AirPrime BX Series

Customer Process Guidelines

41112096
3.0
April 01, 2019

Important Notice

Due to the nature of wireless communications, transmission and reception of data can never be guaranteed. Data may be delayed, corrupted (i.e., have errors) or be totally lost. Although significant delays or losses of data are rare when wireless devices such as the Sierra Wireless modem are used in a normal manner with a well-constructed network, the Sierra Wireless modem should not be used in situations where failure to transmit or receive data could result in damage of any kind to the user or any other party, including but not limited to personal injury, death, or loss of property. Sierra Wireless accepts no responsibility for damages of any kind resulting from delays or errors in data transmitted or received using the Sierra Wireless modem, or for failure of the Sierra Wireless modem to transmit or receive such data.

Safety and Hazards

Do not operate the Sierra Wireless modem in areas where cellular modems are not advised without proper device certifications. These areas include environments where cellular radio can interfere such as explosive atmospheres, medical equipment, or any other equipment which may be susceptible to any form of radio interference. The Sierra Wireless modem can transmit signals that could interfere with this equipment. Do not operate the Sierra Wireless modem in any aircraft, whether the aircraft is on the ground or in flight. In aircraft, the Sierra Wireless modem **MUST BE POWERED OFF**. When operating, the Sierra Wireless modem can transmit signals that could interfere with various onboard systems.

Note: *Some airlines may permit the use of cellular phones while the aircraft is on the ground and the door is open. Sierra Wireless modems may be used at this time.*

The driver or operator of any vehicle should not operate the Sierra Wireless modem while in control of a vehicle. Doing so will detract from the driver or operator's control and operation of that vehicle. In some states and provinces, operating such communications devices while in control of a vehicle is an offence.

Limitations of Liability

This manual is provided "as is". Sierra Wireless makes no warranties of any kind, either expressed or implied, including any implied warranties of merchantability, fitness for a particular purpose, or noninfringement. The recipient of the manual shall endorse all risks arising from its use.

The information in this manual is subject to change without notice and does not represent a commitment on the part of Sierra Wireless. SIERRA WIRELESS AND ITS AFFILIATES SPECIFICALLY DISCLAIM LIABILITY FOR ANY AND ALL DIRECT, INDIRECT, SPECIAL, GENERAL, INCIDENTAL, CONSEQUENTIAL, PUNITIVE OR EXEMPLARY DAMAGES INCLUDING, BUT NOT LIMITED TO, LOSS OF PROFITS OR REVENUE OR ANTICIPATED PROFITS OR REVENUE ARISING OUT OF THE USE OR INABILITY TO USE ANY SIERRA WIRELESS PRODUCT, EVEN IF SIERRA WIRELESS AND/OR ITS AFFILIATES HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES OR THEY ARE FORESEEABLE OR FOR CLAIMS BY ANY THIRD PARTY.

Notwithstanding the foregoing, in no event shall Sierra Wireless and/or its affiliates aggregate liability arising under or in connection with the Sierra Wireless product, regardless of the number of events, occurrences, or claims giving rise to liability, be in excess of the price paid by the purchaser for the Sierra Wireless product.

Patents

This product may contain technology developed by or for Sierra Wireless Inc.

This product includes technology licensed from QUALCOMM®.

This product is manufactured or sold by Sierra Wireless Inc. or its affiliates under one or more patents licensed from MMP Portfolio Licensing.

Copyright

© 2019 Sierra Wireless. All rights reserved.

Trademarks

Sierra Wireless®, AirPrime®, AirLink®, AirVantage®, WISMO®, ALEOS® and the Sierra Wireless and Open AT logos are registered trademarks of Sierra Wireless, Inc. or one of its subsidiaries.

Watcher® is a registered trademark of NETGEAR, Inc., used under license.

Windows® and Windows Vista® are registered trademarks of Microsoft Corporation.

Macintosh® and Mac OS X® are registered trademarks of Apple Inc., registered in the U.S. and other countries.

