APPLICATION NOTES
AN-CERAMIC-MS-20181003

KYOCERA AVX

Savvi™ Embedded Ceramic Antennas

BT/GPS/WiMAX/ZigBee/ISM/WLAN802.11 a/b/g

Applications:

Cellular Handsets
Wireless Headsets
PC/mini

PCI Cards
PDAs Notebook PCs
Tablet PCs Printers

Industrial Devices
Navigation Equipment
Media Players

M2M

IMPORTANT INFORMATION/DISCLAIMER

All product specifications, statements, information and data (collectively, the "Information") in this datasheet or made available on the website are subject to change. The customer is responsible for checking and verifying the extent to which the Information contained in this publication is applicable to an order at the time the order is placed. All Information given herein is believed to be accurate and reliable, but it is presented without guarantee, warranty, or responsibility of any kind, expressed or implied.

Statements of suitability for certain applications are based on KYOCERA AVX's knowledge of typical operating conditions for such applications, but are not intended to constitute and KYOCERA AVX specifically disclaims any warranty concerning suitability for a specific customer application or use.

ANY USE OF PRODUCT OUTSIDE OF SPECIFICATIONS OR ANY STORAGE OR INSTALLATION INCONSISTENT WITH PRODUCT GUIDANCE VOIDS ANY WARRANTY.

The Information is intended for use only by customers who have the requisite experience and capability to determine the correct products for their application. Any technical advice inferred from this Information or otherwise provided by KYOCERA AVX with reference to the use of KYOCERA AVX's products is given without regard, and KYOCERA AVX assumes no obligation or liability for the advice given or results obtained.

Although KYOCERA AVX designs and manufactures its products to the most stringent quality and safety standards, given the current state of the art, isolated component failures may still occur. Accordingly, customer applications which require a high degree of reliability or safety should employ suitable designs or other safeguards (such as installation of protective circuitry or redundancies) in order to ensure that the failure of an electrical component does not result in a risk of personal injury or property damage.

Unless specifically agreed to in writing, KYOCERA AVX has not tested or certified its products, services or deliverables for use in high risk applications including medical life support, medical device, direct physical patient contact, water treatment, nuclear facilities, weapon systems, mass and air transportation control, flammable environments, or any other potentially life critical uses. Customer understands and agrees that KYOCERA AVX makes no assurances that the products, services or deliverables are suitable for any high-risk uses. Under no circumstances does KYOCERA AVX warrant or guarantee suitability for any customer design or manufacturing process.

Although all product–related warnings, cautions and notes must be observed, the customer should not assume that all safety measures are indicted or that other measures may not be required.

Savvi™ Embedded Ceramic Antennas

Table of Contents

Savvi™ Embedded Ceramic Antennas

Purpose

This document provides information for incorporating KYOCERA AVX's Savvi™ embedded ceramic antennas 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 Savvi Ceramic Product Line

The Savvi series of antennas, listed below, represents a new category of small form factor, internal antennas. Some Savvi antennas, developed for the same application, come in more than one form factor. These provide additional flexibility for designers, for instance, a much smaller size where slightly less performance is acceptable.KYOCERA AVX 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.

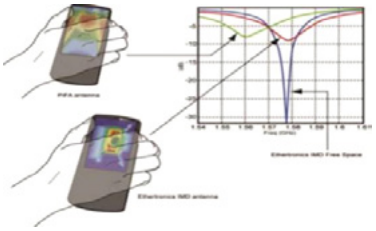
Part Number	Frequency	Application	Size
M310220	2.4 - 2.5 GHz	Bluetooth	3 x 1.5 x 1.08 mm
M620720	868 - 870 MHz / 902 MHz	ISM	6 x 2 x 1.10 mm
M830120	1.575 GHz	GPS	8 x 3 x 1.38 mm
M830320	2.4 - 2.5 GHz	Bluetooth	8 x 3 x 1.38 mm
M830520	2.4 - 2.5 GHz	WLAN	8 x 3 x 1.38 mm

Additional antennas are under development, please see KYOCERA AVX's Website,or ask yourKYOCERA AVX sales person about additional products to meet your needs.

Real-World Performance and Implementation

Ceramic antennas may look alike on the outside, but the important difference is inside. 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.

Stays in Tune



IMD technology provides superior RF field containment, so antennas resist de-tuning to provide a robust radio link regardless of the usage position. Other antennas may experience substantial frequency shifts, and lowered performance, when held by users or placed next to the head.

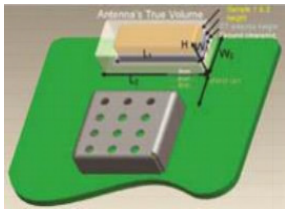
Standard Parts Work in Multiple Locations

High isolation also means a single part can be used for a variety of PCB sizes and in a variety of positions on the PCB. If re-tuning is required, it can be done by slightly changing the antenna ground clearance area on the PCB.

The GPS antenna should be located on the longest size of the PCB for best performance. Other Savvi antennas can be located along any edge.

Savvi™ Embedded Ceramic Antennas

Smallest Effective Size



A ceramic dielectric leads to a small physical size. However, unlike antennas using other technologies, IMD antennas require minimal ground clearance and keep-out areas for surrounding components. This can lead to a smaller “effective” size when all factors are taken into account. In addition to a small “x,y” footprint, Savvi antennas have very low component height to enable ultra- thin, end-user device designs. Multi-band antennas can be packaged in a single device, which further saves PCB real estate while in creasing functionality. And diversity antennas can be included to improve range, coverage and user experience.

High Performance

IMD technology enables antennas with high efficiency and high selectivity. High efficiency enables longer range and greater design margins in end products. High selectivity eliminates the cost and PCB space for additional filters.

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.

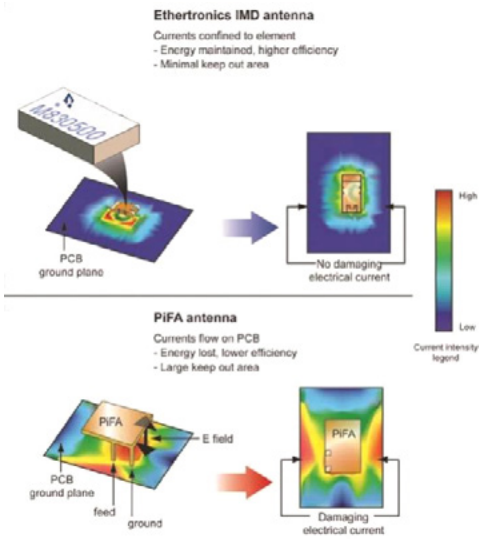
Features, Advantages and Benefits Summar

Feature	Advantage	Benefits
High Performance	High efficiency	Meet and exceed design performance specs. Lower design risks. Enhance end-user satisfaction. Potential for additional device sales.
	High isolation	Less interaction with surrounding components. Smallest effective antenna size when component keep out areas included. Resists de-tuning 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.
Compact Size	Enables design of smallest, thinnest product designs.	Saves board space. Fits into areas other antennas won’t. Increases functionality and sales potential.
Superior RF Field Containment	Virtually eliminates detuning due to device handling during use.	Better performance. Higher end-user satisfaction. Potential for higher sales.

IMD Technology: How it works

IMD technology uses confinement of the electrical field to create the antenna’s mode. The strongly confined antenna mode reduces its coupling to the surrounding environment. The diagram to the right shows the electrical field created on the PCB ground plane for anKYOCERA AVX IMD antenna and a PIFA (Planar Inverted F Antenna). Red areas indicate the highest current while blue areas signify the lowest. As demonstrated, currents from the IMD design are highly localized, while high currents are observed all the way over to the ground plane edge on the PIFA.

KYOCERA AVX's IMD antennas are ideally suited for wireless data devices,where performance, size and system costs are critical. The surface mount design and compact size are suited for high volume applications. Standard antenna profiles are available or can be configured to suit individual OEM requirements



062722

Design Guidelines

Introduction

The Savvi™ line of KYOCERA AVX's ceramic embedded antennas can be designed into many wireless product types. The following sections explain KYOCERA AVX's recommended layouts to help the designer integrate the antenna into a product with optimum performance.

Antenna Location

In order to create an optimized layout for the antenna, one must first understand the Savvi ceramic antenna's unique characteristics, which enable it to offer superior performance. The antenna's performance behavior is controlled by many variables in the surrounding environment. KYOCERA AVX's technical team has determined that several rules are important to take into account when designing a product using this antenna, see Figure 6 below:

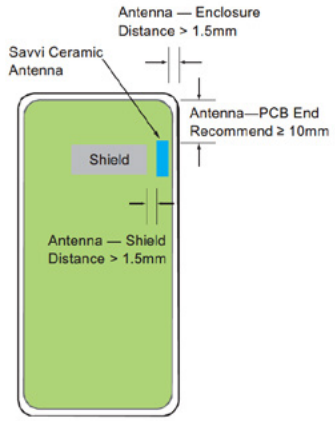


Figure 6

1. Long side of antenna must be along the edge of the ground plane.
2. Ground plane is removed from all layers below the antenna.
3. The distance from the antenna to the enclosure or plastic cover referred to as "**Antenna – Enclosure" distance**, should be greater than 1.5mm.
4. The distance from the antenna to relatively large perturbations, such as a nearby shield or large components, is referred to as "**Antenna - Shield**" distance. This distance is dependent on the height of surrounding components, but should not be less than 1.5mm. See Figure 6.
5. The minimum distance from the end of antenna to either end of PCB is referred to as the "**Antenna--PCB End**" distance. We recommend a distance equal to or greater than 10mm, with better performance from larger distances. Because of IMD's high performance, some designs may allow for smaller distances.

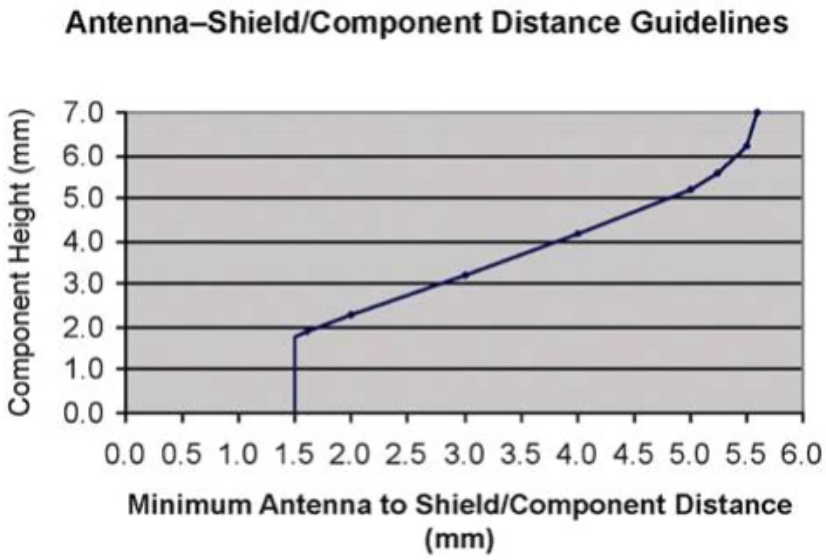


Figure 7 above shows the Minimum Antenna – Shield and/or Component Distance

Tuning Antenna Response on the PCB

Recommendations for ground clearance on an antenna-by-antenna basis are provided in the appendix. In some cases, it may be necessary to tune the response of the antenna once it is placed on the PCB. This can be done by changing the ground clearance around the antenna. The amount of clearance along the width of the antenna is the most sensitive at effecting changes. Increasing ground clearance in the direction of the arrow (see Figure 8 to the left), lowers the frequency at which return loss peaks. (Note M830520 does not follow the above guidelines). Matching can also be used for additional tuning alternatives.

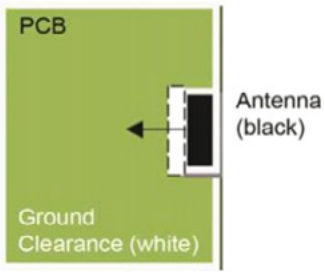
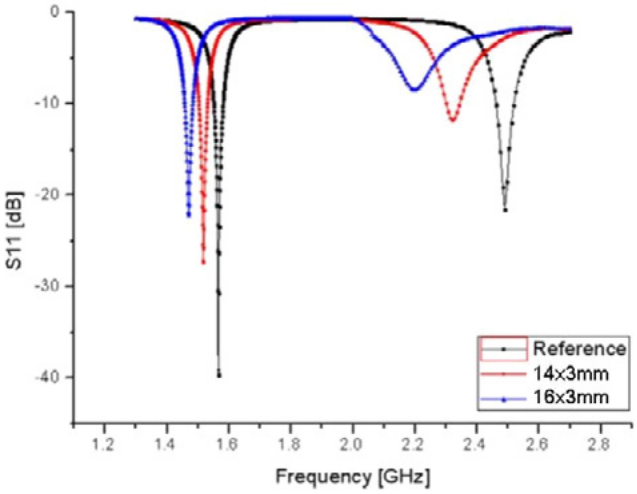
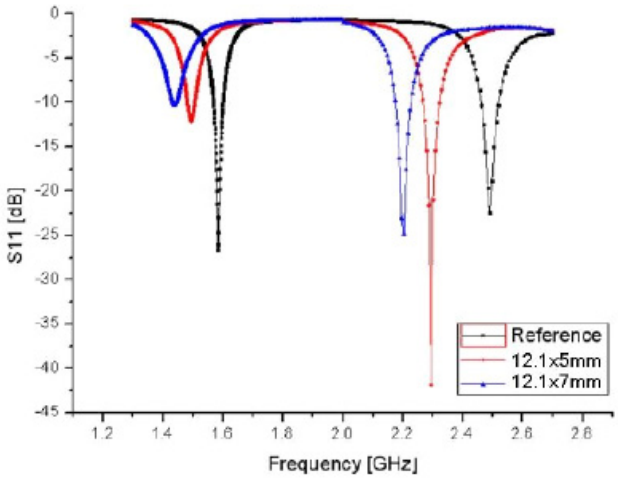
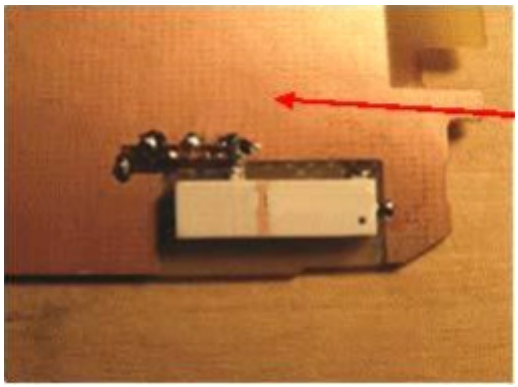


Figure 8

Figure 9 below shows changes in response for the 12x3mm GPS/Bluetooth ceramic antenna. The size differences noted on the charts, such as 12.2mmx5mm, are the length and width of the ground clearance under and around the antenna. The diagram on the left shows changes in response to variations along the width of the antenna (these also correspond to the drawing above). The second chart shows changes in antenna response due to increases in ground clearance along the length of the antenna.





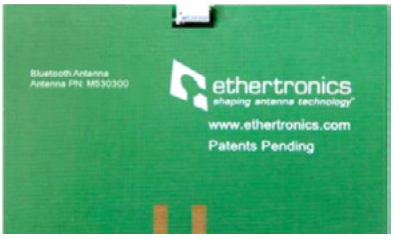


It is clear from the diagrams that the Bluetooth® frequency ranges were impacted the most. It is also clear that changes to the ground clearance along the width of the antenna had the most impact, although changes to both sides had some impact.

In the case of prototypes, it is possible to quickly experiment with changes by adding copper tape or strips, or removing copper around the antenna. See Figure 10 to the left for an example.

