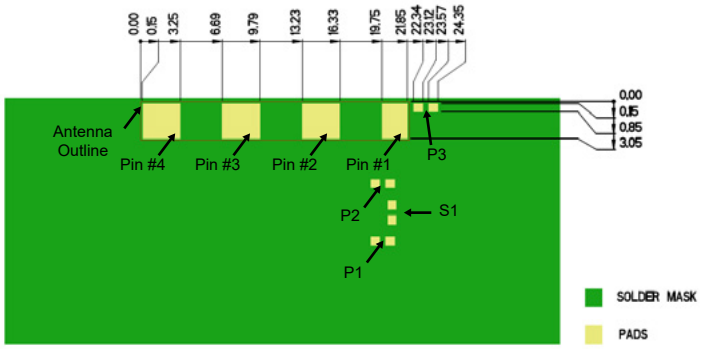
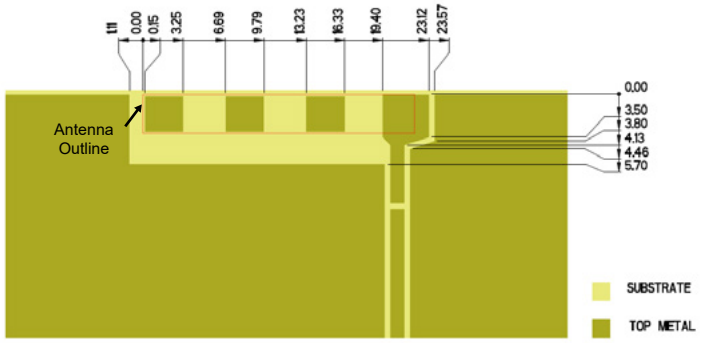


Figure 2: Antenna Layout for 1001011 Off-Ground

Figure 3 below shows the GNSS PCB Antenna Layout for 1001011_On-Ground







Pin Descriptions

Pin#	Description
1	Feed
2	Dummy Pad
3	Dummy Pad
4	Dummy Pad

Matching Pi Network (Demo Board)

Component	Value	Tolerance
P1	2.4pF	±0.1pF
S1	0Ω	N/A
P2	DNI	N/A
P3	0Ω	N/A

*Actual matching values depend on customer design

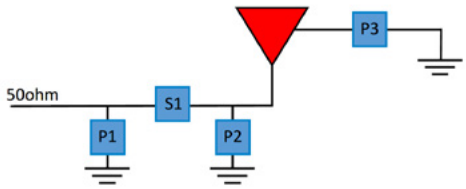


Figure 3: Antenna Layout for 1001011 On-Ground

Typical Measured Data

VSWR , Efficiency and Radiation Pattern

Below are the typical performances using KYOCERA AVX standard demo-board 1001011-01 and tuning specified above.

Figure 4 below shows the GNSS PCB Antenna Typical Performance for 1001011_Off-Ground

Figure 5 below shows the GNSS PCB Antenna Typical Performance for 1001011_On-Ground