QUALCOMM® is a registered trademark of QUALCOMM Incorporated. Used under license.

Other trademarks are the property of their respective owners.

Contact Information

Sales information and technical support, including warranty and returns	Web: sierrawireless.com/company/contact-us/ Global toll-free number: 1-877-687-7795 6:00 am to 5:00 pm PST
Corporate and product information	Web: sierrawireless.com

Document History

Version	Date	Updates
1.0	May 03, 2018	Creation
1.1	May 15, 2018	Updated 2.2.2 Marking Description
2.0	October 15, 2018	Updated: • Figure 4 Recommended Footprint – Copper Layout • Figure 5 Recommended Solder Resist and Paste Mask Layout • 3.2 Layout Recommendations
3.0	April 01, 2019	Updated: • 2.1.1 Storage Condition • Table 1 Tape and Reel Specifications • Chapters 3 and 4 (reorganized) • Terminology – replaced “component” with “module” where appropriate

Contents

1. INTRODUCTION 7
    - 1.1. Overview 7
    - 1.2. Reference Documents 7
2. HANDLING 8
    - 2.1. Storage and Handling 8
        - 2.1.1. Storage Condition 8
        - 2.1.2. ESD 8
        - 2.1.3. Moisture Sensitivity 8
    - 2.2. Component Package 9
        - 2.2.1. Package Description 9
        - 2.2.2. Marking Description 9
    - 2.3. Component Packing 10
        - 2.3.1. Packing Description 10
        - 2.3.2. Packing Label 10
3. CUSTOMER PRODUCT DESIGN 11
    - 3.1. PCB Surface Finish 11
    - 3.2. Layout Recommendations 11
    - 3.3. Module Footprint 11
        - 3.3.1. Footprint 11
        - 3.3.2. Solder Mask 13
4. BOARD MOUNTING GUIDELINES 14
    - 4.1. Lead-Free Process 14
    - 4.2. Stencil Design 14
    - 4.3. Solder Reflow Profile 16
    - 4.4. Washing and Potting 17
5. REWORK GUIDELINES 18
    - 5.1. Module Removal 18
    - 5.2. Pad Redress 18
    - 5.3. Solder Paste Deposit 19
    - 5.4. New Module Placement 19
    - 5.5. New Module Soldering 19

List of Figures

Figure 1. Marking Example 9
Figure 2. Tape and Reel 10
Figure 3. Packing Label 10
Figure 4. Recommended Footprint – Copper Layout 12
Figure 5. Recommended Solder Resist and Paste Mask Layout 15
Figure 6. Recommended Reflow Profile 16

1. Introduction

1.1. Overview

This document presents guidelines for the industrial assembly of an AirPrime BX Embedded Module on an application.

1.2. Reference Documents

[1] AirPrime BX310x Product Technical Specification

Reference number: 41111444

[2] JEDEC standard JESD625, Requirements for Handling Electrostatic Discharge – Sensitive (ESDS) Devices

[3] ANSI/ESD S20.20: Protection of Electrical and Electronics Parts, Assemblies and Equipment

[4] IPC/JEDEC J-STD-033 – Handling, Packing, Shipping and Use of Moisture / Reflow Sensitive Surface Mount Devices

2. Handling

2.1. Storage and Handling

2.1.1. Storage Condition

AirPrime BX modules can be stored in the following manner:

- In their sealed, original packages;
- Up to 1 year;
- Temperature ≤ +40°C; RH ≤ 90%.

  For optimal results, the recommended storage temperature is +20°C ± 10°C.

2.1.2. ESD

The AirPrime BX module is ESD sensitive. For ESD level information, refer to the corresponding Product Technical Specification of each product as listed in section 1.2 Reference Documents.