Tuning was done by reducing the ground plane on the Bluetooth side

Performance Variation

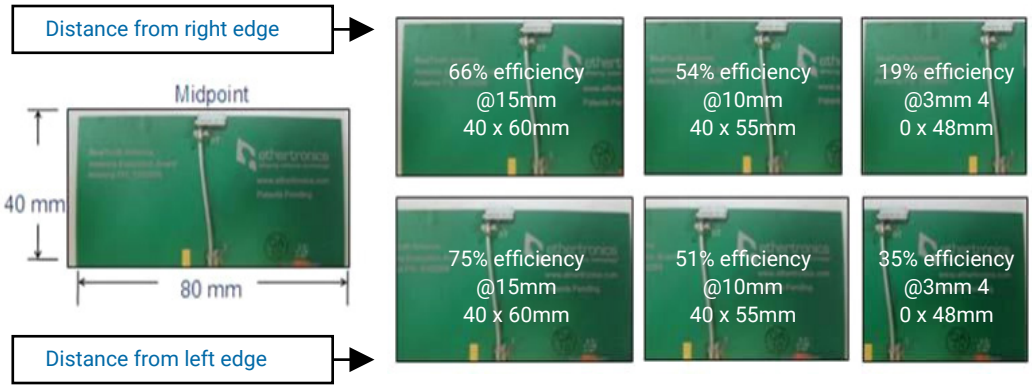


The performance data contained in the appendices are based on recommended ground clearance, pad layouts and antenna placement. Each antenna was positioned at the edge of an 80mmx40mm PCB, at the middle of the longest side. The antenna feed was routed through a via hole to the underside of the board, and connected to a 50 ohm micro strip. This micro strip was routed across the short side of the board to a 50 ohm connector. A picture of the test board for a Bluetooth® antenna is shown in Figure 11 to the left. The dotted line indicates the transmission line on the underside of the PCB.

This section of the application note is intended to provide information on how performance varies outside the conditions used to measure the data in the appendices. It should be noted that there are variations that exist from one type of antenna to another, such as Bluetooth versus GPS. And combination antennas present their own special circumstances. Nevertheless, knowledge of how antenna placement, board size, and other factors impact performance can be helpful in making implementation decisions.

Variations in GPS efficiency based on proximity to PCB edge

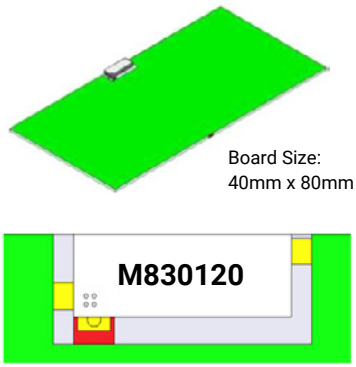
Tests were run on the 8x3mm GPS antenna to document how efficiency varied from the distance the antenna was placed from the right and left PCB edges, see Figure 12 below for results. Substantial efficiency increases were obtained at distances above 10mm from the short end of the board, with further increases obtained at increasing distances from the end. There was less efficiency drop off from the left end. In general,KYOCERA AVX recommends that antennas be positioned 10mm or more from the PCB end. Because of the high efficiency of IMD antennas, some designs may be able to meet their performance

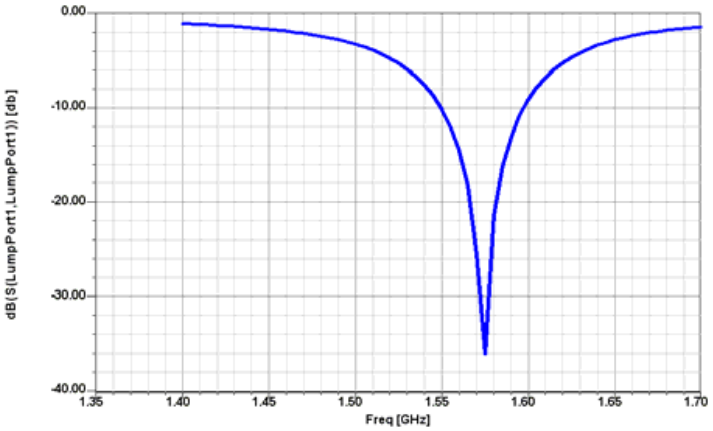


Increasing efficiency through board layout changes

Additional testing was done to increase efficiency and performance when the 8x3mm GPS antenna is placed only 3mm from the board edge. By making changes to the board layout, efficiencies greater than 60% with 31 to 36 MHZ bandwidth can be achieved.

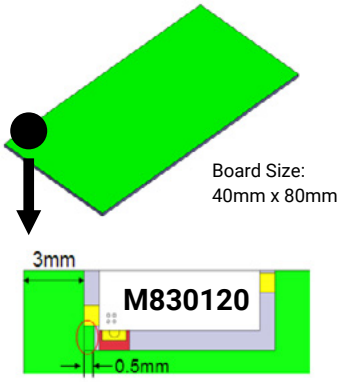
Demo Board

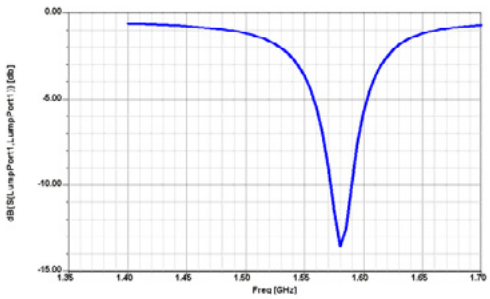




- Peak efficiency: 80%
- Bandwidth (-7 dB): 83MHz

Location 1

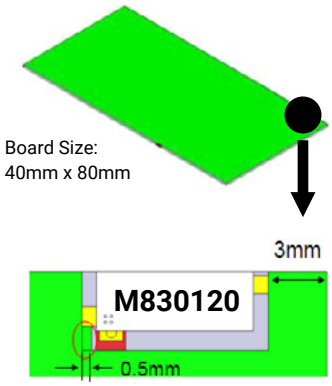


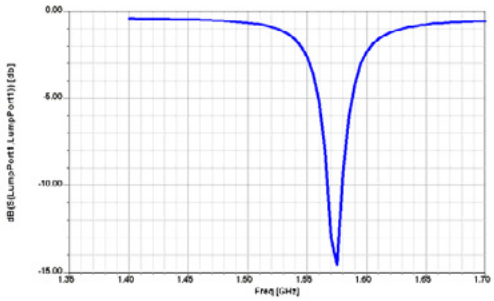


- Peak efficiency: 67%
- Bandwidth (-7 dB): 36MHz

Recommended board layout change in red circle: needs to fill the gap by 0.5mm to optimize the performance.

Location 2





- Peak efficiency: 48%
- Bandwidth (-7 dB): 24MHz

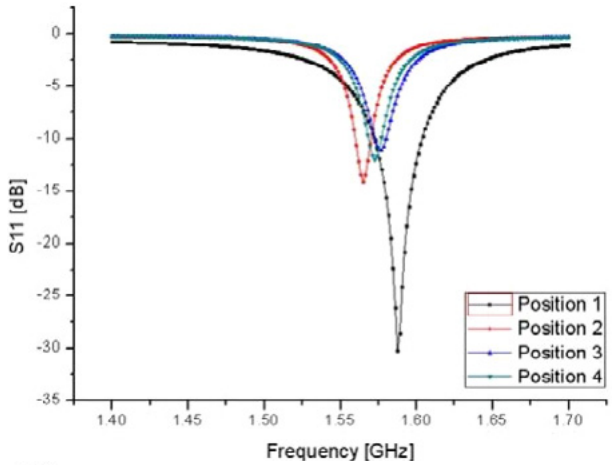
Recommended board layout change in red circle: needs to fill the gap by 0.5mm to optimize the performance.

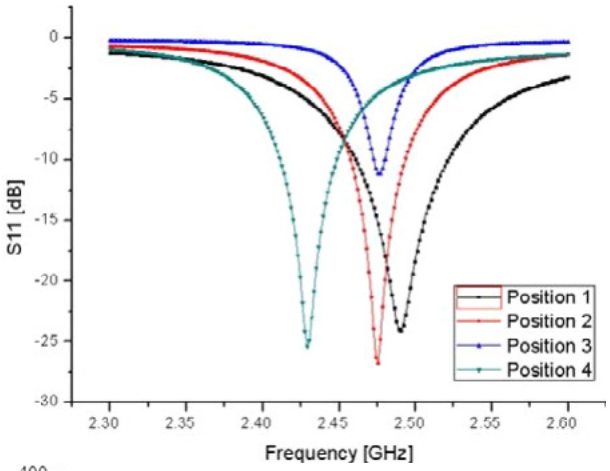
In summary, with minimal board layout changes, the 8x3mm GPS antenna can be optimized on the corner location, 3mm away from the PCB edge. Locating the antenna closer to the edge of the PCB will reduce the bandwidth. Location 1, Figure 14 on previous page, shows good efficiency and more bandwidth compared to Location 2, Figure 15 above. However, Location 2 can be avoided by placing the antenna on the bottom side of the PCB. This placement is the same as Location 1.

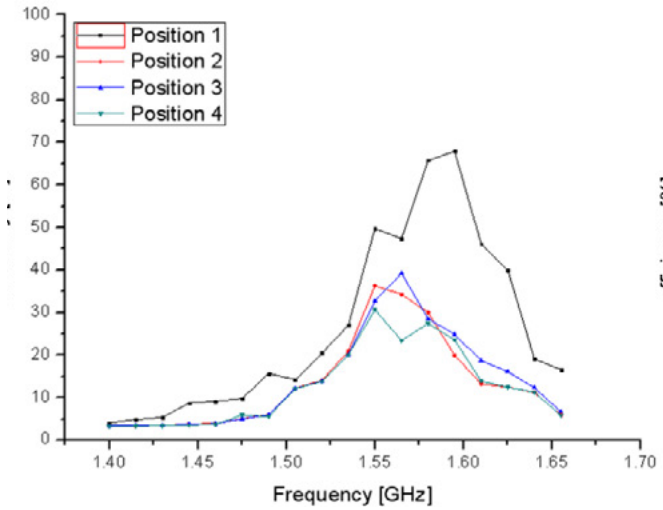
In general,KYOCERA AVX recommends that antennas be positioned on the long side of the PCB if possible. To achieve the best performance, antennas should be placed in Location 1 with 10mm or more from the PCB edge.

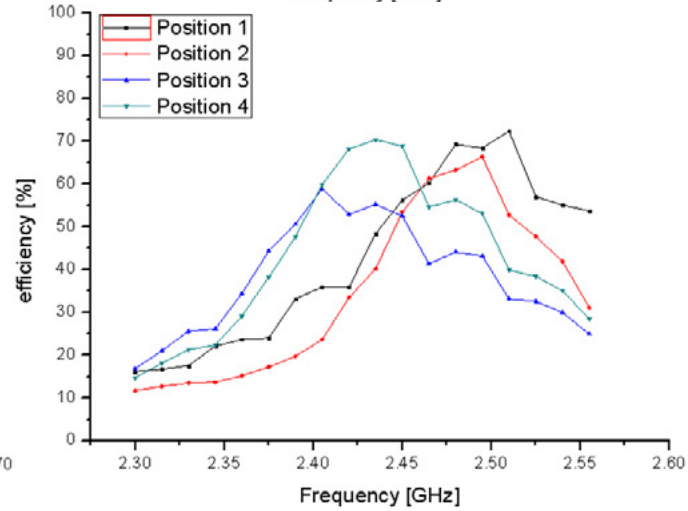
Performance variations based on antenna orientation

Tests were run at different positions on a 40x80mm PCB using the 12x3mm GPS/Bluetooth dual band antenna (see placement in Figure 16 to the left an d results in Figures 17 and 18 below). Test findings indicate that GPS performance was better along the long side of the board. For this reason,KYOCERA AVX recommends that GPS–only antennas be placed along the long PCB side—a better position for the GPS/ Bluetooth antenna as well. The Bluetooth® antenna in the combo unit had better return loss and efficiency on the short edge. If it is necessary to place the combination unit on the short edge, the shift in frequency response can be adjusted by changing the ground clearance, per section 3.2 of this application note.



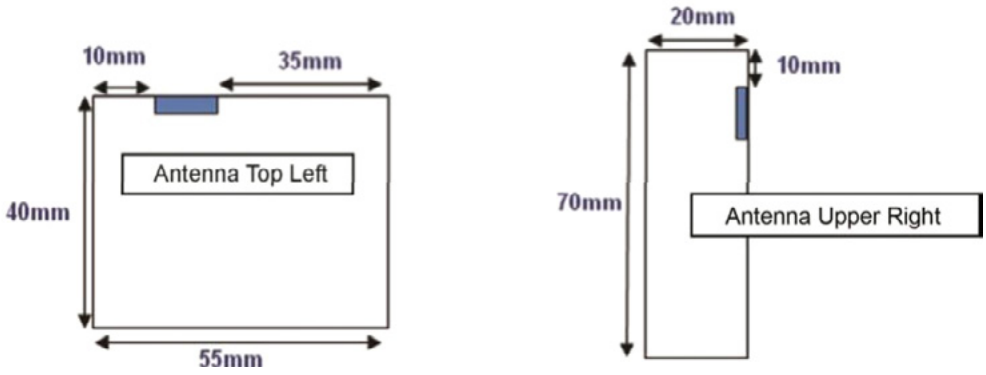






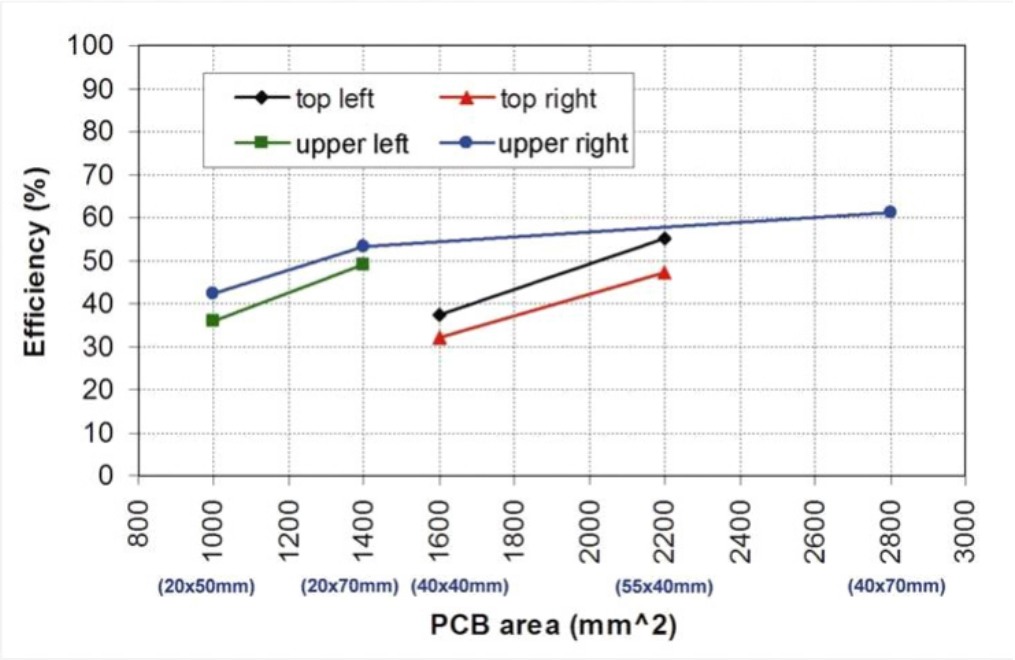
Performance variations based on PCB board size

Impacts of board size were investigated using four different antenna placements of the 8x3mm GPS antenna. Antennas were placed 10mm from the top left and top right, and 10mm from upper right and upper left sides (see Figure 19 below for placement examples and Figure 20 below for results). The size of the PCB was varied along the width and length of each configuration. Note, the GPS antenna was not tested on the "short" side of the board. Current GPS products should always be placed on the long side of the PCB.