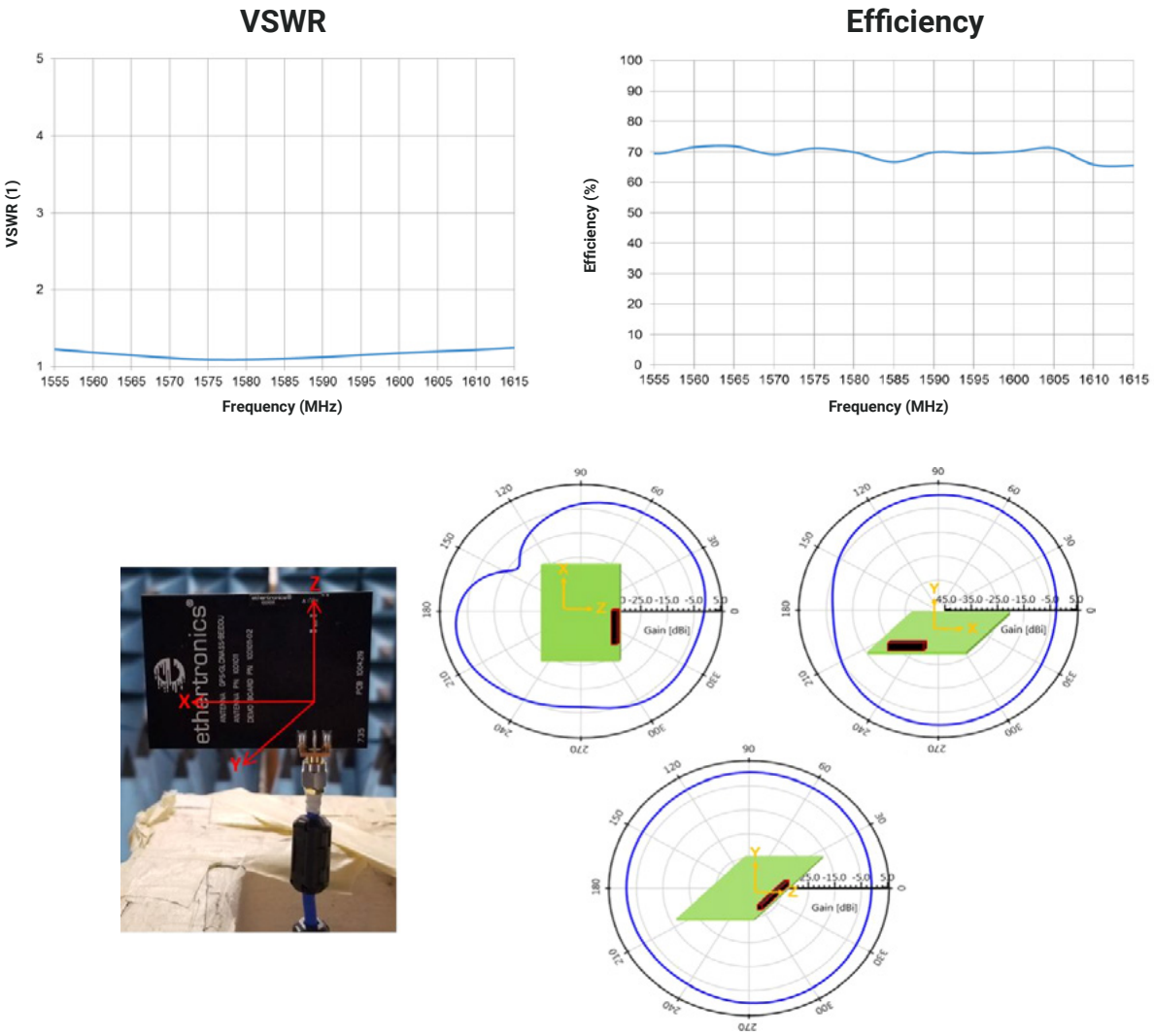


Figure 4: Typical Performance for 1001011_Off-Ground

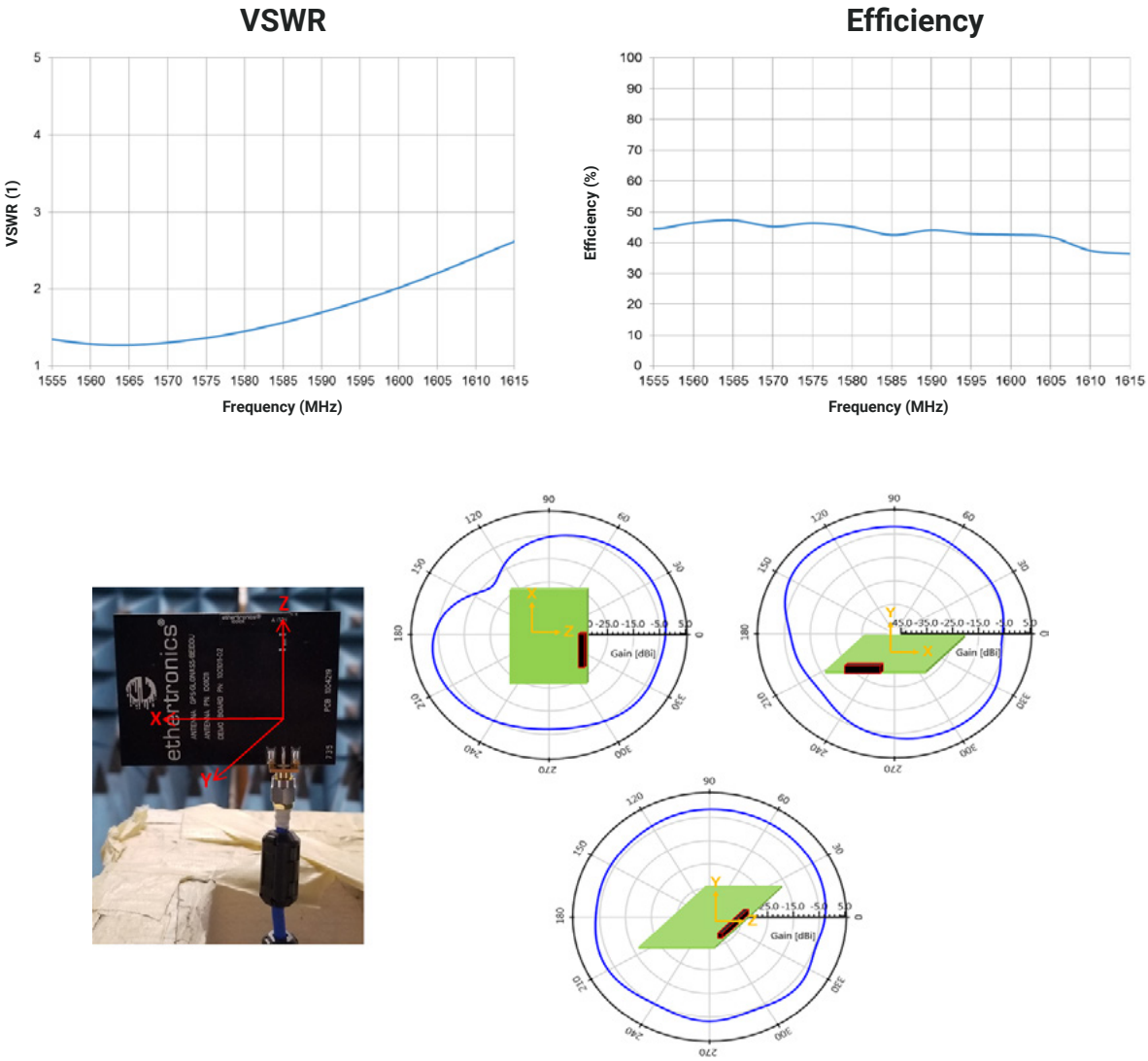


Figure 5: Typical Performance for 1001011_On-Ground

Antenna Placement Guidelines on PCB

The 1001011 antenna can be mounted on any PCB using KYOCERA AVX' recommended footprint layout and ground layout with proper PCB size. For the purposes of the Design Guidelines section, the standard KYOCERA AVX demo board 1001011-01 has been used. It reflects the standard sized PCB in most common end device.

Figure 6 shows below the optimal placement of the antenna on a PCB following the guidelines below:

- The antenna should always be placed along the edge of the board unless this location is not available on your design.
- The recommended antenna location is at the right side of the long edge with a certain distance "B" (recommend 5mm ≤ B ≤15mm) from the feed point to the edge of PCB. KYOCERA AVX demo board default location is X=72mm, Y=50mm, B=15mm. The edge where the antenna is located should be greater than 50mm, and more locations will be available based on the conditions below:
    - X ≥ 70mm,Y < 70mm
        - Only one location available (Location1). It can be increase to two, if the second antenna is place reversed on the opposite edge.