It is recommended to use standard ESD precautions, as described in the following standards:

- JEDEC standard JESD625, Requirements for Handling Electrostatic Discharge-Sensitive (ESDS) Devices
- ANSI/ESD S20.20: Protection of Electrical and Electronics Parts, Assemblies and Equipment

2.1.3. Moisture Sensitivity

The AirPrime BX is sensitive to moisture absorption:

- MSL 3, 245°C, 2 reflows allowed on customer PCB; see additional details in section 4.2 Solder Reflow Profile.

Caution: *If the tape & reel vacuum pack is open for more than 168h, material should be baked at 40°C for 13 days. If parts are on tray, baking conditions are 24 hours minimum at 85°C.*

It is recommended to follow the standard MSL procedure, as described in the following standard:

- IPC/JEDEC J-STD-033 - Handling, Packing, Shipping and Use of Moisture / Reflow Sensitive Surface Mount Devices.

2.2. Component Package

2.2.1. Package Description

The AirPrime BX is a scalable QFN (quad flat no lead) package with 70 terminals. Module size depends on the product (11.5x9.5 mm for BXxxx0 and 11.5x13.5 mm for BXxxx5), but bottom footprints are compatible with 9.5x11.5 footprint, pitch 0.65 mm.

Module thickness is 2.4 mm

The terminals include:

- 54 signal pads
- 16 ground pads

The PCB material is FR4. Plating is NiAu (3 µm < Nickel < 8 µm and 0.05 < Gold < 0.13 µm).

For additional information, refer to the corresponding Product Technical Specification of each product as listed in section 1.2 Reference Documents.

2.2.2. Marking Description

Marking contents and marking methods on the module may differ between each variant of the product family.

The marking method can be via paper label or laser marking. For variants that use laser marking, there are rare instances where these modules are marked with a paper label instead.

Common label content includes:

- Model name
- FSN barcode
- Base MAC address barcode
- Pin 1 indicator
- Country of fabrication

Note: Regulatory compliancy markings are not present due to space limitations. Details regarding regulatory numbers are available in the Product Technical Specification.

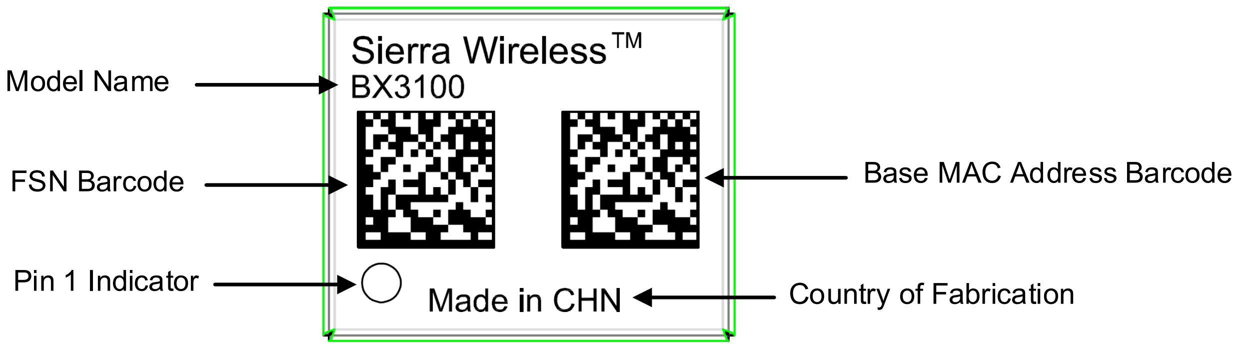


Figure 1. Marking Example

2.3. Component Packing

2.3.1. Packing Description

The AirPrime BX is delivered in tape and reel.