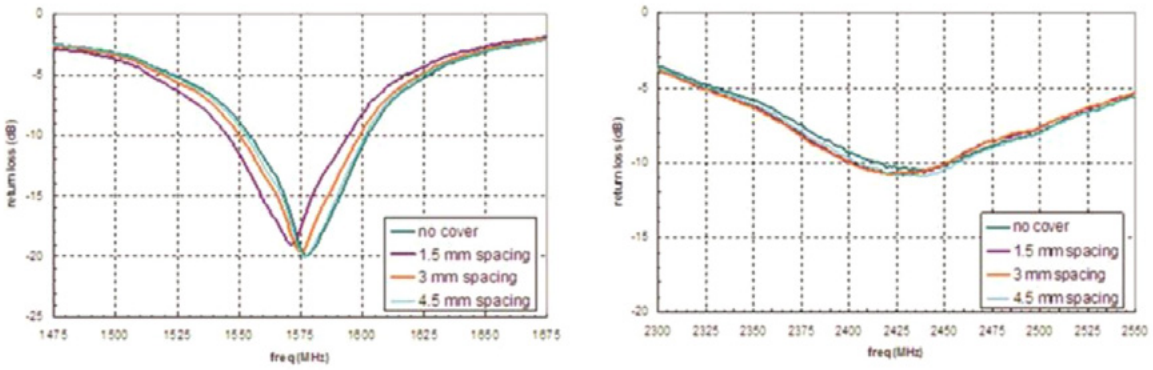
Conclusions were:

- Efficiency increases with overall PCB area
- Efficiency is better on rectangular PCBs than square
- Expanding the PCB along either the width or length increases efficiency
- Antenna placement on the upper right side increased efficiency 5-10% versus placement on the upper left side
- Antenna placement near the top left increased efficiency 5-10% versus placement on the top right



Antenna performance unaffected by plastic housings

1.2 mm ABS plastic was positioned at various distances fromKYOCERA AVX GPS and Bluetooth® antennas to simulate the impact that plastic covers might have on antenna performance. The result was negligible detuning over spacing of 1.5mm or greater (see Figure 21 below)



Tuning Guidelines (Savvi Antennas 3x1.5mm)

EV-Board Structures

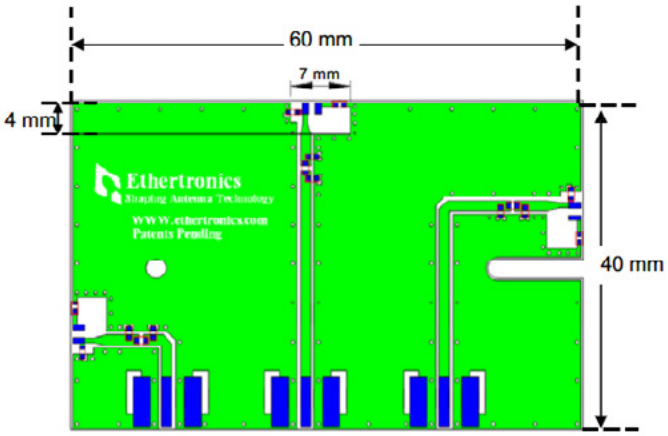
The following are tuning guidelines for 3x1.5mm Savvi Ceramic Antennas. These antennas require the use of a matching circuit for tuning.

BTEV-Board

The Evaluation Board for Bluetooth 3x1.5mm has an antenna placed in the three different locations for evaluation purposes. The optimal location is the long center side. The clearance size is 7x4mm (see Figure 22 below).

BT Part Numbers:

- M310220 (3x1.5mm)



Tuning the frequency by changing the major tuning component

- The frequency can be tuned across a broad range by changing the Capacitor value, see Figure 23 to the right.

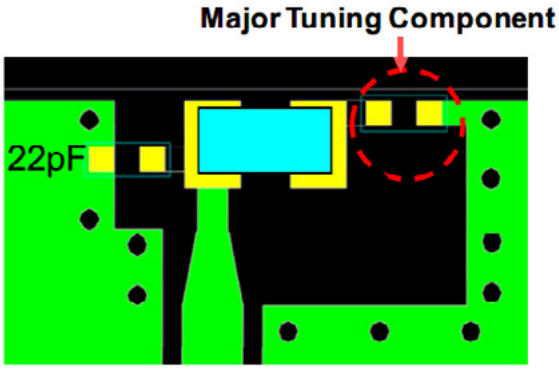


Figure 23

See Figure 24 to the right to see the results of changing the Capacitor value on Return Loss

- When the Capacitor value is decreased, the frequency sifts higher

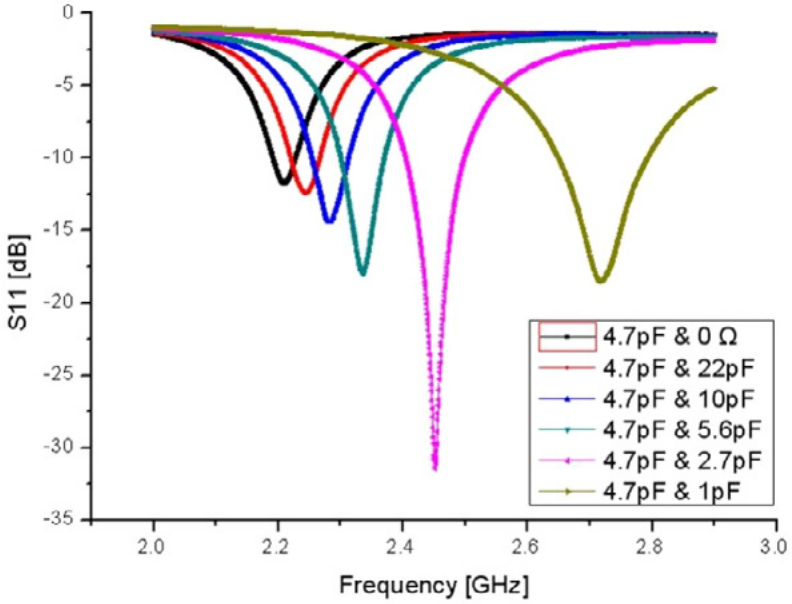


Figure 24

See Figure 25 to the right to see the corresponding efficiency graphs

- Varying the Capacitor value does not severely degrade efficiency

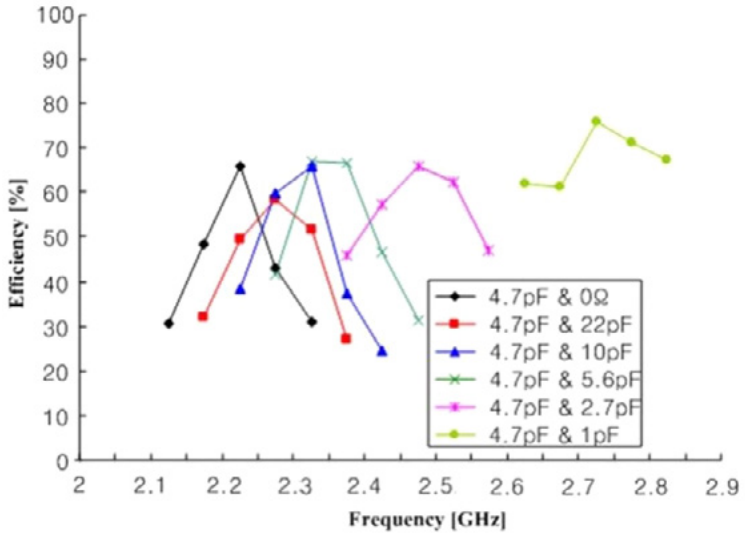


Figure 25

A low value Inductor can be used to shift the frequency lower;
instead of a Capacitor. However, using to high of an inductance value will degrade the efficiency.

See Figure 26 to the right and Figure 27 below to see the results of changing the Inductor value on Return Loss and the corresponding Efficiency.

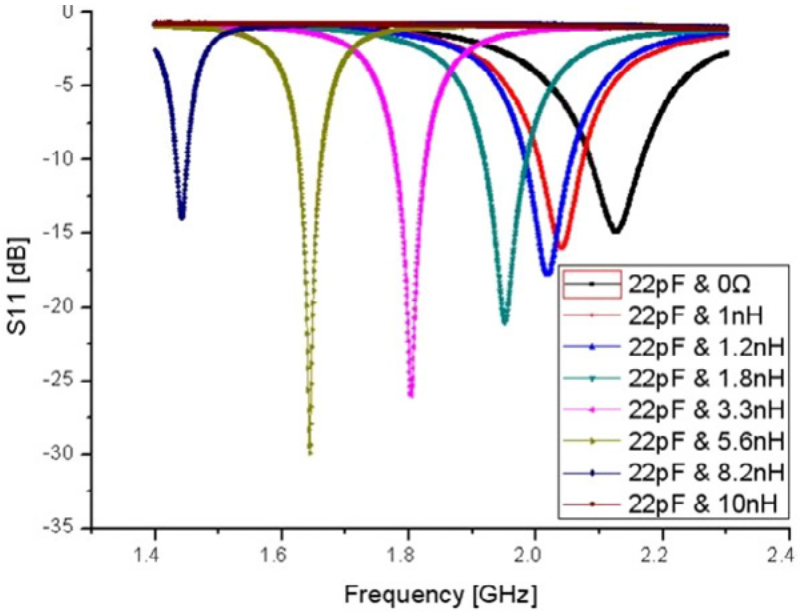


Figure 26

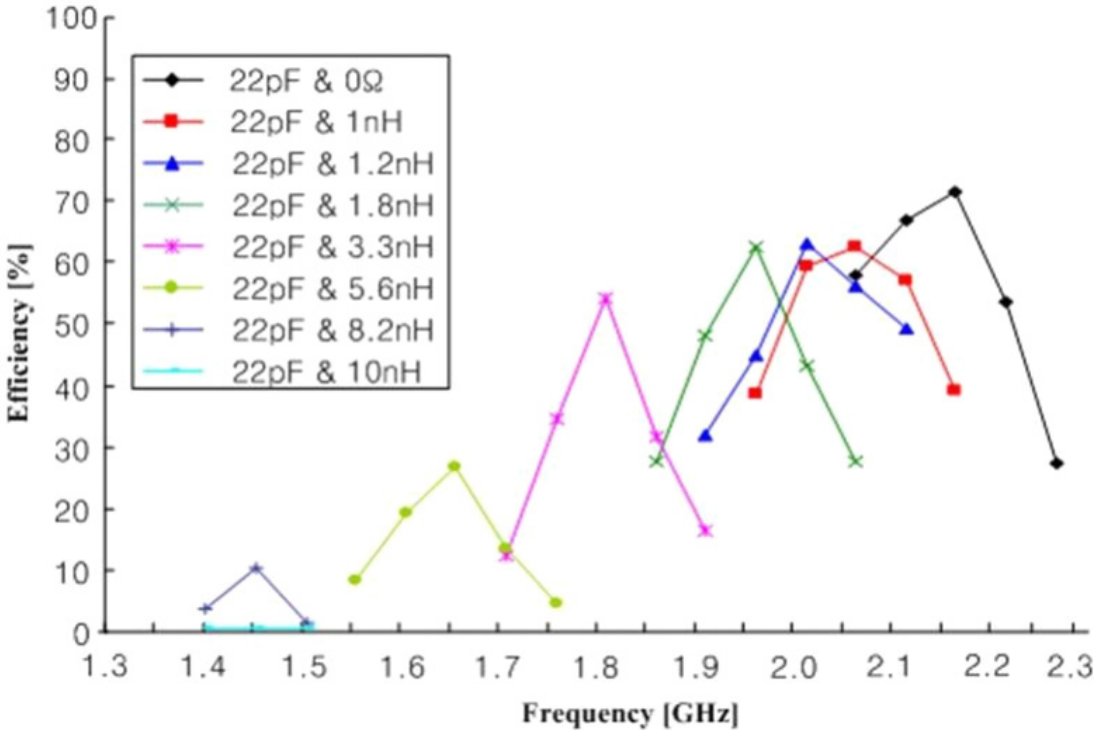


Figure 27

Tuning the frequency by changing the minor tuning component

- VSWR and the operating bandwidth can be optimized by changing the Capacitor value.

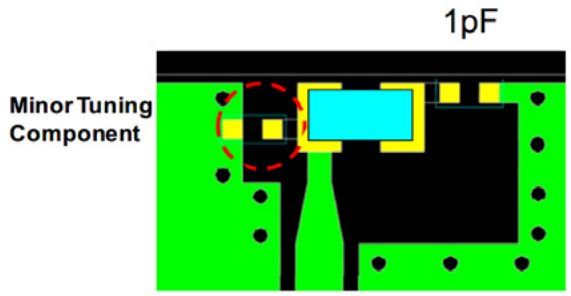


Figure 28

See FIgure 29 to the right and Figure 30 below to see the results of changing the Capacitor value on Return Loss and the corresponding Efficiency.