Antenna Placement Guidelines on PCB (continued)

- X ≥ 70mm, Y ≥ 70mm
  - Two locations available (Location1/Location2). It can be increased to four if the reverse antennas are placed on the two other edges.
- X ≥ 100mm, 70mm ≤ Y < 100mm
  - Three locations available (Location1/Location2/Location3), It can be increased to six if the reverse antennas are placed on the two other edges.

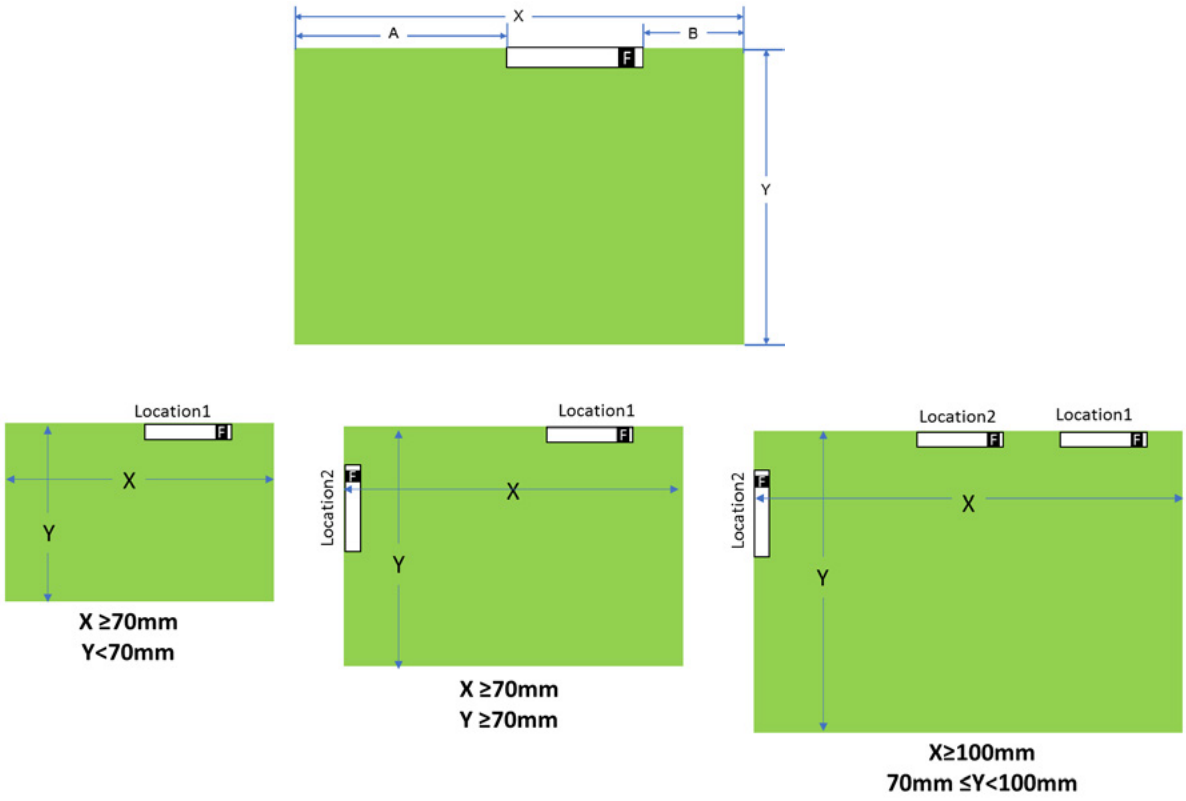


Figure 6: Optimal placement of the 1001011 on a PCB

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, components and shield cans located close to the antenna, and outside cover…
To solve the above effects, there are four methods can be applied :

- Tuning using the matching network
- Tuning the antenna ground clearance
- Changing the antenna location
- Adding ground extension elements

Tuning using the Matching Network

Performance can be improved by tuning the matching circuit. In general, adjusting the value of the matching components allow to control resonance shifting slightly and optimize the impedance within the required bands. Optimum matching values may vary with different boards transmission line design and antenna working environments. The following pages show the return loss and efficiency variation when using different matching value.