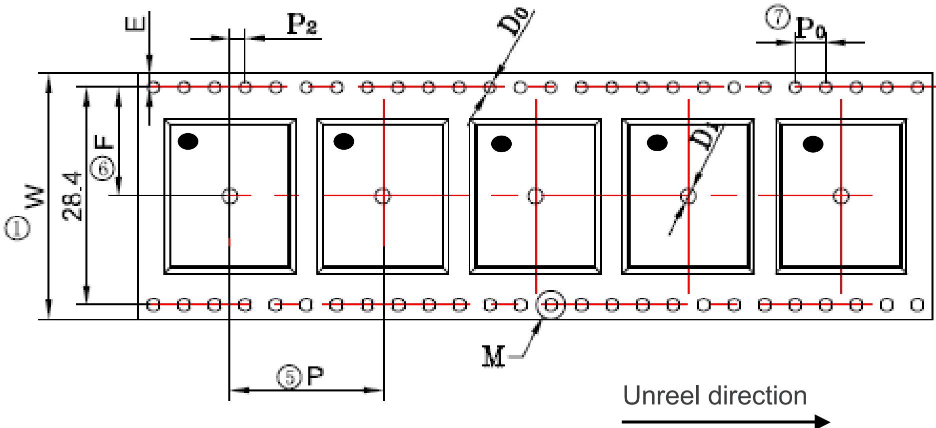


Figure 2. Tape and Reel

Table 1. Tape and Reel Specifications

Product	Quantity per Tape and Reel	P	P0	W	Pin 1 Orientation
BX3100	1500	16.0 mm	4.0 mm	24.0 mm	Left top corner (see Figure 2)
BX3105	1200	20.0 mm	4.0 mm	24.0 mm	Right bottom corner

2.3.2. Packing Label

Product Module: BX3100

MANUFACTURER PN:

XXXXXXX

DATE CODE: **1810** 1810

QTY: **1500** 1500

REEL ID: **BXXXXYYYYMMDDSSSS**

BXXXXYYYYMMDDSSSS

Seal date: **2018-03-07 08:57** **MADE IN CHINA**

Figure 3. Packing Label

3. Customer Product Design

3.1. PCB Surface Finish

The PCB surface finish recommended is Electroless Nickel, immersion Gold. Organic Solderability Preservative (OSP) may also be used.

Caution: *Hot Air Solder Leveled finish (HASL) is not recommended because the process does not give consistent solder volumes on each pad because of poor pad flatness.*

3.2. Layout Recommendations

Sierra Wireless' layout recommendations include:

- A GROUND area under the BX module. This ground area should be a whole area of copper with proper ground vias to provide a good grounding system between the application and the embedded module and improved thermal dissipation. It should be covered by solder resist on the non-soldered area.

  The ground vias may be micro-vias, filled or unfilled. Through-holes can be used in between each of the 16 ground pads (under the solder resist).
- There should not be any SIGNAL trace or hole / micro-via under the module.
- The antenna pad and its track should be adapted according to RF constraints, based on customer layout. Refer to each corresponding Product Technical Specification for more details.
- Allow enough space to accommodate larger BX modules (the BX3105, for example).
- Leave a component-free area of 1 mm around the module for accessing the surrounding components.

