Applications:
Embedded design
Handheld
Wireless Headsets
Tablets
Gateway
Access Point
Telematics
Tracking
M2M
Healthcare
Industrial Devices
Smart Grid
OBD-II
Media Players
Bluetooth
# Table of Contents

1. PURPOSE  
2. OVERVIEW  
3. DESIGN GUIDELINES  
   A. Introduction  
   B. Electrical Specifications  
   C. Mechanical Specifications  
   D. Antenna Dimension and Pad Layout  
   E. Antenna Footprint Layout  
   F. Typical Measured Data  
   G. Antenna Placement Guidelines on PCB  
   H. Antenna Tuning Guidelines  
      • Major Tuning Through the Tuning Pad Printed on the PCB  
      • Minor Tuning Through Matching Circuit Guidelines  
      • Change Antenna Location  
      • Extend Ground Width  
      • Increase the Gap between Antenna and Bottom Ground (On-Ground Test Condition)  
      • Increase the Gap between Antenna and Side Ground (Side-Ground Test Condition)  
4. MATERIAL SPECIFICATIONS  
5. PRODUCT TESTING  
6. MANUFACTURING AND ASSEMBLY GUIDELINES  
   A. Component Handling Recommendations  
   B. Paste Stencil Recommendations  
   C. Soldering Recommendation  
   D. Additional Manufacturing Recommendations  
   E. Cleaning Recommendations  
   F. Tape and Reel Specifications
RESTRICTED PROPRIETARY INFORMATION

The information disclosed herein is the exclusive property of Ethertronics Inc. and is not to be disclosed without the written consent of Ethertronics Inc. No part of this publication may be reproduced or transmitted in any form or by any means including electronic storage, reproduction, execution or transmission without the prior written consent of Ethertronics Inc. The recipient of this document by its retention and use agrees to respect the security status of the information contained herein.

Ethertronics may make changes to specifications and product descriptions at any time, without notice.

Ethertronics, Inc. assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

Copies of documents that are referenced in this document or other Ethertronics literature may be obtained by calling (858) 550-3820, email at ether.info@avx.com or the website www.ethertronics.com.

Contact your local sales office or manufacturers’ representative to obtain the latest specifications.

© 2019 Ethertronics. All rights reserved. The Ethertronics logo, Isolated Magnetic Dipole and Savvi are trademarks of Ethertronics. All other trademarks are the property of their respective owners. Product specifications subject to change without notice.
1. Purpose

This document provides information for incorporating Ethertronics’ Prestta standard Wi-Fi / BT / Zigbee Ceramic embedded SMT antenna into wireless products. Specifications, design recommendations, board layout, packaging, and manufacturing recommendations are included.

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

2. Overview

Product Selection Guide

<table>
<thead>
<tr>
<th>Antenna PN</th>
<th>Application</th>
<th>Antenna PN Application Type Typical Deliverable</th>
<th>Typical Deliverable Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>1001312</td>
<td>• WiFi 2.4 GHz • BT • Zigbee</td>
<td>• Ground Cleared • 2.4 GHz • Flexible antenna placement</td>
<td>• Antenna element only • SMT • 2.00 x 1.20 x 0.55 mm</td>
</tr>
</tbody>
</table>

Prestta Features and Benefits Summary

<table>
<thead>
<tr>
<th>Features</th>
<th>Benefits</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ceramic Antennas with SMD capability</td>
<td>• Flexibility in antenna placement with direct SMT on board&lt;br&gt;• Ease of manufacturing</td>
</tr>
<tr>
<td>Embedded Solutions for 2.4 GHz</td>
<td>• Eliminates external antennas&lt;br&gt;• More desirable form factors&lt;br&gt;• Can be used in Access Points, Routers, Gateways, Wireless Displays/TVs, and other consumer electronic devices</td>
</tr>
<tr>
<td>Compact Size and High Performance</td>
<td>• Fit on small PCB within small devices&lt;br&gt;• Comparable same level performance as in a large room</td>
</tr>
<tr>
<td>Ground Cleared Solution</td>
<td>• Enables flexibility in antenna placement within end device</td>
</tr>
<tr>
<td>Extensive Design Collateral and Apps Support</td>
<td>• Speeds development time</td>
</tr>
<tr>
<td>Standard “Off the Shelf” Product</td>
<td>• Standard “Off the Shelf” Product • Speeds development time and reduces costs by reducing NRE and custom development time</td>
</tr>
</tbody>
</table>
3. Design Guidelines