All the tests below are Based on 72 x 50 mm demo board . Antenna performance is detuned when changing the antenna ground clearance, the study below shows how to using matching tuning to optimize antenna performance.

Plot	Test State	Matching Condition	Ground Clearance used	Matching Tuning Value			
- - -	State 1	Before Matching	3.3 mm	P1=DNI	S1=4.3pF	P2=1pF	P3=0ohm
				• Default matching of 1001011-01 demo board			
———	State 1	After Matching	3.3 mm	P1=0.5pF	S1=8.2pF	P2=DNI	P3=0ohm
				• Improved tuning for state 1			
- - -	State 2	Before Matching	0 mm	P1=DNI	S1=4.3pF	P2=1pF	P3=0ohm
				• Default matching of 1001011-01 demo board			
———	State 2	After Matching	0 mm	P1=2.4pF	S1=0ohm	P2=DNI	P3=0ohm
				• Improved tuning for state 2			

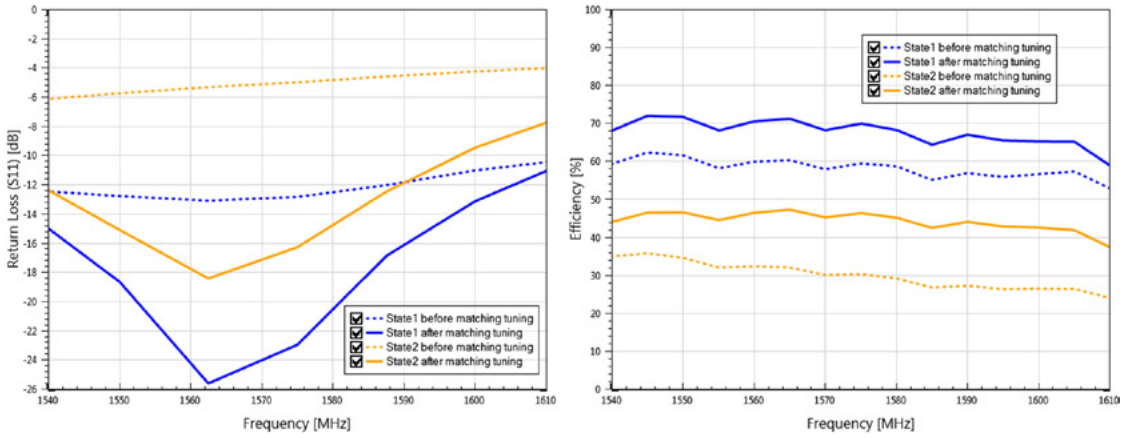
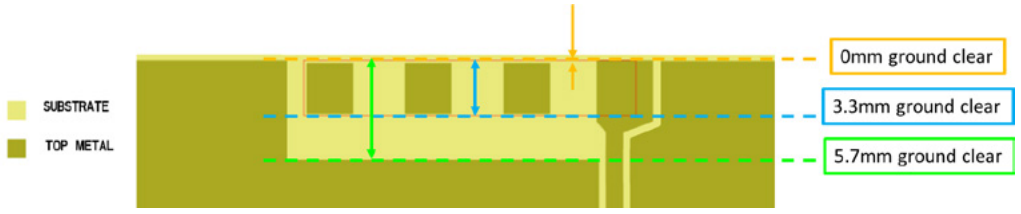


Figure 7: Matching Tuning adjustment for 1001011

Tuning the antenna ground clearance

Ground clearance is one of the most critical factor for antenna performance. Up to 2dB difference can be observed from antenna off ground to on ground condition. Increasing the ground clearance will result in wider antenna frequency bandwidth and higher efficiency data. Figure 8 below illustrates the ground clearance changes options for 1001011