3.3. Module Footprint

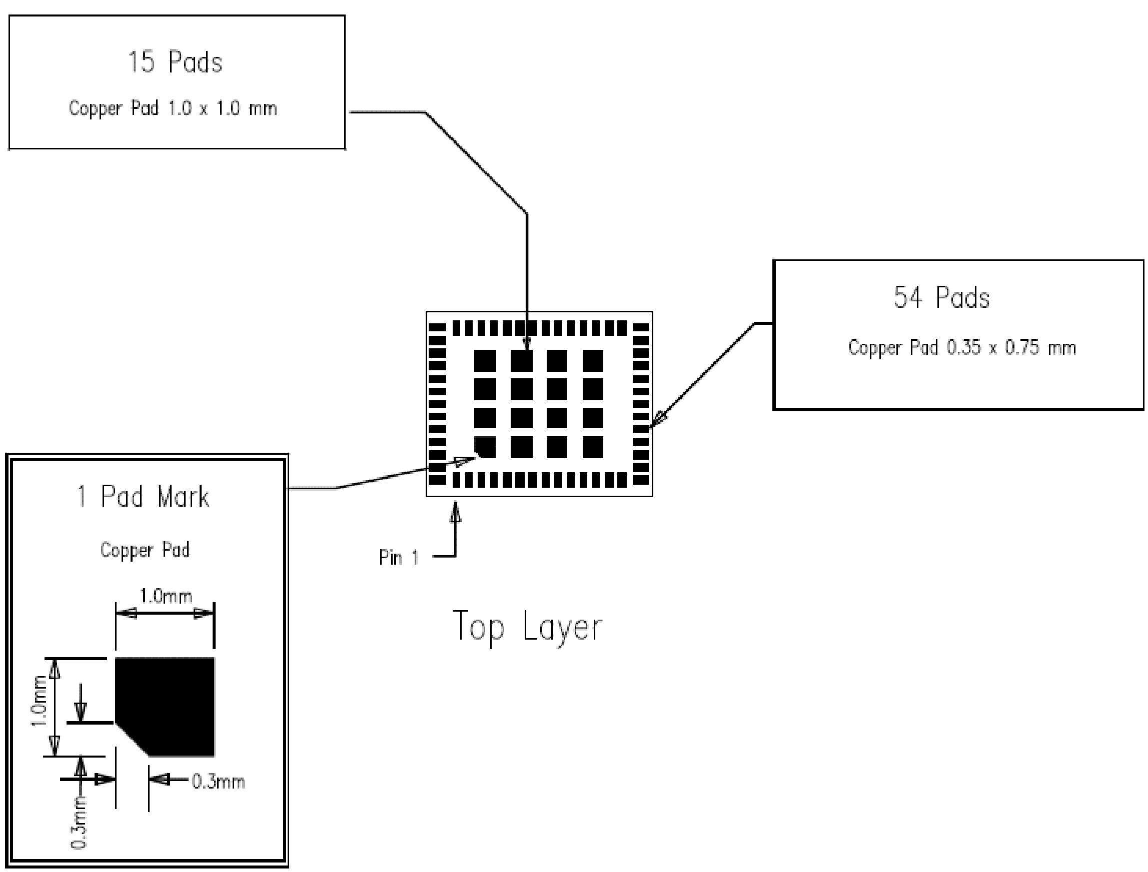
3.3.1. Footprint

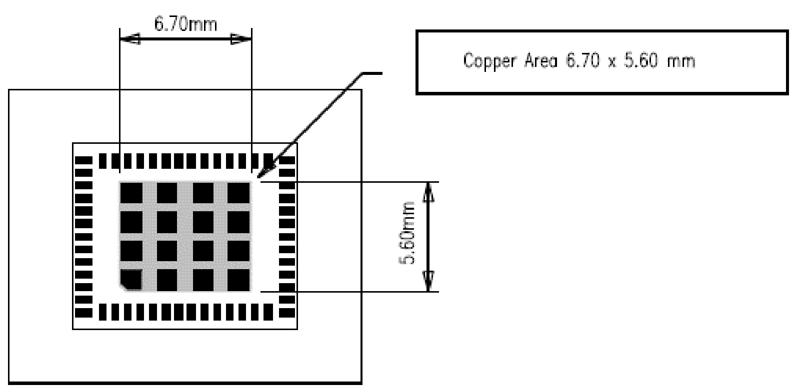
To produce high assembly yields and a reliable solder joint, the footprint on the customer application board should match Figure 4 below.

Note that:

- The 16 inner pads are ground pads.
- Manufacturing tolerance for copper pads is 30 µm.

Mechanical drawings of the module footprint (including dimensions and pitch) are available in the Product Technical Specification of each product as listed in section 1.2 Reference Documents.





Top Layer

Figure 4. Recommended Footprint – Copper Layout

3.3.2. Solder Mask

The pads on the printed circuit board are either Solder Mask Defined (SMD) or Non-Solder Mask Defined (NSMD).

Since the copper etching process has tighter control than the solder masking process, NSMD pads are preferred over SMD pads.