A. Introduction

The Prestta standard Wi-Fi / BT / Zigbee Ceramic embedded antenna can be designed into many wireless product types. The following sections explain Ethertronics’ recommended layouts to help the designer integrate the 1001312 antenna element into a device with optimum performance.

B. Electrical Specifications

Typical Characteristics Measurements taken on a 55 x 25 mm PCB

<table>
<thead>
<tr>
<th>Frequency</th>
<th>2400 – 2485 MHz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Peak Gain</td>
<td>1.88 dBi</td>
</tr>
<tr>
<td>Average Efficiency</td>
<td>80%</td>
</tr>
<tr>
<td>VSWR Match</td>
<td>1.8:1 max</td>
</tr>
<tr>
<td>Feed Point Impedance</td>
<td>50 ohms unbalanced</td>
</tr>
<tr>
<td>Polarization</td>
<td>Linear</td>
</tr>
<tr>
<td>Power Handling</td>
<td>0.5 Watt CW</td>
</tr>
</tbody>
</table>

C. Mechanical Specifications

<table>
<thead>
<tr>
<th>Ordering Part Number</th>
<th>1001312</th>
</tr>
</thead>
<tbody>
<tr>
<td>Size (mm)</td>
<td>2.00 x 1.20 x 0.55</td>
</tr>
<tr>
<td>Mounting</td>
<td>SMT</td>
</tr>
<tr>
<td>Weight (grams)</td>
<td>0.003</td>
</tr>
<tr>
<td>Packaging</td>
<td>Tape &amp; Reel, 1001312 – 1,000 pieces per reel</td>
</tr>
<tr>
<td>Demo Board</td>
<td>1001312-01</td>
</tr>
</tbody>
</table>
D. Antenna Dimension and Pad Layout

Figure 1 below shows the Antenna Dimensions and Pad Layout for 1001312

**Antenna Dimensions**
Typical antenna dimensions (mm)

<table>
<thead>
<tr>
<th>Part Number</th>
<th>A (mm)</th>
<th>B (mm)</th>
<th>C (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1001312</td>
<td>2.0 ± 0.3</td>
<td>1.2 ± 0.3</td>
<td>0.55 ± 0.2</td>
</tr>
</tbody>
</table>

---

**Pin Description**

- 1: Feed
- 2: Ground

*Pin #1 and Pin #2 are interchangeable.

---

Figure 1: Antenna Dimensions and Pad Layout for 1001312
E. Antenna Footprint Layout

Figure 2 below shows the 1001312 Antenna Footprint and Tuning Layout

<table>
<thead>
<tr>
<th>Pin#</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Feed</td>
</tr>
<tr>
<td>2</td>
<td>Ground</td>
</tr>
</tbody>
</table>

Matching Pi Network (Demo Board)

<table>
<thead>
<tr>
<th>Component</th>
<th>Value</th>
<th>Tolerance</th>
</tr>
</thead>
<tbody>
<tr>
<td>P1</td>
<td>4.7nH</td>
<td>±0.1nH</td>
</tr>
<tr>
<td>P2</td>
<td>DNI</td>
<td>N/A</td>
</tr>
<tr>
<td>S1</td>
<td>0Ω</td>
<td>N/A</td>
</tr>
<tr>
<td>R1</td>
<td>0Ω</td>
<td>N/A</td>
</tr>
<tr>
<td>R2 – R8</td>
<td>DNI</td>
<td>N/A</td>
</tr>
</tbody>
</table>