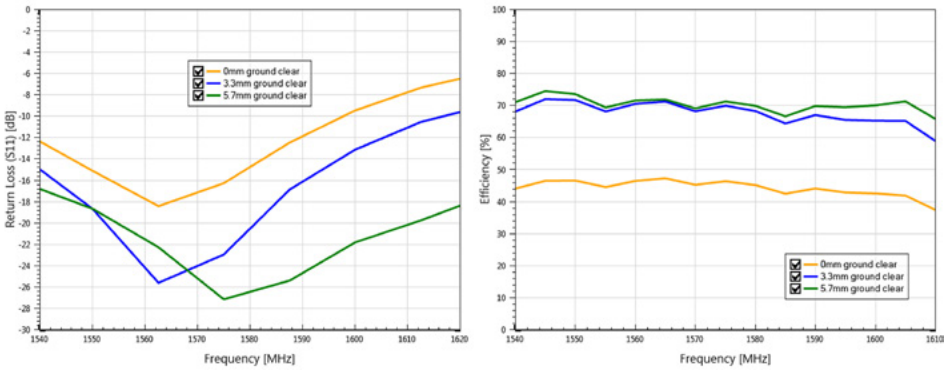


Figure 8: Ground clearance changes for 1001011

Changing the antenna location

Antenna performances will be impacted by the antenna location even if the PCB size remains the same.

Based on a 70mm x 50mm PCB, we considered 5 different locations :

- Location 1: right side of the long edge - which performs the best
- Location 2: center of the long edge
- Location 3: right side of the short edge
- Location 4: left side of the long edge
- Location 5: center of the short edge - which performs the worst (the performance could be improved even better than Location 4 by increasing the short edge length)

Figure 9 below illustrates the 1001011 antenna on different board locations

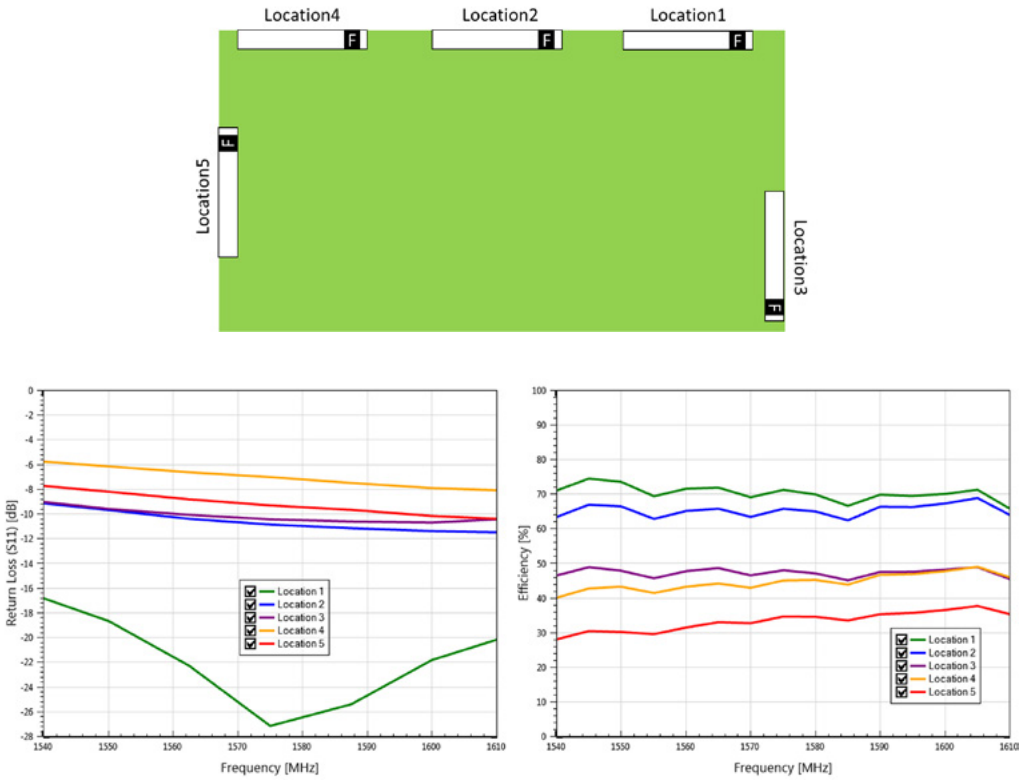


Figure 9: 1001011 antenna on different board locations

Adding ground extension element

In many cases, the ground length available next to the antenna location might be limited by the products ID design and PCB layout, and all the preferred locations might be occupied by other components. In this case, adding a thin or small ground extension may provide important performance improvement. Below shows three cases which have a disadvantageous antenna location but still maintain good performance by adding a ground extension element.