Moreover, NSMD pads with solder mask opening larger than the metal pad size also improve the reliability of solder joints, as this limits the stress concentration at the solder-to mask corner interface.

Recommendations:

- For external pads, the solder mask opening should be 100–150 μm larger than the pad, resulting in 50–75 μm clearance between the copper pad and solder mask. This allows for solder mask registration tolerances, depending upon the PCB fabricator's capabilities.
- For ground pads, SMD pads should be used if a copper ground area is under the module as described in section 3.2 Layout Recommendations.
- Recommended solder mask thickness on top copper is 10–30 μm.

Refer to Figure 5 for recommended solder resist and paste mask layout.

4. Board Mounting Guidelines

This section presents information and recommendations for the industrial assembly of the module on the application.

Note: The module should be assembled by reflow process.

The polarity mark is indicated by a cut corner on one of the 16 ground pads on the bottom side of the module.

4.1. Lead-Free Process

In compliance with directive 2011/65/CE and its amendments, Sierra Wireless products do not contain the following hazardous substances.

Table 2. Restricted Substances

Substance Name	Limit (%)
Lead	0.1%
Mercury	0.1%
Cadmium	0.01%
Hexavalent chromium	0.1%
Polybrominated biphenyls (PBB)	0.1%
Polybrominated diphenyl ethers (PBDE)	0.1%
Bis(2-ethylhexyl) phthalate (DEHP)	0.1%
Butyl benzyl phthalate (BBP)	0.1%
Dibutyl phthalate (DBP)	0.1%
Diisobutyl phthalate (DIBP)	0.1%

The module is manufactured with RoHS-compliant components and processes.

It is recommended that the customer process be ROHS-compliant.

4.2. Stencil Design

The recommended stencil thickness is 125 µm.

The proposed stencil design is presented in the figure below.

It is highly recommended to monitor the solder paste height, registration and proper placement during the squeegee printing.