*Actual matching values depend on customer design

*ΩΩ may be added to shift frequency higher

Figure 2: 1001312 Antenna Footprint and Tuning Layout
F. Typical Measured Data

VSWR, Efficiency and Radiation Pattern

Figure 3 below shows the 1001312 Antenna Typical VSWR & Efficiency Plots on 55 x 25 mm PCB
Figure 4 below shows the 1001312 Antenna Typical Radiation Pattern Plots on 55 x 25 mm PCB

![VSWR and Efficiency Plots](image)

Figure 3: 1001312 Antenna Typical VSWR & Efficiency Plots on 55 x 25 mm PCB

![Radiation Pattern Plots](image)

Figure 4: 1001312 Antenna Typical Radiation Pattern Plots on 55 x 25 mm PCB Measured @ 2440 MHz
G. Antenna Placement Guidelines on PCB

The 1001312 is a ceramic antenna element that can be mounted onto any PCB using Ethertronics’ recommended footprint layout and ground layout, including proper PCB size. This tiny and compact antenna solution can easily be integrated into a radio module reference design. Antenna placement on module reference design will still need to follow Ethertronics’ recommended antenna location, otherwise the module’s PCB ground size may not be able to meet the antenna’s requirement.

- Antenna should always be placed along the edge of the board unless there are special conditions preventing this.
- The antenna can be placed on either the top or bottom side of the PCB. The recommended antenna location is at the center of long edge, and the suggested long edge length: “A” ≥ 55mm.

Figure 5 shows the optimal antenna placement for 1001312

![Figure 5: optimal single antenna placement for 1001312](image-url)
H. Antenna Tuning Guidelines

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

Additionally, any plastic loading needs to be considered for fine tuning steps. For example, 2mm thickness PC/ABS material will detune the antenna frequency by ~30MHz.

The following methods can be applied to solve the above effects
- Major Tuning Through the Tuning Pad Printed on the PCB
- Minor Tuning Through Matching Circuit Guidelines
- Change Antenna Location
- Extend Ground Width
- Increase the Gap between Antenna and Bottom Ground (On-Ground Test Condition)
- Increase the Gap between Antenna and Side Ground (Side-Ground Test Condition)

**Major Tuning Through the Tuning Pad Printed on the PCB**

Antenna Tuning Pad can shift frequency resonance lower or higher by adding/removing 0ohm resistors on the tuning pad layout. Adding 0ohm resistors shortens the length of the slot which shifts antenna frequency higher. In opposite, removing 0ohm resistors increases the length of the slot which shifts antenna frequency lower. The advantage of using a tuning pad layout structure is an antenna can be tuned on the board directly to avoid or reduce the re-spin times of the customer PCB.

Figure 6 shows 1001312 Tuning Layout Structure
Figure 7 shows 1001312 Tuning Pad Configurations

![Figure 6: 1001312 Tuning Layout Structure](image)
**Ceramic Antenna**

**2400-2485 MHz**

<table>
<thead>
<tr>
<th>Low Band Tuning Pad Length</th>
<th>R1</th>
<th>R2</th>
<th>R3</th>
<th>R4</th>
<th>R5</th>
<th>R6</th>
<th>R7</th>
<th>R8</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pad Slot 1 (Default)</td>
<td>0Ω</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
</tr>
<tr>
<td>Pad Slot 2</td>
<td>DNI</td>
<td>0Ω</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
</tr>
<tr>
<td>Pad Slot 3</td>
<td>DNI</td>
<td>DNI</td>
<td>0Ω</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
</tr>
<tr>
<td>Pad Slot 4</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>0Ω</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
</tr>
<tr>
<td>Pad Slot 5</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>0Ω</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
</tr>
<tr>
<td>Pad Slot 6</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>0Ω</td>
<td>DNI</td>
<td>DNI</td>
</tr>
<tr>
<td>Pad Slot 7</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>0Ω</td>
<td>DNI</td>
</tr>
<tr>
<td>Pad Slot 8</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>0Ω</td>
</tr>
<tr>
<td>Pad Slot 9</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
<td>DNI</td>
</tr>
</tbody>
</table>

**Figure 7** : 1001312 Tuning Pad Configurations
**Minor Tuning Through Matching Circuit Guidelines**

Performance can be also improved by tuning the matching circuit. Optimum matching values may vary with different boards transmission line design, different antenna location, different PCB size and antenna working environments. Nevertheless, antenna performance can be improved by modifying the tuning pad and optimizing the matching components accordingly.