- Case1: The antenna is at the left side of the PCB edge with a small ground extension connected by an inductor in series (the inductor can help enhance the ground length without adding a physical length on the ground extension)
- Case2: The antenna is at the short edge of the PCB with a thin ground extension (two directions are possible)
- Case3: The antenna is at the center of the longest edge with a thin ground extension added (two directions are possible)

Figure 10 below is 1001011 antenna performance improvement with ground extensions

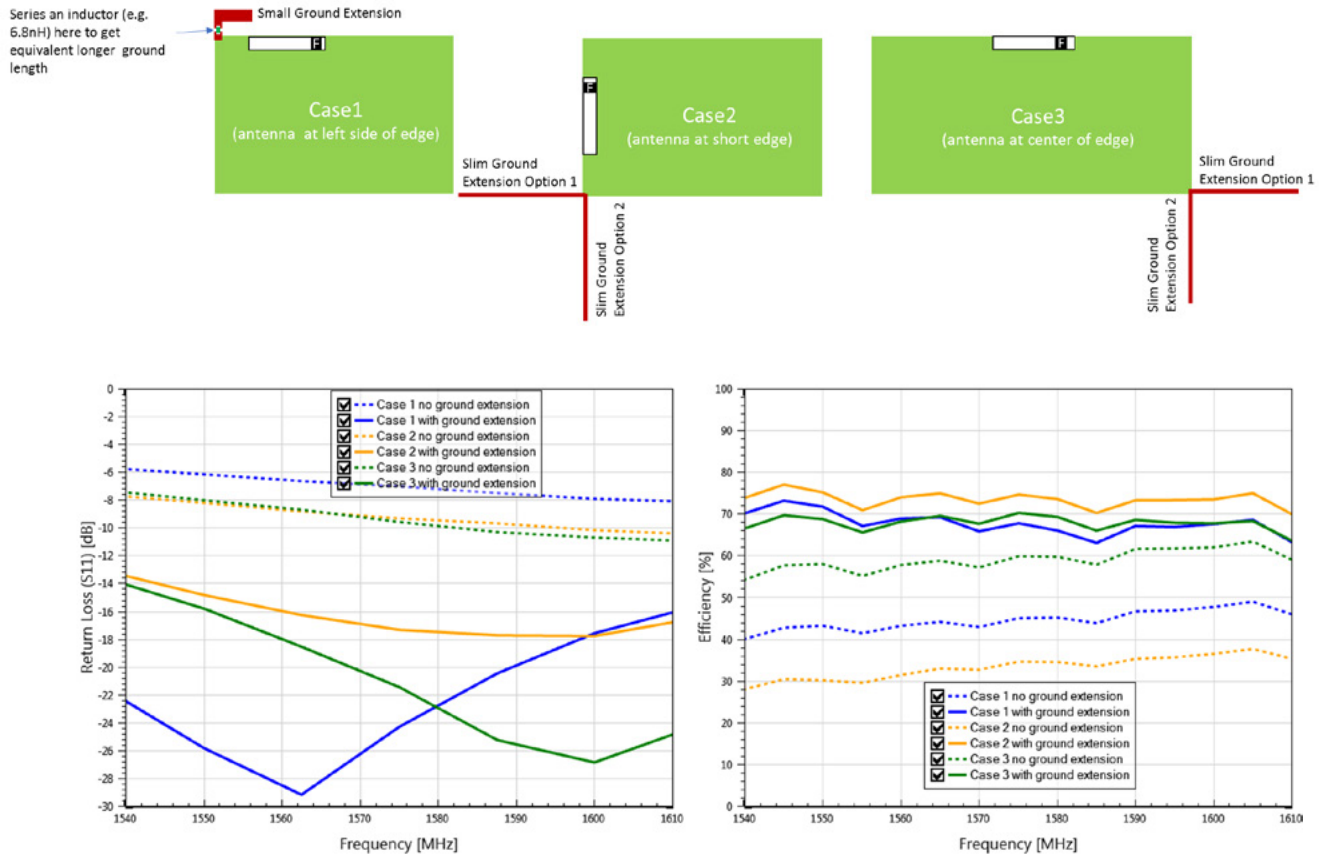


Figure 10: 1001011 antenna performance improvements with ground extensions

More Examples

1001011 tested performance with vary PCB sizes and vary antenna locations

PCB Size (mm)	1001011 Antenna Location	Average Efficiency (%)
72x72	right side of edge	61
100x50	right side of long edge	61
100x50	center of long edge	60
100x72	right side of short edge	64