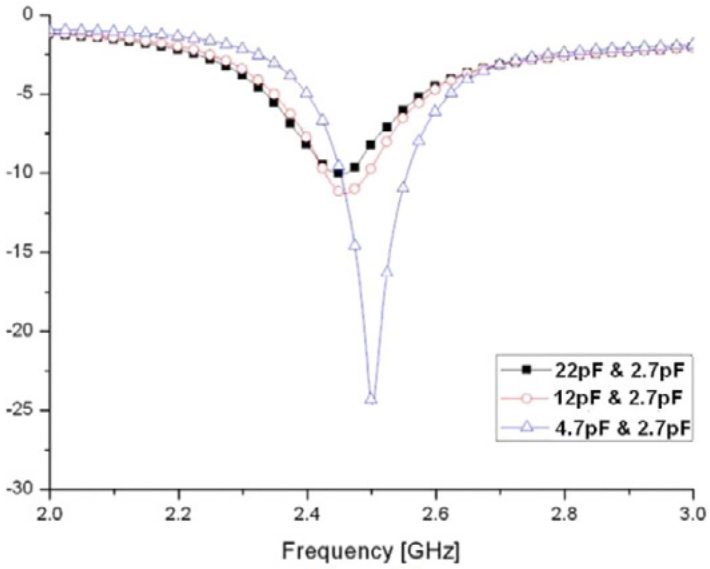


Figure 29

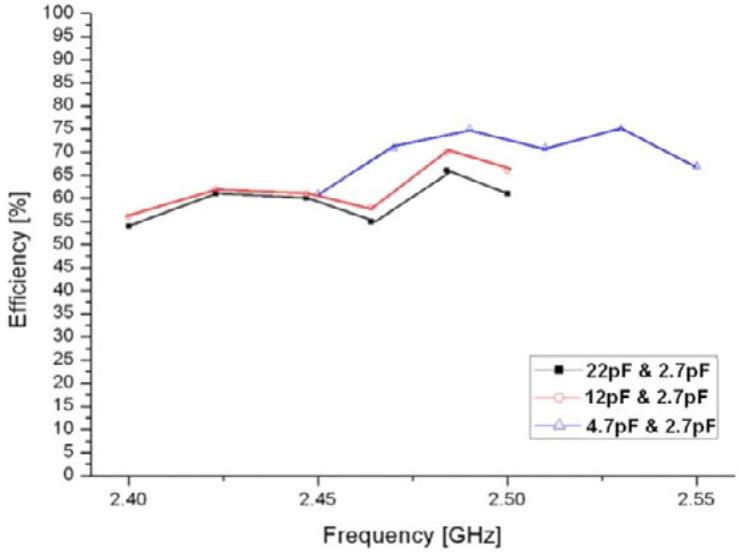


Figure 30

Performance variations based on board size

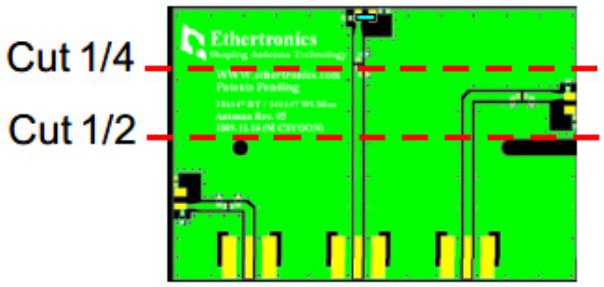


Figure 31

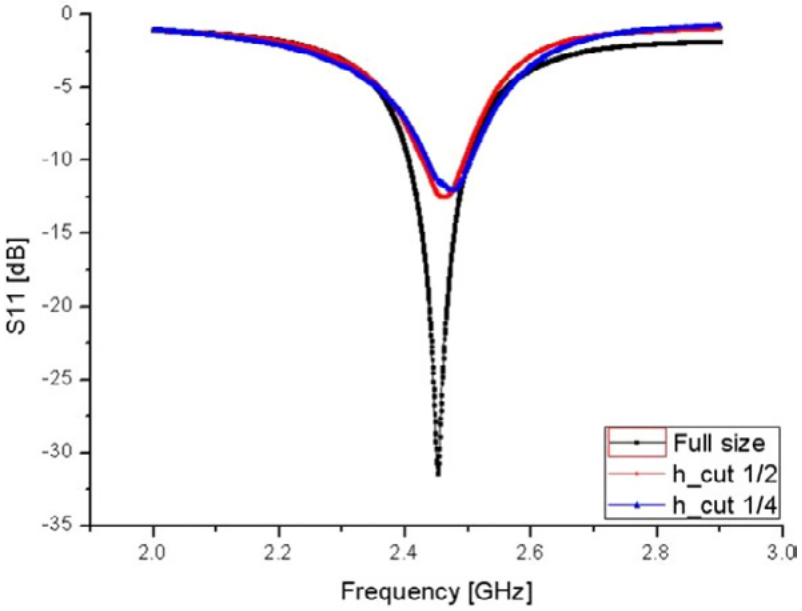


Figure 32

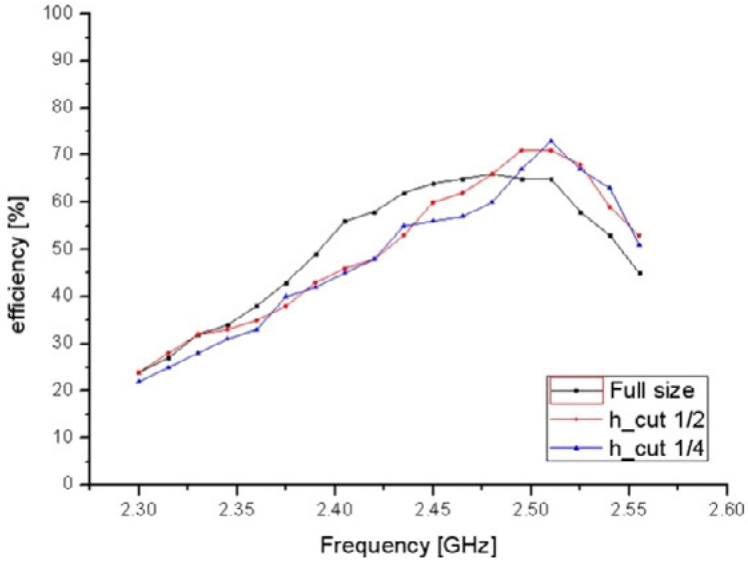
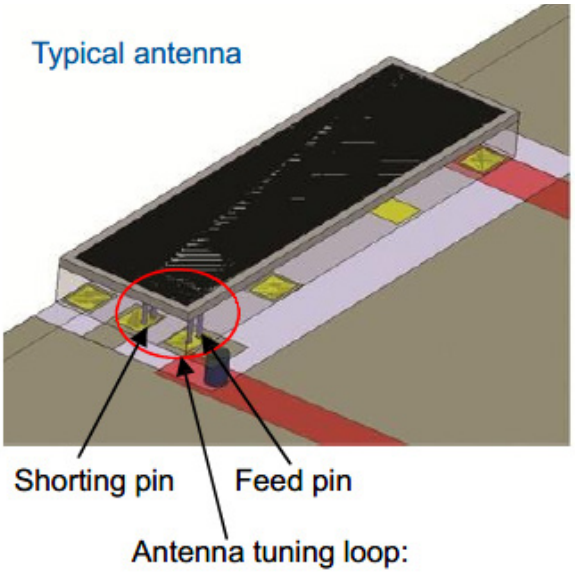


Figure 33

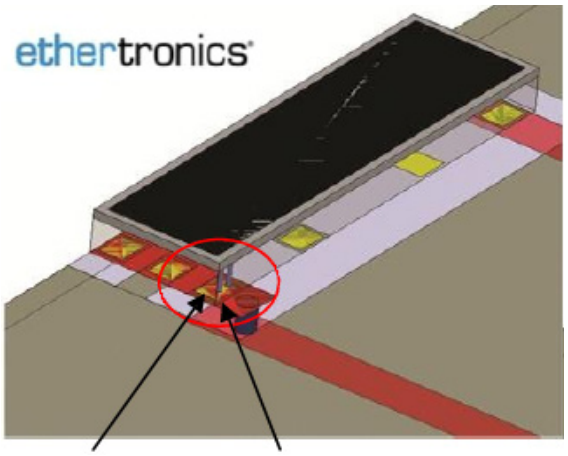
Pad Layout Tips

Important, layout guidelines for correct operation ofKYOCERA AVX Savvi Ceramic Antennas. Please read guidelines below before laying out the antenna in your device. Figure 34 shows the typical antenna layout. Figure 35 shows KYOCERA AVX's antenna layout.



- Change loop size in Smith chart to control bandwidth.
- Give more separation between the feed and the shorting pin to increase loop size in Smith chart

Figure 34



Shorting pin and feed pin are shared inKYOCERA AVX ceramic antenna

- The antenna tuning loop is formed in the PCB board.
- The feed pin and shorting pin are combined because it required very close proximity to achieve more bandwidth.
- It can relieve constraint on bottom pad layout (min. 0.65mm distance between pads
- It shifts frequency down.

Figure 35

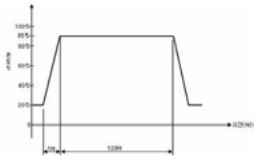
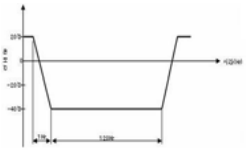
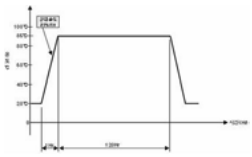
Material Specifications

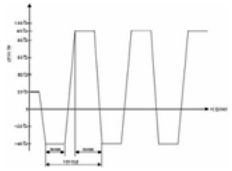
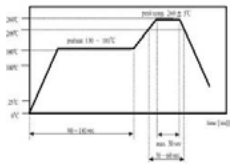
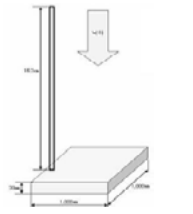
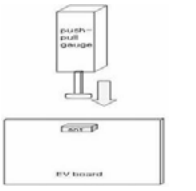
Item	Material
Metal Element	Silver ink
Composite Structure	Ceramic
Contact Finish	Ni Au

Product Testing

KYOCERA AVX's' antennas comply with RoHS directives.KYOCERA AVX' antenna undergo product qualification testing as part of the product development process. The following are the core tests used to qualify the Savvi™ ceramic antennas.

Table 1 Product Qualification Test

NO	Test Type	Items	Test condition	Test Method	
1	Environment Test	High Temp	85°C ± 3°C 120hr ± 2hr		Step 1: Test VSWR by jig Step 2: Put it in the chamber. Step 3: Test it like this picture which explains temp. cycle. Step 4: Test VSWR after 1hr in normal Temp. & normal Humidity
2		Low Temp	-40°C ± 3°C 120hr ±2hr		Step 1: Test VSWR by jig. Step 2: Put it in the chamber. Step 3: Test it like this picture which explains temp. cycle. Step 4: Test VSWR after 1hr in normal Temp. & normal Humidity
3		High Temp. & High Humidity	85°C±3°C RH=85% 120hr ±2hr		Step 1: Test VSWR by jig. Step 2: Put it in the chamber. Step 3: Test it like this picture which explains temp. cycle. Step 4: Test VSWR after 1hr in normal Temp. & normal Humidity
4		Salt Spray	Naci 5% RH=85% 120hr ± 2hr	Step 1: Test VSWR by jig. Step 2: Put it in the chamber. Step 3: Start test. Step 4: Wash the samples. Step 5: Test VSWR after 1hr in normal Temp. & normal Humidity	
5		T	120°C PC RH=100% 96hr	Step 1: Test VSWR by jig. Step 2: Put it in the chamber. Step 3: Test it like this picture which explains temp. circle. Step 4: Test VSWR after 1hr in normal Temp. & normal Humidity	

NO	Test Type	Items	Test condition	Test Method	
6	Environment Test	Thermal Shock	-40°C ± 3°C/30min, 85°C ± 3°C/30min, 32 cycle		Step 1: Test VSWR by jig Step 2: Put it in the chamber. Step 3: Test it like this picture which explains temp. cycle. Step 4: Test VSWR after 1hr in normal Temp. & normal Humidity
7	Reflow Test	Reflow Test	Pre Heating 200°C ± 5°C 30~60sec Peak Heating 260°C ± 5°C 30sec Max		Step 1: Put it in REFLOW Step 2: Test it like this picture which explains temp. Cycle by EV board
8	Mechanical Test	Vibration	Frequency: 10~500hz Acceleration: 10*9.8m/s^2 (G) Sweep time 15min X.Y.Z each 5 times	Step 1: Solder antenna on EV board. Step 2: Assemble EV board (+antenna) on set. Step 3: Test it.	
9		Drop	From 100cm height, drop the sample to the bottom 18 times per one test by drop jig. (each 3 times on 6 surfaces) Jig: using the plastic jig (120 ± 20G) Material of Bottom: Iron Plate		Step 1: Solder anteena on EV board Step 2: Assemble EV board (+antenna) on set. Step 3: Test it like this picture which explains how which explains how to do it.
10		Adhesive Strength	Measure the intensity by pulling the sample on PCB fixed by SMT. Equipment: PUSH-PULL GAUGE		Step 1: Solder antenna on EV board Step 2: Assemble EV board (+antenna) on set. Step 3: Test it like this picture which explains how to do it

Manufacturing and Assembly Guidelines

KYOCERA AVX' ceramic 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 ceramic 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:

- Pick and place machines should use mounting heads that have a compliant nozzle or force control.
- For manual mounting and handling, vacuum pens should use mounting heads that have a compliant nozzle or force control.

KYOCERA AVX' antennas are not moisture sensitive and the ceramic antennas meet the requirement 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 antenna 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.)

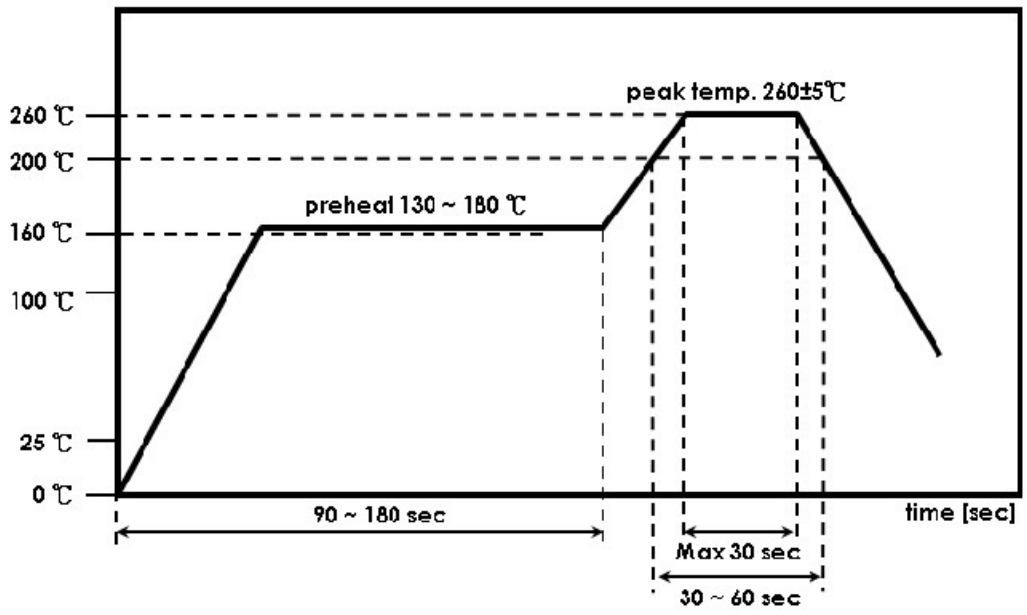
Paste Stencil Recommendation

KYOCERA AVX recommends application of paste stencil to a thickness of 0.1mm, applied to within 0.05 mm of the solder mask surrounding each exposed metal pad on the PCB. PCB layouts for each antenna are provided below.

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 ceramic antenna:

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



Glue Under/Edge Fill

KYOCERA AVX requires using glue as an under fill for increased adhesion strength. Please contact KYOCERA AVX's for more information.Recommended glue: ThreeBond 2212B or similar.

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's.

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

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 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 vaccum pen to lift the antenna straight up off the PCB. Avoid twisting or rotating the device while removing it.

Tape & Reel Specifications

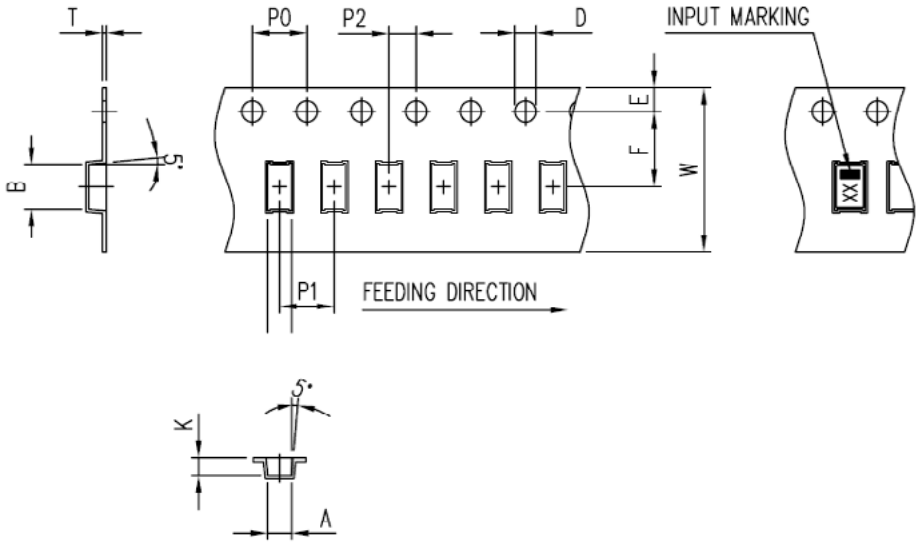
Tape and Reel specifications vary with the size of the antenna. Below is a indicating the standard quantity per reel and box for each given antenna size.