Following is a matching application demonstration. Antenna performance is detuned when the 1001312 antenna module is placed on a piece of ground with four different gaps (1mm/3mm/5mm/7mm), and changing matching value for each state will be able to help to tune antenna in band.

Figure 8 shows 1001312 Matching Value with Different Gaps
Figure 9 shows 1001312 Frequency Shift Before Matching and After Matching
Change Antenna Location

Antenna performance can vary with different antenna locations placement. In general, the preferred location is in the center of one PCB edge. If within a tiny PCB, 1001312 antenna will be recommended locating in the center of long edge which length should be exceed 55mm. The two studies below shows the antenna performance variation with different antenna locations based on two PCB sizes.

- Study 1 (both edge > 55 mm):
  - Based on 100 x 110 mm board,
    - check the antenna performance change by placing antenna in three different locations of one edge (Left side; Center side; Right side)
- Study 2 (long edge ≥ 55mm & short edge < 55mm):
  - Based on 55 x 23 mm board,
  - check the antenna performance change by placing antenna in the center of each edge (Location 1: center of short edge; Location 2: center of long edge)

Figure 10 shows the Study1 1001312 tested with three different locations on a 100 x 110 mm PCB
Figure 11 shows the Study1 1001312 performance vary with different locations on the 100 x 110 mm PCB.
Figure 12 shows the Study2 1001312 tested with two different locations on a 55 x 23 mm PCB
Figure 13 shows the Study2 1001312 performance vary with different locations on the 55 x 23 mm PCB.

Figure 10 : Study1 1001312 tested with three different locations on a 100 x110 mm PCB

Figure 11 : Study1 1001312 performance vary with different locations on the 100 x 110 mm PCB
Wi-Fi / BT / Zigbee Embedded Ceramic Antenna
2400-2485 MHz

Figure 12 : Study2 1001312 tested with two different locations on a 55 x 23 mm PCB

Figure 13 : Study2 1001312 performance vary with different locations on the 55 x 23 mm PCB
**Extend Ground Width**

The PCB ground width is a critical factor to the 1001312 antenna performance. In general, the width needs to reach 55mm or greater while the antenna is located in the center edge.

Figure 14 shows 1001312 tested with different ground width
Figure 15 shows the 1001312 performance vary with different ground width
**Increase the Gap between Antenna and Bottom Ground (On-Ground Test Condition)**

Based on On-Ground test condition (antenna or antenna module is located on a piece of ground), a certain gap between antenna and ground plane is necessary to be added. In general, the larger gap distance, the higher antenna performance will be achieved.

The study below shows how the gap effect on antenna performance.

Figure 16 shows 1001312 antenna module is located on a ground plane with different gaps. Figure 17 shows the 1001312 performance change with different On-Ground gaps.

![Figure 16: 1001312 antenna module is located on a ground plane with different gaps](image)

![Figure 17: 1001312 performance change with different On-Ground gaps.](image)
Increase the Gap between Antenna and Side Ground (Side-Ground Test Condition)

Based on Side-Ground test condition (antenna or antenna module is located beside a piece of ground), a certain gap between antenna and ground plane is necessary to be added. In general, the larger gap distance, the higher antenna performance will be achieved.

The study below shows how the gap effect on antenna performance.

Figure 18 shows 1001312 antenna module is located beside a ground plane with different gaps. Figure 19 shows the 1001312 performance change with different Side-Ground gaps.

Figure 16 : 1001312 antenna module is located beside a ground plane with different gaps.