100x100	right side of long edge	71
160x50	right side of long edge	63
160x72	right side of short edge	64
160x100	right side of long edge	75
160x100	right side of short edge	63

MIMO Application Examples

Figure 11 below shows typical MIMO configurations where two 1001011 antennas located:

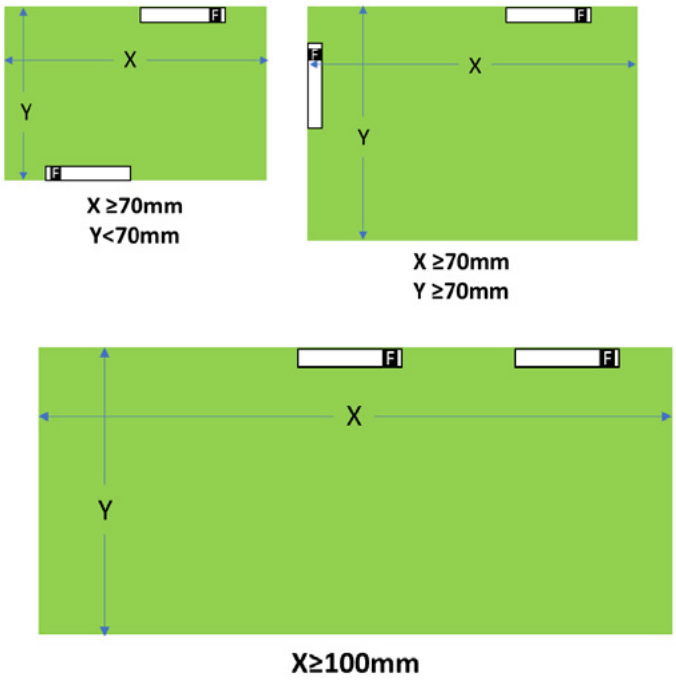


Figure 11: 1001011 Typical MIMO Configurations

Design Guidelines

Item	Material
Antenna Substrate	FR4
Contact Finish	Hot Air Solder Level (HASL) or Au

Manufacturing and Assembly Guidelines

KYOCERA AVX' Prestta™ 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 1001011 antennas ship in tape and reel..

KYOCERA AVX' 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.

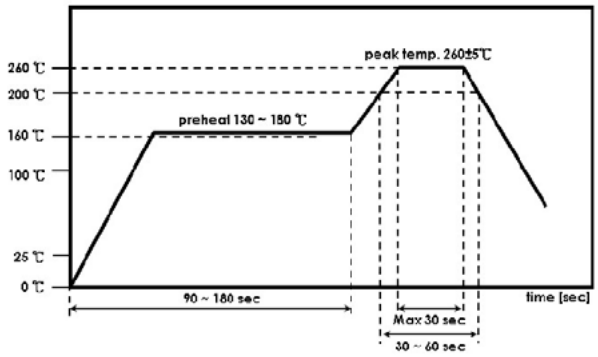
Paste Stencil Recommendation

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, Isotropy 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' 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