Part Number	Antenna Size	Qty per Reel	Minimum Order Qty
M310220	3x1.5 mm	1,000	1,000 / 1 Reel
M310220-01	3x1.5 mm	(DEMO BOARD)	(DEMO BOARD)
M620720	6x2 mm	1,000	1,000 / 1 Reel
M620720-01	6x2 mm	(DEMO BOARD)	(DEMO BOARD)
M830120	8x3 mm	1,000	1,000 / 1 Reel
M830120-01	8x3 mm	(DEMO BOARD)	(DEMO BOARD)
M830320	8x3 mm	1,000	1,000 / 1 Reel
M830320-01	8x3 mm	(DEMO BOARD)	(DEMO BOARD)
M830520	8x3 mm	1,000	1,000 / 1 Reel
M830520-01	8x3 mm	(DEMO BOARD)	(DEMO BOARD)

Demo boards can be ordered through KYOCERA AVX Authorized distributors: http://www.kyocera-avx.com/contact-us/authorized/
Live stock can be checked at: http://www.kyocera-avx.com/resources/distributor-stock-check/
Listed on the following pages are the tape and reel specifications for the various KYOCERA AVX ceramic antennas

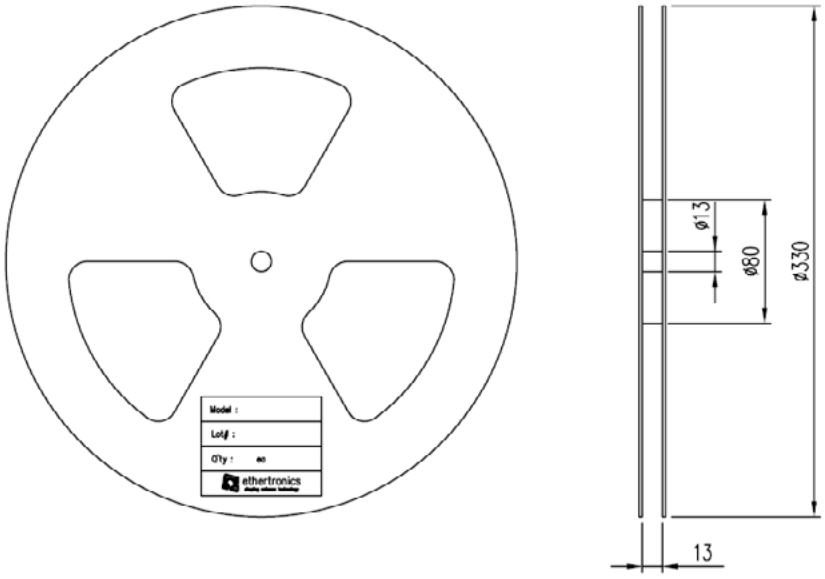
Tape & Reel Specifications

3mm x 1.5mm



Dimension			
A	1.75±0.1	E	1.75±0.1
B	3.30±0.1	F	5.5±0.1
D	1.55±0.05	K	1.35±0.1
P0	4.0±0.1	T	0.3±0.05
P1	4.0±0.1	W	12.0±0.3
P2	2.0±0.1		

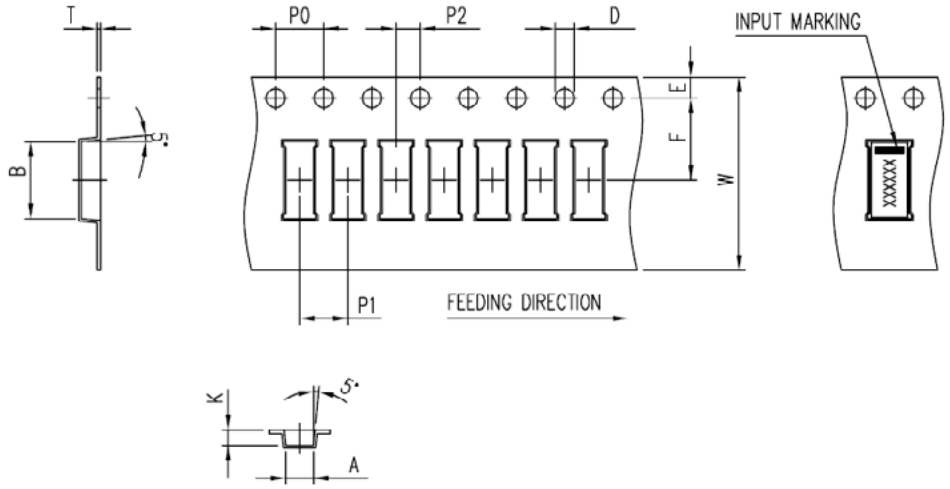
Part Name	Carrier Tape	**Unit**	mm
Material	A - PET	**Scale**	2/1
Inner Chip Size	3x1.5x1.08t	**General Tolerance**	±0.1
Packing QTY.	10000 PCS	**Designed By**	SW CHOI



Part Name	Reel	**Unit**	mm
Material	PS	**Scale**	1/4
Designed By	SW CHOI	**Packing QTY.**	10000 PCS

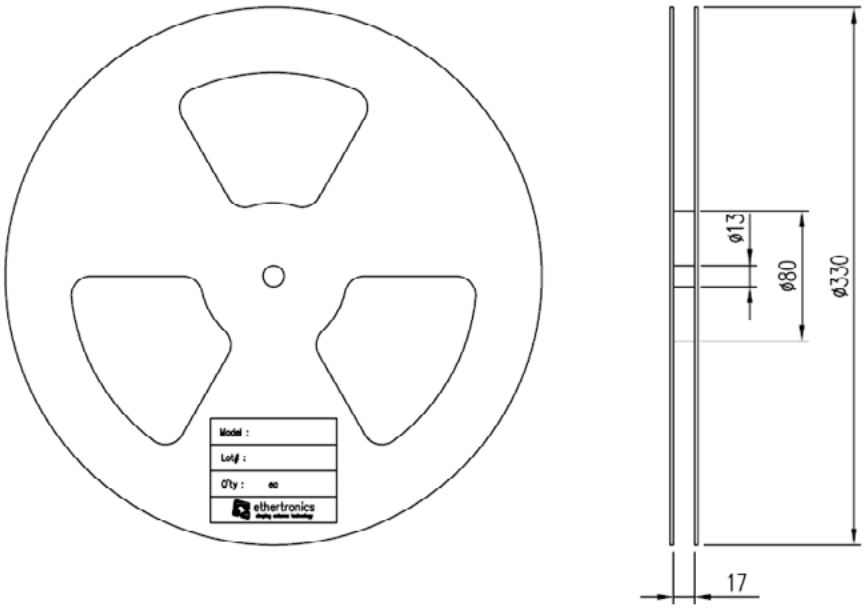
Tape & Reel Specifications

6mm x 2mm



Dimension			
A	2.3±0.1	E	1.75±0.1
B	6.4±0.1	F	7.5±0.1
D	1.55±0.05	K	1.35±0.1
P0	4.0±0.1	T	0.3±0.05
P1	4.0±0.1	W	16.0±0.3
P2	2.0±0.1		

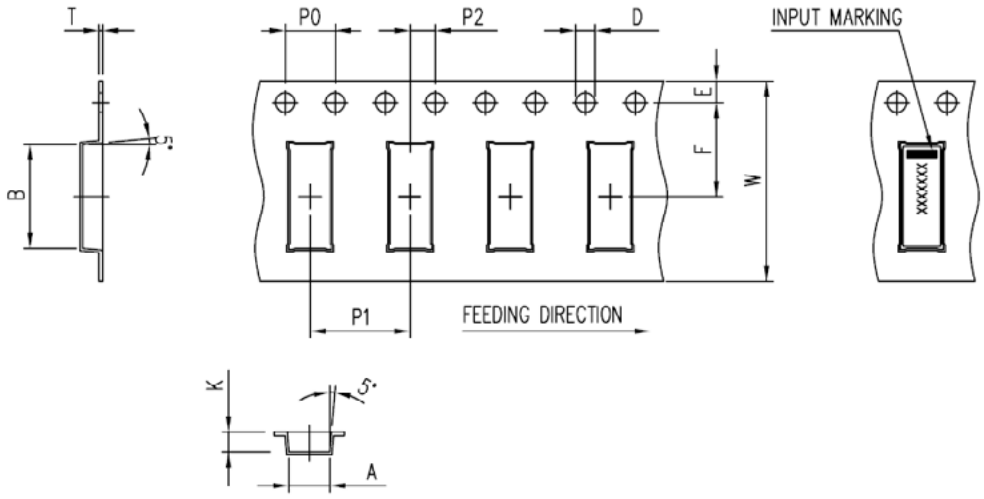
Part Name	Carrier Tape	**Unit**	mm
Material	A - PET	**Scale**	2/1
Inner Chip Size	6x2x1	**General Tolerance**	±0.1
Packing QTY.	1000 PCS	**Designed By**	SW CHOI



Part Name	Reel	**Unit**	mm
Material	PS	**Scale**	1/4
Designed By	SW CHOI	**Packing QTY.**	1000 PCS

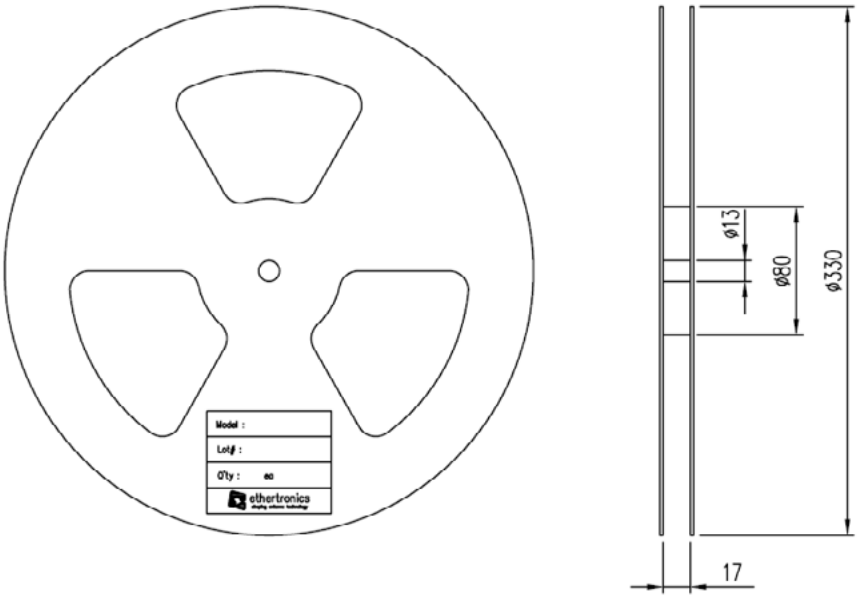
Tape & Reel Specifications

8mm x 3mm



Dimension			
A	2.3±0.1	E	1.75±0.1
B	8.3±0.1	F	7.5±0.1
D	1.55±0.05	K	1.6±0.1
P0	4.0±0.1	T	0.3±0.05
P1	8.0±0.1	W	16.0±0.3
P2	2.0±0.1		

Part Name	Carrier Tape	**Unit**	mm
Material	A - PET	**Scale**	2/1
Inner Chip Size	8x3x1.33	**General Tolerance**	±0.1
Packing QTY.	4500 PCS	**Designed By**	SW CHOI



Part Name	Reel	**Unit**	mm
Material	PS	**Scale**	1/4
Designed By	SW CHOI	**Packing QTY.**	4500 PCS

Product Definitions

Product Date Code

The Savvi™ line of ceramic antennas include a printed code to indicate the manufacturing date. Figure 36, shows the date code marking and the full Part Number.

Y = Year of Manufacture

- 9 = 2009
- 0 = 2010
- 1 = 2011

WW = Week of Manufacture

- 01 = Week 1
- 02 = Week 2
- 38 = Week 38

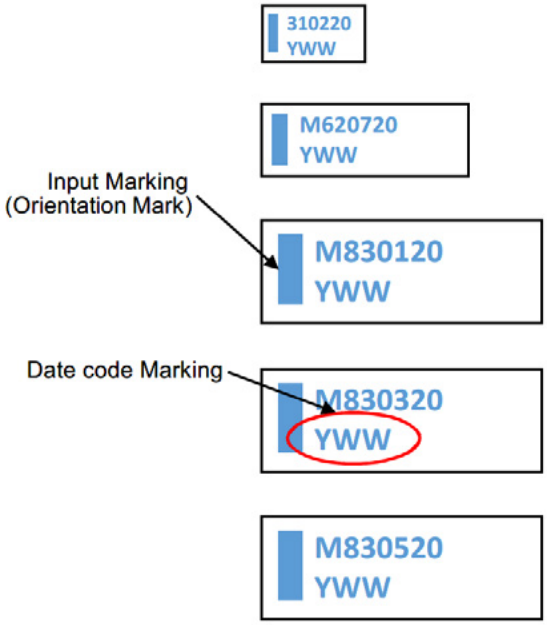


Figure 36

Appendix 1

Summary of Savvi™ 3x1.5 Bluetooth®/WiMAX/ZigBee Ceramic Antenna Part No. M310220

Electrical Specifications

Typical Characteristics

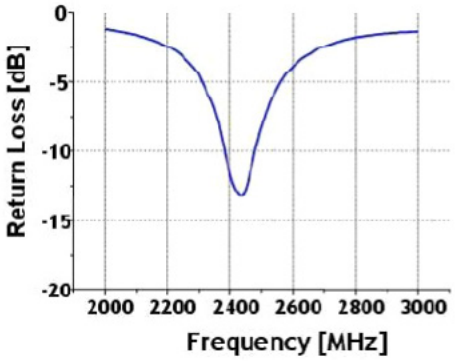
(WiMAX 2.5-2.7 GHz capable with tuning.
See Frequency Tuning Guidelines in Section 3.5).

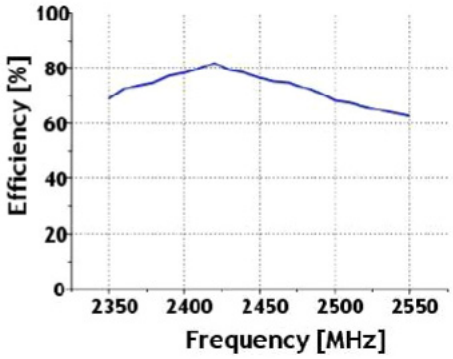
BT/WiFi Antenna	2.400–2.480 GHz
Peak Gain	-1.3 dBi
Average Efficiency	75%
VSWR Match	2.0:1 max
Feed Point Impedance	50 ohms unbalanced
Power Handling	0.5 Watt cw
Polarization	Linear

Mechanical Specifications

Size	3.00x1.50x1.08mm
Mounting	Surface mount
Packaging	Tape & Reel M310220: Minimum Order Quantity of 1,000 pcs Order multiples of 1,000 pcs

Typical Return Loss, Efficiency

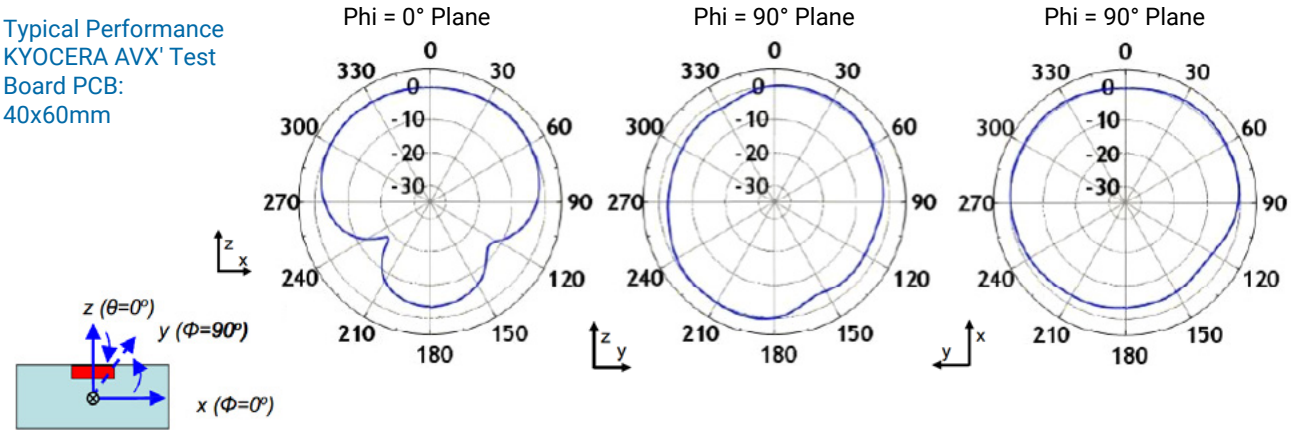




Antenna Radiation Patterns

2.4 GHz Band

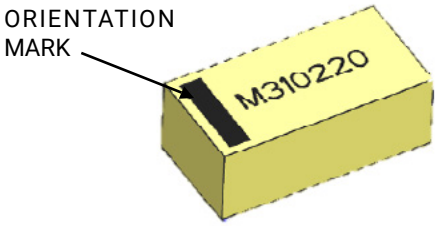
Typical Performance
KYOCERA AVX' Test Board PCB:
40x60mm