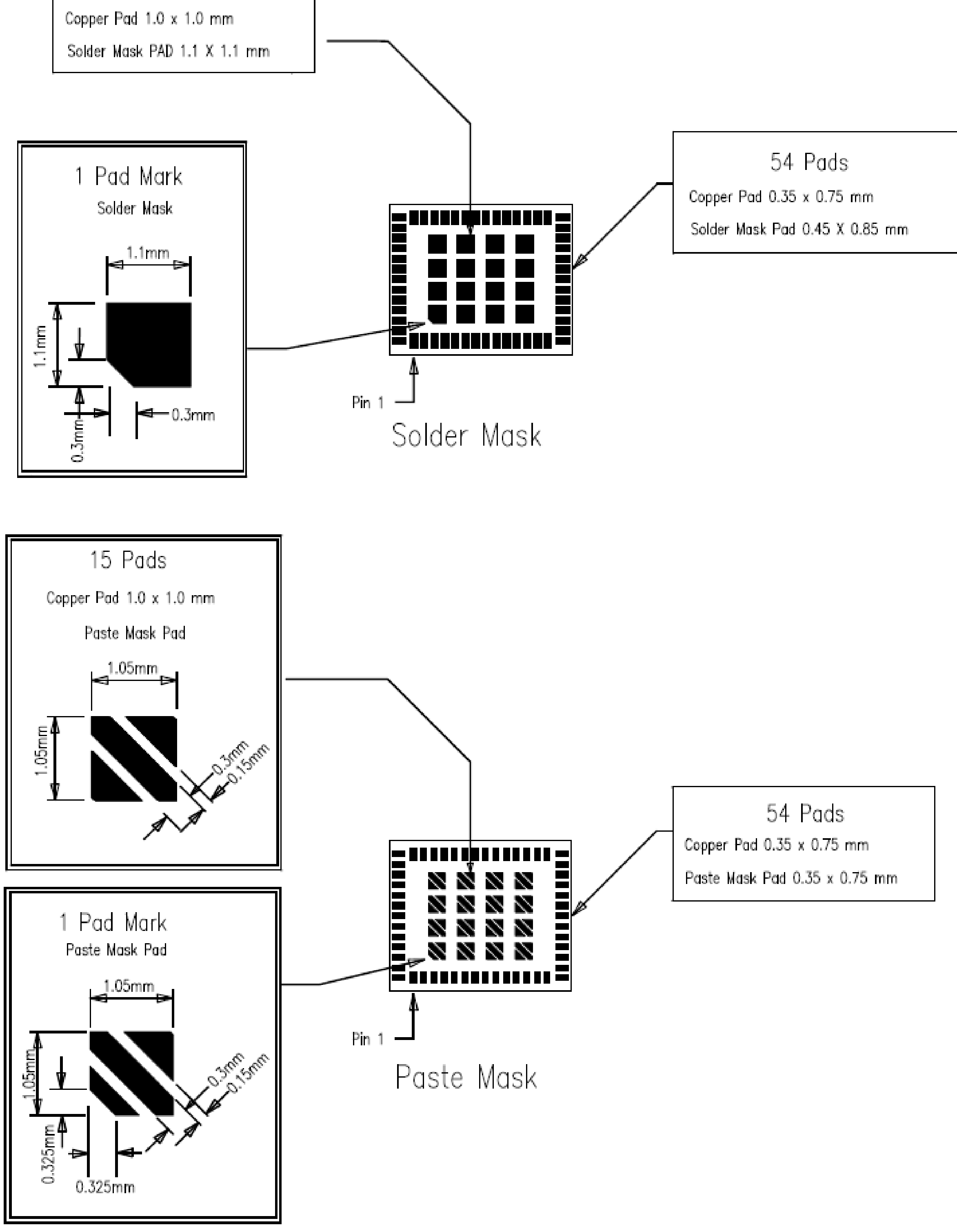


Figure 5. Recommended Solder Resist and Paste Mask Layout

4.3. Solder Reflow Profile

Lead-free SMT reflow profiles should be used to surface mount the module.

The reflow profile depends on PCB density and type of solder paste being used. The paste manufacturer's recommendation should also be considered to determine the proper reflow profile.

The solder reflow profile specified in Table 3 is a mandatory requirement to ensure reliable assembly.

Table 3. Solder Reflow Profile

Peak Temperature	245°C max
Number of reflow cycles	2

Caution: *It is recommended to use only one reflow cycle for module assembly.*

If repairs or other rework are performed on the customer board near the BX modules, care must be taken to ensure the module is not reflowed.

Figure 6 is a reflow profile example.

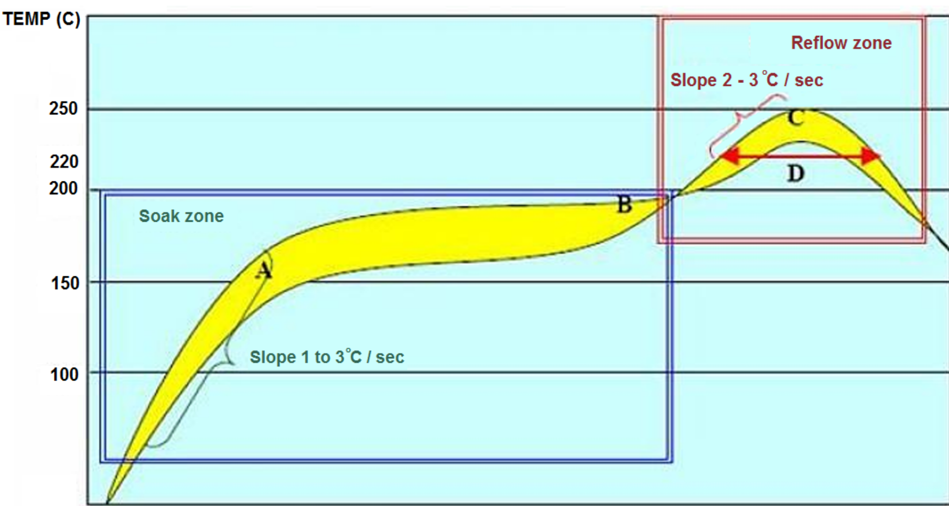


Figure 6. Recommended Reflow Profile

Additional recommendations are presented in Table 4 for consideration.