Figure 17 : 1001312 performance change with different Sid-Ground gaps.
4. Material Specifications

<table>
<thead>
<tr>
<th>Item</th>
<th>Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>Metal Element</td>
<td>Silver ink</td>
</tr>
<tr>
<td>Composite Structure</td>
<td>Ceramic</td>
</tr>
<tr>
<td>Contact Finish</td>
<td>Ni Au</td>
</tr>
</tbody>
</table>

5. Product Testing

Ethertronics’ antennas comply with RoHS directives. Ethertronics’ antennas undergo product qualification testing as part of the product development process. The following are the core tests used to qualify the ceramic antennas.

Table 1 Product Qualification Tests

<table>
<thead>
<tr>
<th>NO</th>
<th>Test Type</th>
<th>Items</th>
<th>Test condition</th>
<th>Test method</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>High Temp.</td>
<td>85°C±3°C 120hr ±2hr</td>
<td></td>
<td><strong>Step 1:</strong> Test VSWR by jig.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 2:</strong> Put it in the chamber.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 3:</strong> Test it like this picture which explains temp. cycle.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 4:</strong> Test VSWR after 1hr in normal Temp. &amp; normal Humidity.</td>
</tr>
<tr>
<td>2</td>
<td>Low Temp.</td>
<td>-40°C±3°C 120hr ±2hr</td>
<td></td>
<td><strong>Step 1:</strong> Test VSWR by jig.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 2:</strong> Put it in the chamber.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 3:</strong> Test it like this picture which explains temp. cycle.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 4:</strong> Test VSWR after 1hr in normal Temp. &amp; normal Humidity.</td>
</tr>
<tr>
<td>3</td>
<td>Environment test</td>
<td>High Temp. &amp; High Humidity</td>
<td>85°C±3°C 120hr ±2hr</td>
<td><strong>Step 1:</strong> Test VSWR by jig.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 2:</strong> Put it in the chamber.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 3:</strong> Test it like this picture which explains temp. cycle.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 4:</strong> Test VSWR after 1hr in normal Temp. &amp; normal Humidity.</td>
</tr>
<tr>
<td>4</td>
<td>Salt Spray</td>
<td>Nacl 5% 35°C 72hr</td>
<td></td>
<td><strong>Step 1:</strong> Test VSWR by jig.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 2:</strong> Put it in the chamber.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 3:</strong> Start test.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 4:</strong> Wash the samples.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 5:</strong> Test VSWR after 1hr in normal Temp. &amp; normal Humidity.</td>
</tr>
<tr>
<td>5</td>
<td>T</td>
<td>120°C PC RH=100% 96hr</td>
<td></td>
<td><strong>Step 1:</strong> Test VSWR by jig.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 2:</strong> Put it in the chamber.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 3:</strong> Test it like this picture which explains temp. cycle.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Step 4:</strong> Test VSWR after 1hr in normal Temp. &amp; normal Humidity.</td>
</tr>
<tr>
<td>NO</td>
<td>Test Type</td>
<td>Items</td>
<td>Test condition</td>
<td>Test method</td>
</tr>
<tr>
<td>----</td>
<td>-----------------</td>
<td>------------------------</td>
<td>---------------------------------------------</td>
<td>-------------------------------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
| 6  | Environment test| Thermal shock          | -40°C to 3°C/30min, 85°C to 3°C/30min, 32cycle| 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 1 hr in normal Temp. & no normal Humidity |
| 7  | 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  | Vibration       | -Frequency: 10~500hz  
-Acceleration: 10*9.8m/s² (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  | Mechanical Test | 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 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. |
| 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. |
6. Manufacturing and Assembly Guidelines

Ethertronics’ 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 Ethertronics, based on successful manufacturing processes.

These ceramic antennas are designed for automated pick and place surface mounting. However, as with any SMT device, Ethertronics 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 be used to pick-up, transfer and mount the antennas.

Ethertronics’ 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, Ethertronics antennas are dry-packed.

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

Paste Stencil Recommendation

Ethertronics 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

Ethertronics requires using glue as an under fill for increased adhesion strength. Please contact Ethertronics 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 Ethertronics.

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 Ethertronics’ 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°C. 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.