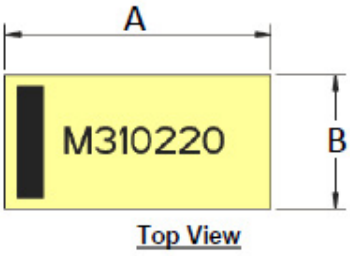
To optimize design using KYOCERA AVX's Savvi™ Bluetooth®/WiMAX single band antenna, the PCB should use the recommended land pattern shown in the Figures below. The land patterns are composed of a 50 ohm line connected to each antenna feed point. Ground clearance under the antenna, as shown in the PCB layout below, is recommended in order to maximize the antenna's performance. The antenna should be located along an edge of the PCB

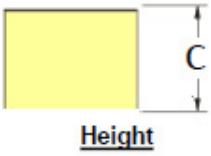
*Note: Please see Tuning Guidelines (page 14) for more information on the matching circuit

Antenna Pad Layout

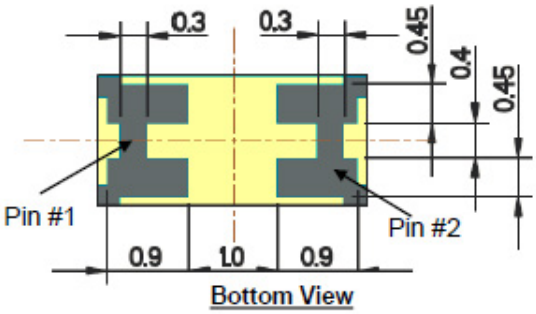


Part number	A (mm)	B (mm)	C (mm)
M310220	3.00 ± 0.2	1.50 ± 0.2	1.08 ± 0.1





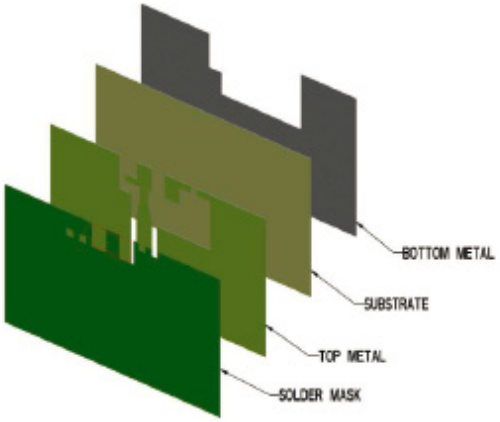
Pin	Description
1	Feed
2	Ground

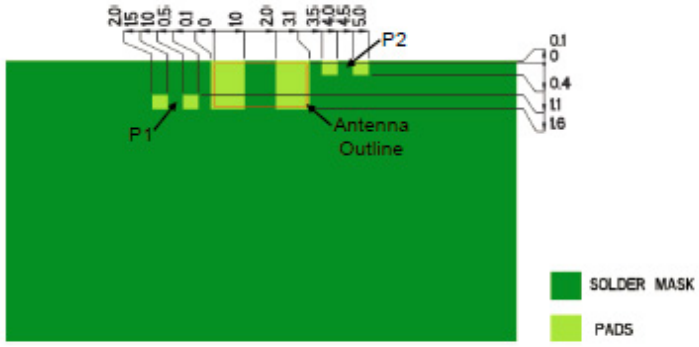


To optimize design using KYOCERA AVX' Savvi™ Bluetooth®/WiMAX single band antenna, the PCB should use the recommended land pattern shown in the Figures below. The land patterns are composed of a 50 ohm line connected to each antenna feed point. Ground clearance under the antenna, as shown in the PCB layout below, is recommended in order to maximize the antenna's performance. The antenna should be located along an edge of the PCB

*Note: Please see Tuning Guidelines (page 14) for more information on the matching circuit

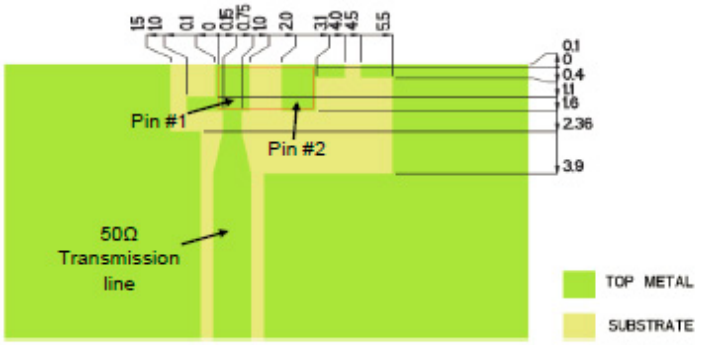
PCB Layout





Pin Description

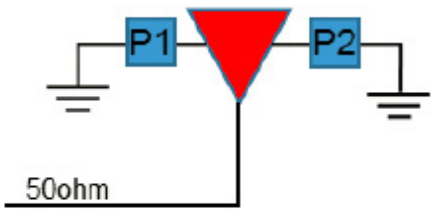
Pin #	Description
1	Feed
2	Ground

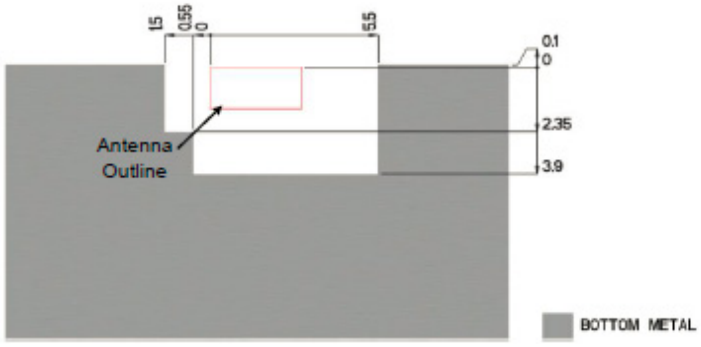


Matching Network (Demo Board)

Component	Value	Tolerance
P1	4.7pF	±0.05pF
P2	2.7pF	±0.05pF

*Actual matching values depend on customer design





Appendix 2

Summary of Savvi™ 6x2mm ISM Ceramic Antenna Part No. M620720

Electrical Specifications

Typical Characteristics

(ISM 868–870 MHz capable by changing the Major Tuning Component. See PCB Layout on following page).

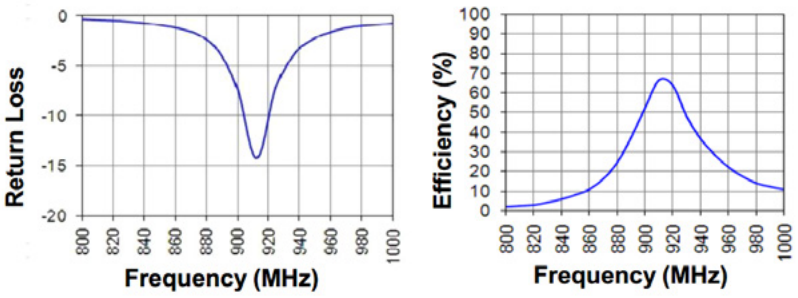
ISM Antenna	902–928 MHz
Peak Gain	2.56 dBi
Average Efficiency	58%
VSWR Match	2.6:1 max
Feed Point Impedance	50 ohms unbalanced
Power Handling	0.5 Watt cw
Polarization	Linear

Mechanical Specifications

Size	3.00x1.50x1.08mm
Mounting	Surface mount
Packaging	Tape & Reel M620720: Minimum Order Quantity of 1,000 pcs Order multiples of 1,000 pcs M620720-10K: Minimum Order Quantity of 100,000 pcs. Order multiples of 10,000 pcs.

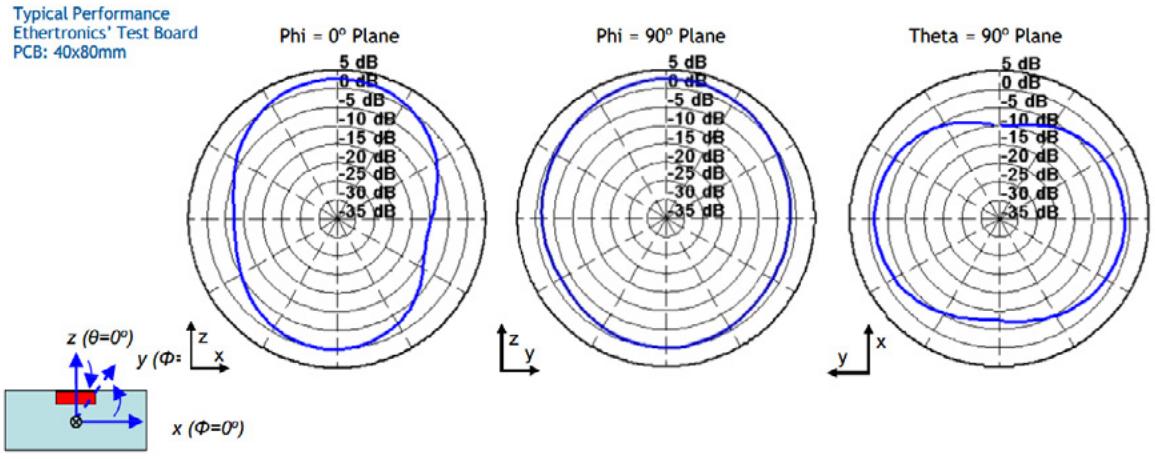
Typical Efficiency, Return Loss (902–928 MHz)

(Note: the Return Loss plot at right used a 4.7 pF capacitor by Murata (GJM1555C1H4R7BB01). Results may change when using capacitors from different vendors due to manufacturing tolerances. See PCB Layout Guidelines).



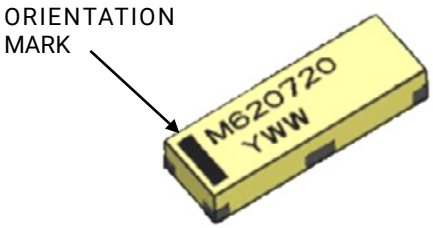
Antenna Radiation Patterns

2.4 GHz Band

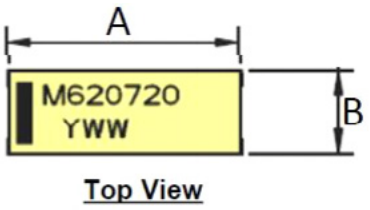


To optimize product designs using KYOCERA AVX SavviTM ISM 800/900 antennas, the PCB should use the recommended land pattern shown in the Figures below. The land patterns are composed of a 50 ohm line connected to each antenna feed point. Ground clearance around and under the antenna, as shown in the PCB layout below, is recommended in order to maximize the antenna's performance. The antenna should be located along an edge of the PCB.

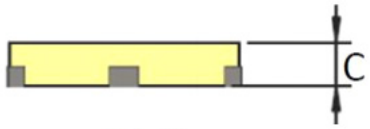
Antenna Pad Layout



Part number	A (mm)	B (mm)	C (mm)
M620720	6.00 ± 0.2	2.00 ± 0.2	1.08 ± 0.1

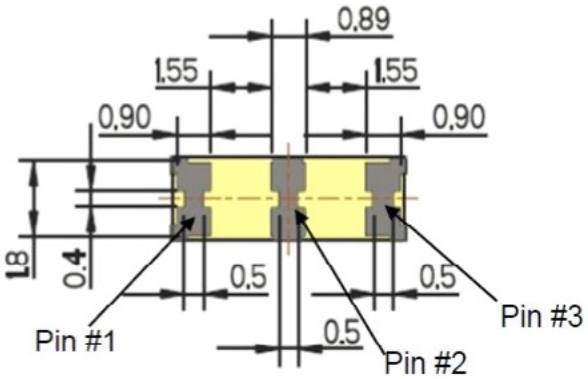


Top View



Height

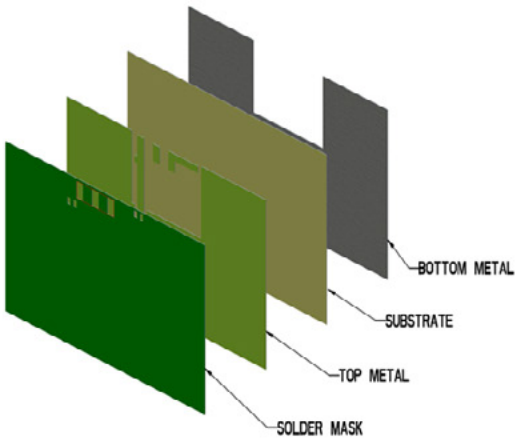
Pin	Description
1	Feed
2	Dummy
4	Ground

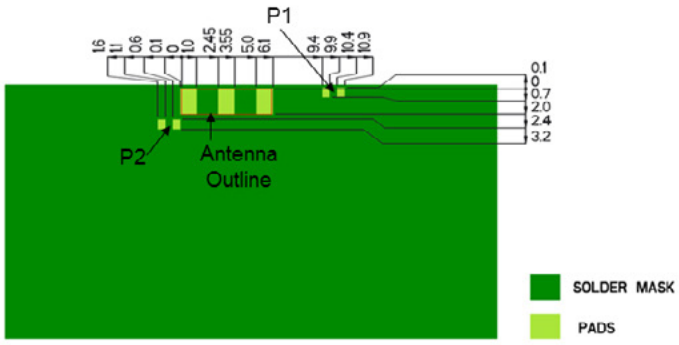


Bottom View

To optimize product designs using KYOCERA AVX SavviTM ISM 800/900 antennas, the PCB should use the recommended land pattern shown in the Figures below. The land patterns are composed of a 50 ohm line connected to each antenna feed point. Ground clearance around and under the antenna, as shown in the PCB layout below, is recommended in order to maximize the antenna's performance. The antenna should be located along an edge of the PCB.

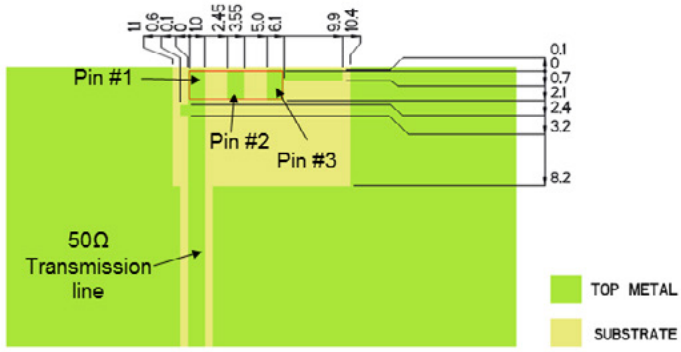
PCB Layout





Pin Description

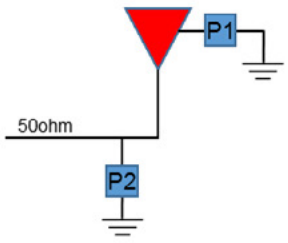
Pin #	Description
1	Feed
2	Dummy
3	Ground

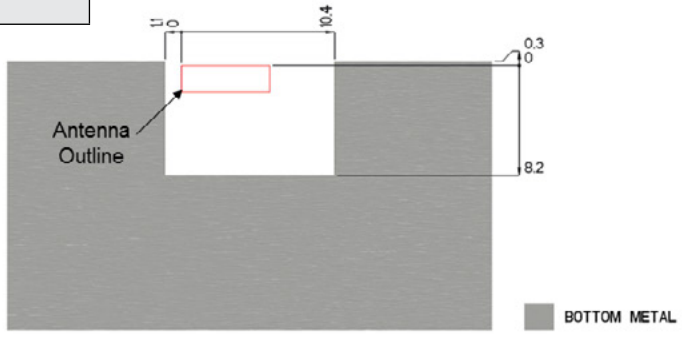


Matching Network (Demo Board)

	868-870 MHz		902-928 MHz	
Component	**Value**	**Tolerance**	**Value**	**Tolerance**
P1	5pF	±0.05pF	3.6 pF	±0.05pF
P2	82 pF ± 5%			

*Actual matching values depend on customer design





KYOCERA AVX provides antenna layout files in .DXF format. This is a universal file format and should be converted into your specific file format.