Table 4. Additional Reflow Recommendations

Factor	Recommendation
Max slope at soak time	1 to 3°C / sec
Max slope at reflow zone	2 to 3°C / sec
Soak time (between A and B: 150 and 190°C)	60 to 120 sec
Reflow time (D: over 220°C)	40 to 60 sec
Max temperature (C)	235 – 245°C
Cooling down slope	1 to 3°C / sec

Note: It is recommended to perform reflow in a nitrogen atmosphere.

4.4. Washing and Potting

Washing considerations:

- Water wash is not recommended with shielded Sierra Wireless embedded modules due to difficulty in ensuring proper drying under the shield.
- Use of ultrasonic process should be avoided as it can damage the module's quartz crystal components.

Potting considerations:

- Sierra Wireless has not performed potting qualification tests on BX modules.
- Customers should select a potting material that will not lower RF performance. After potting a module, RF performance should be re-checked to guarantee that there is no degradation compared to nominal values.

5. Rework Guidelines

Rework tools and operating parameters are customer/application specific. Rework tools, heating profiles and the rework process should be tailored to these specific needs for optimum results.

Prior to any rework:

- If the module floor life has been exceeded, pre-baking the PCB is strongly recommended to remove moisture from the assembly. (See document [4] IPC/JEDEC J-STD-033 – Handling, Packing, Shipping and Use of Moisture / Reflow Sensitive Surface Mount Devices, paragraph 6 — Board rework.)

  If possible for the PCB and the other components of the board, apply the same baking conditions as indicated in section 2.1.3 Moisture Sensitivity).

  The pre-baking process will prevent damage to any component due to moisture vapor pressures caused during reflow.
- Before removing the module, the module's metal shielding must be glued to the module substrate, by using glue able to withstand reflow profile.

5.1. Module Removal

The step consists of reflowing the solder joints attaching components to the PCB. Ideally, the reflow profile for part removal should be the same as the one used for part attachment. However, the time above liquidus can be reduced if the reflow is complete.

Removal process recommendations:

- Heat the board from the bottom side of the module (using convective heaters and hot gas) and/or from the top side of the module (using hot air or IR).
- Use special nozzles or IR lens to direct the heating in the module area and minimize the heating of adjacent components.
- Avoid excessive hot air flow to prevent the module from overheating.

Once the joints have reflowed, the vacuum lift-off should be automatically engaged for pick-up during the transition from reflow to cool down

Warning: *If heating conditions are not properly controlled during manual hot removal from PCB assembly, package integrity can be damaged from overheating.*

5.2. Pad Redress

After the module has been removed, the site and pads need to be cleaned properly.

Recommendations:

- Remove residual solder from the site using a combination of a blade style conductive tool and a fluxed desoldering braid.
- Once the residual solder has been removed, clean the land pads with an appropriate solvent.

  The required solvent is usually specific to the type of solder paste used in the original assembly, so the paste manufacturer's recommendations should be followed.

5.3. Solder Paste Deposit

Once the PCB is properly cleaned and inspected, solder paste should be applied on the solder land (on the module itself or on the customer PCB) with a mini-stencil which has same thickness and apertures as the stencil used for original attachment.

5.4. New Module Placement

A slip-beam optical system should be used to align the module to the PCB. This method will display an image of the land pad overlaid on the mating footprint and aid in proper alignment. Similar to paste printing, the alignment should be done under magnification of 50x to 100x.

5.5. New Module Soldering

The reflow profile developed during original attachment or removal should be used to attach the new module.