Product specifications subject to change without notice.

Appendix 3
Summary of Savvi™ 8x3mm GPS Ceramic Antenna Part No. M830120

Electrical Specifications

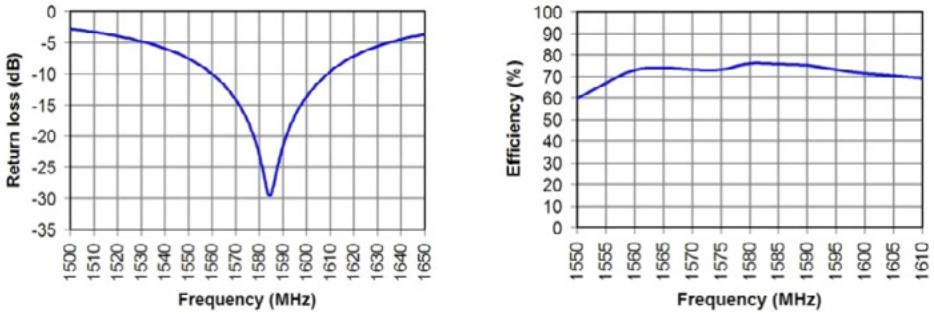
Typical Characteristics

GPS Antenna	1.575 GHz
Peak Gain	1.78 dBi
Average Efficiency	75%
VSWR Match	1.7:1 max
Feed Point Impedance	50 Ω unbalanced
Power Handling	0.5 Watt cw
Polarization	Linear

Mechanical Specifications

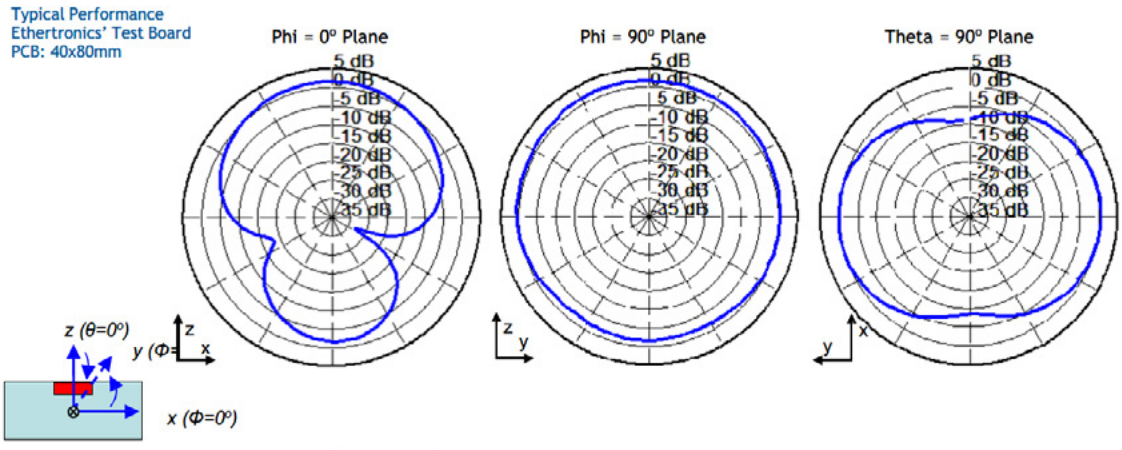
Size	8x3x1.38mm
Mounting	Surface mount
Packaging	Tape & Reel M830120: Minimum Order Quantity of 1,000 pcs Order multiples of 1,000 pcs M830120-4.5K: Minimum Order Quantity of 99,000 pcs. Order multiples of 4,000 pcs.

Typical Return Loss, Efficiency



Antenna Radiation Patterns

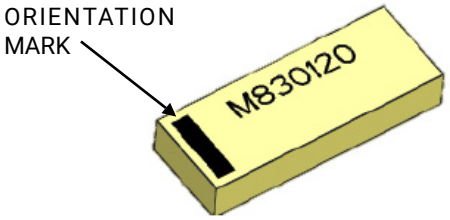
1.575 GHz Band



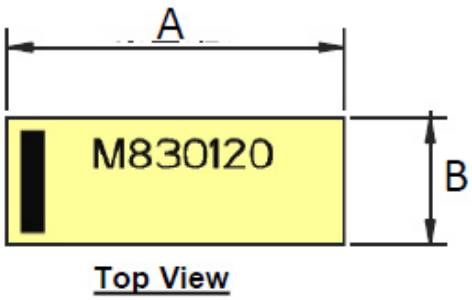
Product specifications subject to change without notice.

To optimize product designs using KYOCERA AVX SavviTM GPS single band antennas, the PCB should use the recommended land pattern shown in the Figures below. The land patterns are composed of a 50 ohm line connected to each antenna feed point. Ground clearance around and under the antenna, as shown in the PCB layout below, is recommended in order to maximize the antenna's performance. The antenna should be located along an edge of the PCB. The recommended default matching component value is 100pF.

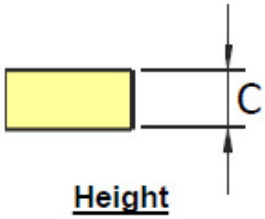
Antenna Pad Layout



Part Number	A (mm)	B (mm)	C (mm)
M830120	8.00 ± 0.2	1.50 ± 0.2	1.33 ± 0.1



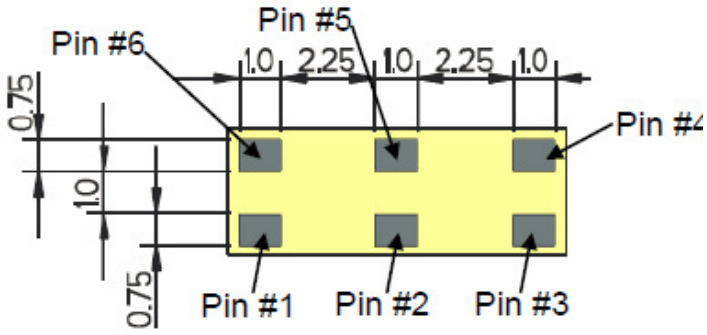
Top View



Height

Pin Description

Pin #	Description
1	Ground
2	Dummy Pad
3	Matching Circuit Connection
4	Dummy Pad
5	Dummy Pad
6	Feed

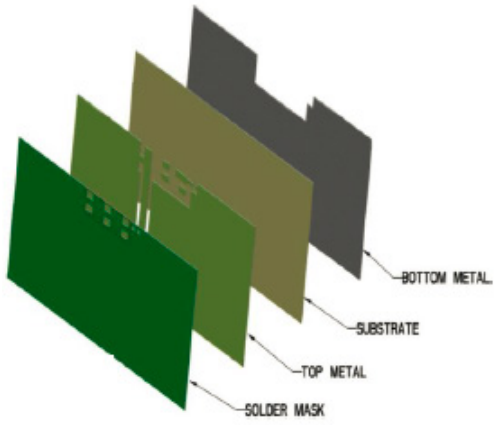


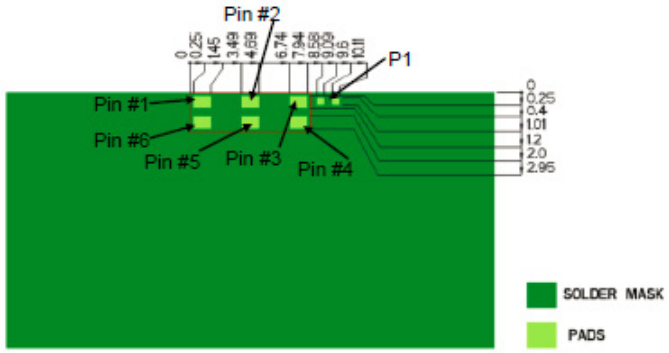
Bottom View

To optimize product designs using KYOCERA AVX SavviTM GPS single band antennas, the PCB should use the recommended land pattern shown in the Figures below. The land patterns are composed of a 50 ohm line connected to each antenna feed point. Ground clearance around and under the antenna, as shown in the PCB layout below, is recommended in order to maximize the antenna's performance. The antenna should be located along an edge of the PCB. The recommended default matching component value is 100pF.

*Note: Please see Tuning Guidelines (page 14) for more information on the matching circuit

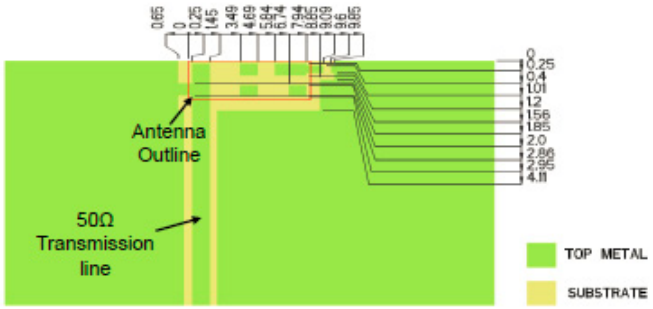
PCB Layout





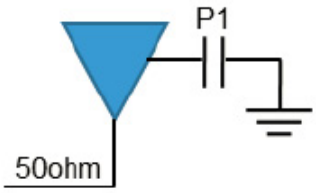
Pin Description

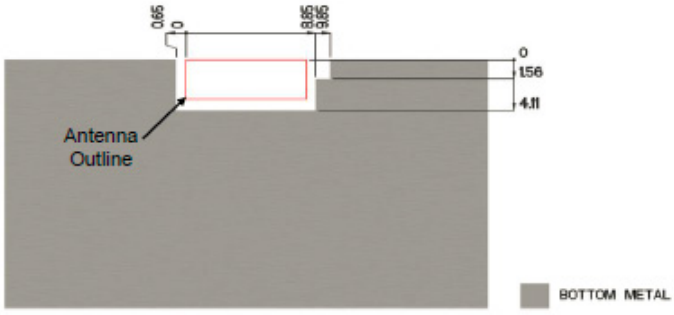
Pin #	Description
1	Ground
2	Dummy Pad
3	Matching Circuit Connection
4	Dummy Pad
5	Dummy Pad
6	Feed



Matching Pi Network

Component	Value	Tolerance
P1	0Ω	N/A





KYOCERA AVX provides antenna layout files in .DXF format. This is a universal file format and should be converted into your specific file format.

Appendix 4

Summary of Savvi™ 8x3mm Bluetooth® Ceramic Antenna Part No. M830320

Electrical Specifications

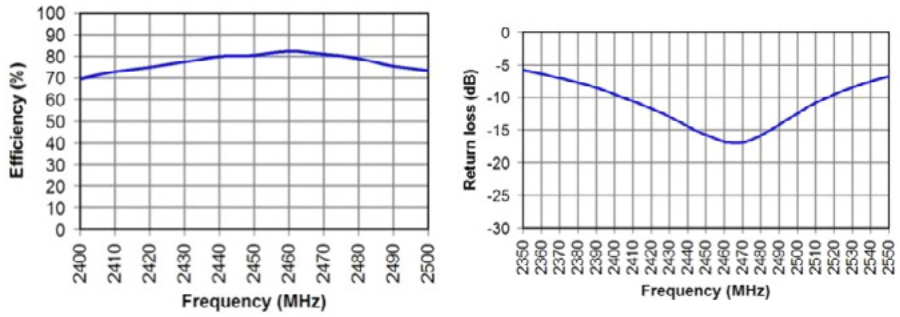
Typical Characteristics

BT Antenna	2400 - 2485 (MHz)
Peak Gain	1.8 dBi
Average Efficiency	72%
VSWR Match	2.0:1 max
Feed Point Impedance	50 Ω unbalanced
Power Handling	0.5 Watt CW
Polarization	Linear

Mechanical Specifications

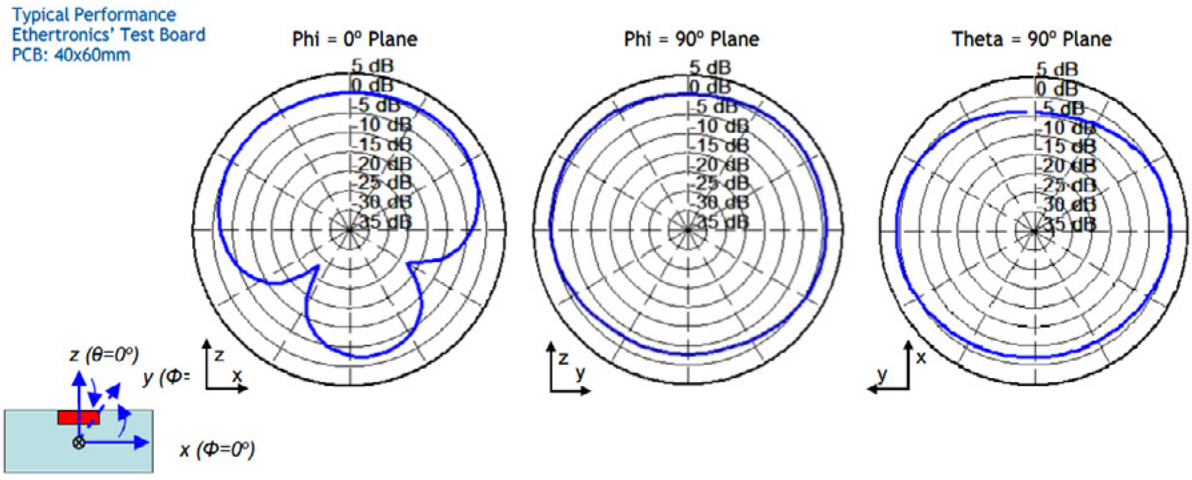
Size	8x3x1.38mm
Mounting	Surface mount
Packaging	Tape & Reel M830320: Minimum Order Quantity of 1,000 pcs Order multiples of 1,000 pcs M830320-4.5K: Minimum Order Quantity of 99,000 pcs. Order multiples of 4,500 pcs.

Typical Efficiency, Return Loss



Antenna Radiation Patterns

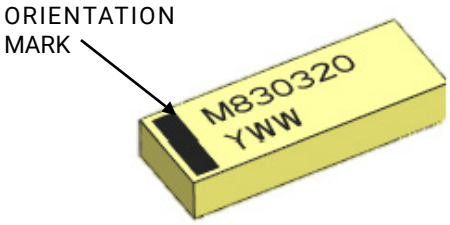
1.575 GHz Band



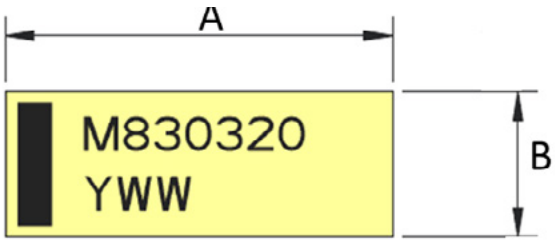
Product specifications subject to change without notice.

To optimize designs using KYOCERA AVX's SavviTM Bluetooth® single band antenna, the PCB should use the recommended land pattern shown in the Figures below. The land patterns are composed of a 50 ohm line connected to each antenna feed point. Ground clearance around and under the antenna, as shown in the PCB layout below, is recommended in order to maximize the antenna's performance. The antenna should be located along an edge of the PCB.

Antenna Pad Layout

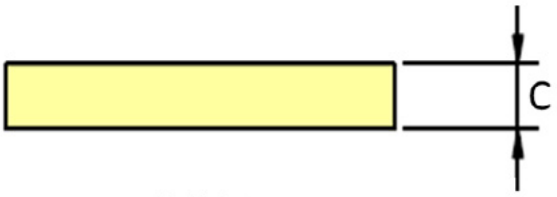


Part Number	A (mm)	B (mm)	C (mm)
M830320	8.00 ± 0.2	3.00 ± 0.2	1.33 ± 0.1

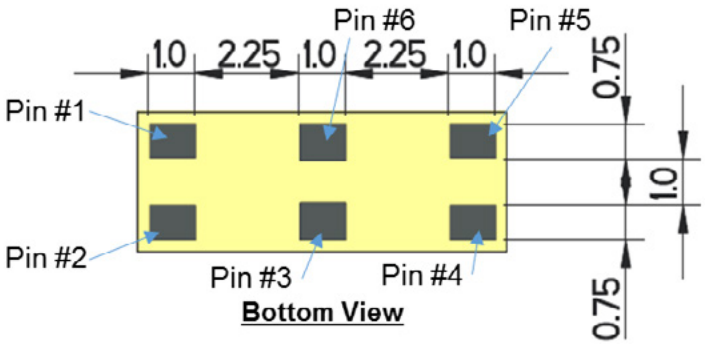


Top View

Pin	Description
1	Feed
2	Ground
3	Dummy Pad
4	Ground
5	Dummy Pad
6	Dummy Pad



Height

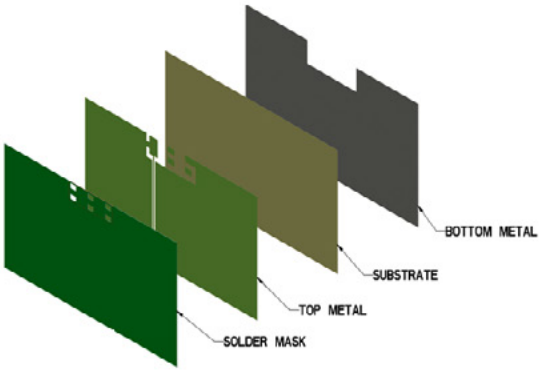


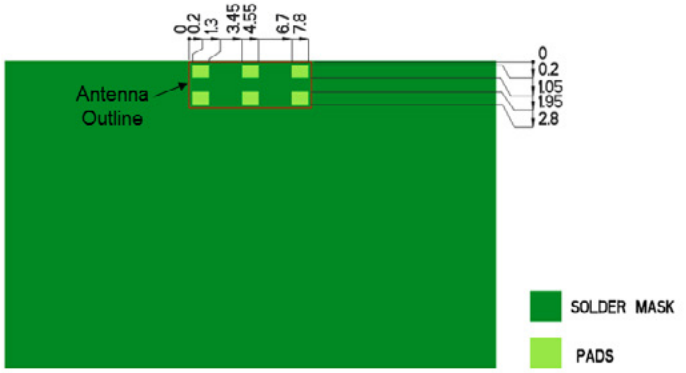
Bottom View

Product specifications subject to change without notice.

To optimize designs using KYOCERA AVX's SavviTM Bluetooth® single band antenna, the PCB should use the recommended land pattern shown in the Figures below. The land patterns are composed of a 50 ohm line connected to each antenna feed point. Ground clearance around and under the antenna, as shown in the PCB layout below, is recommended in order to maximize the antenna's performance. The antenna should be located along an edge of the PCB.

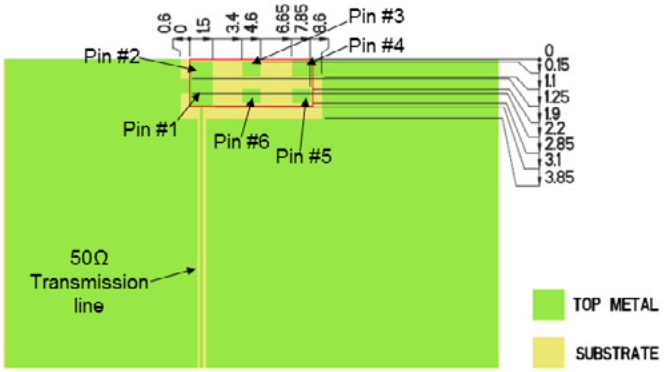
PCB Layout

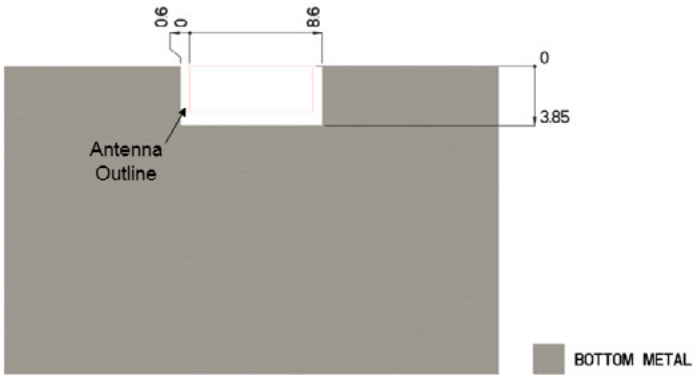




Pin Description

Pin	Description
1	Feed
2	Ground
3	Dummy Pad
4	Ground
5	Dummy Pad
6	Dummy Pad





KYOCERA AVX provides antenna layout files in .DXF format. This is a universal file format and should be converted into your specific file format.

Appendix 5

Summary of Savvi™ 8x3mm WLAN Ceramic Antenna Part No. M830520

Electrical Specifications

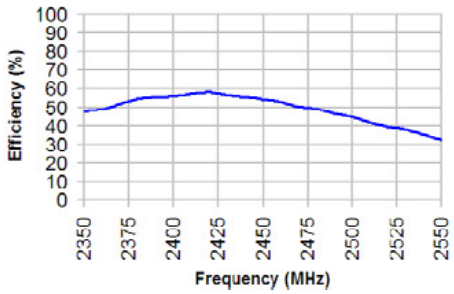
Typical Characteristics

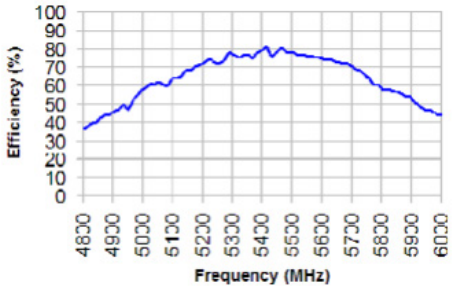
BT Antenna	2.4 – 2.5 GHz (b/g/n)	4.9 – 5.8 GHz (a/n)
Peak Gain	1.10 dBi	3.2 dBi
Average Efficiency	54%	69%
VSWR Match	2.6:1 max	3.0:1 max
Feed Point Impedance	50 Ω unbalanced	

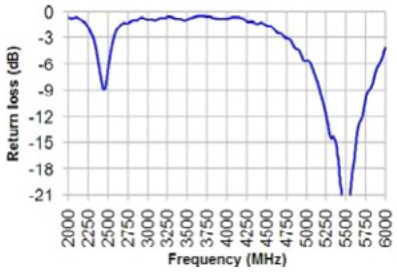
Mechanical Specifications

Size	8.00x3.00x1.38mm
Mounting	Surface mount
Packaging	Tape & Reel M830520: Minimum Order Quantity of 1,000 pcs Order multiples of 1,000 pcs M830520-4.5K: Minimum Order Quantity of 99,000 pcs. Order multiples of 4,500 pcs.

Typical Efficiency, Return Loss

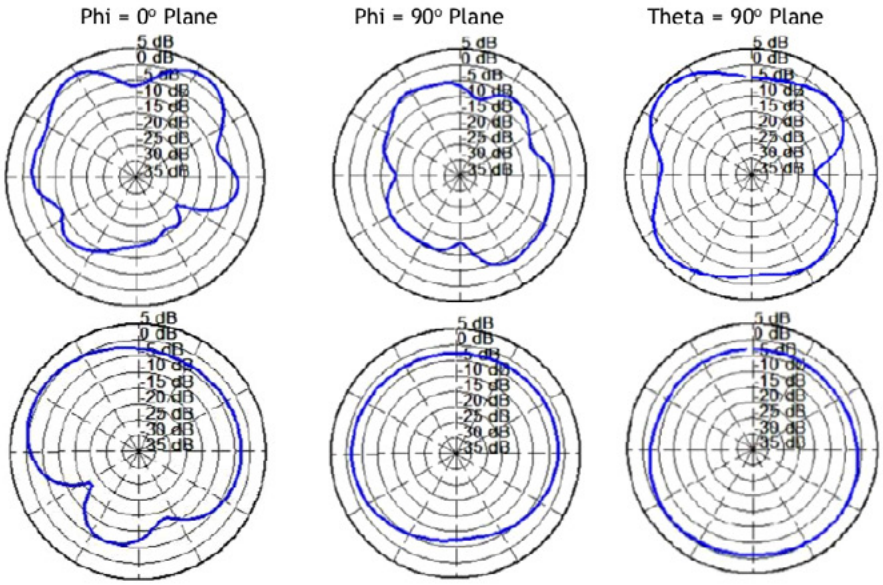


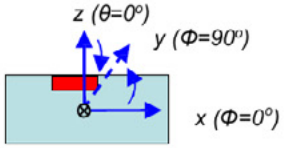




Antenna Radiation Patterns

Typical Performance
Ethertronics' Test Board
PCB: 40x80mm

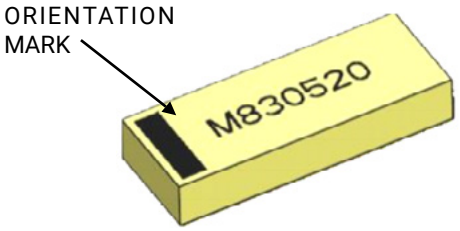




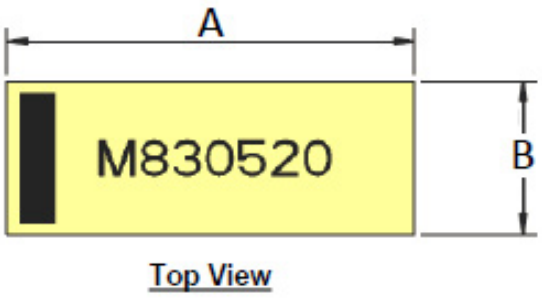
Product specifications subject to change without notice.

To optimize product designs using KYOCERA AVX SavviTM WLAN antennas, the PCB should use the recommended land pattern shown in the Figures below. The land patterns are composed of a 50 ohm line connected to each antenna feed point. Ground clearance around and under the antenna, as shown in the PCB layout below, is recommended in order to maximize the antenna's performance. The antenna should be located along an edge of the PCB.

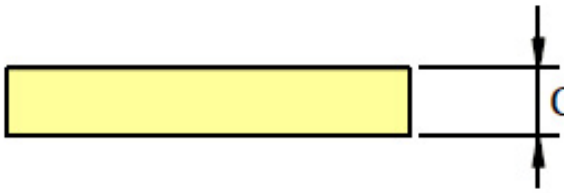
Antenna Pad Layout



Part Number	A (mm)	B (mm)	C (mm)
M830520	8.0 ± 0.2	3.0 ± 0.2	1.3 ± 0.1

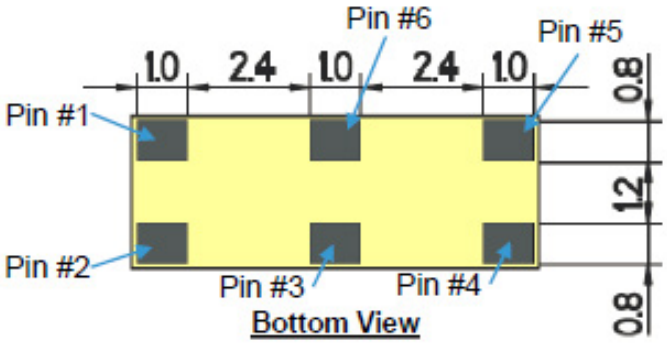


Top View



Height

Pin	Description
1	Feed
2	Ground
3	Dummy Pad
4	Dummy Pad
5	Dummy Pad
6	Dummy Pad

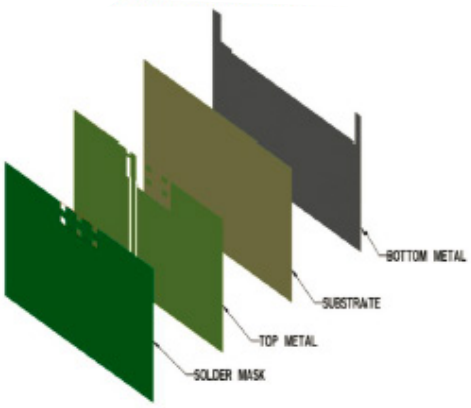


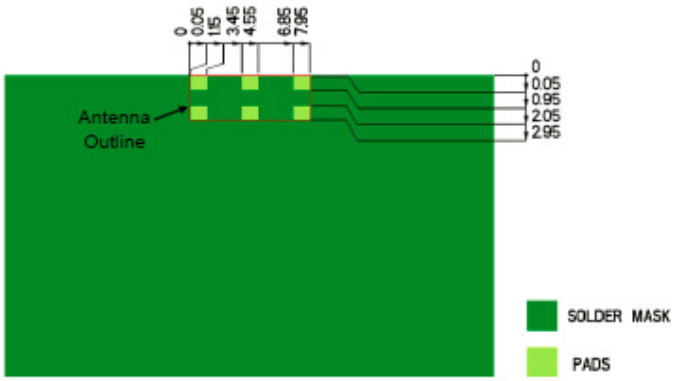
Bottom View

Product specifications subject to change without notice.

To optimize product designs using KYOCERA AVX SavviTM WLAN antennas, the PCB should use the recommended land pattern shown in the Figures below. The land patterns are composed of a 50 ohm line connected to each antenna feed point. Ground clearance around and under the antenna, as shown in the PCB layout below, is recommended in order to maximize the antenna's performance. The antenna should be located along an edge of the PCB.

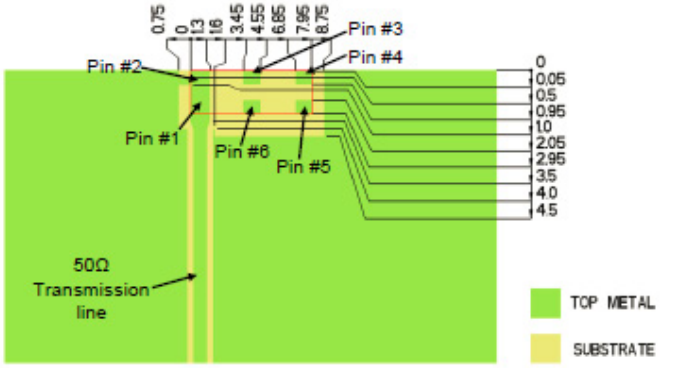
PCB Layout

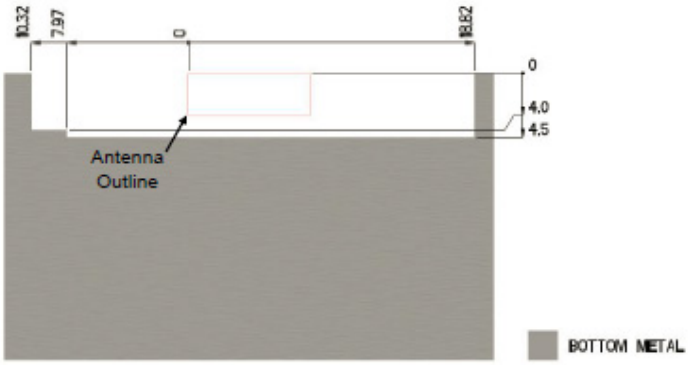




Pin Descriptions

Pin#	Description
1	Feed
2	Ground
3	Dummy Pad
4	Dummy Pad
5	Dummy Pad
6	Dummy Pad





KYOCERA AVX provides antenna layout files in .DXF format. This is a universal file format and should be converted into your specific file format.

Product specifications subject to change without notice.

FOLLOW US:

VISIT US AT WWW.KYOCERA-AVX.COM

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