



Customer Process Guidelines

WPx5/WP76/WP77

Important Notice

Information relating to this product and the application or design described herein is believed to be reliable, however such information is provided as a guide only and Semtech assumes no liability for any errors in this document, or for the application or design described herein.

Semtech reserves the right to make changes to the product or this document at any time without notice. Buyers should obtain the latest relevant information before placing orders and should verify that such information is current and complete. Semtech warrants performance of its products to the specifications applicable at the time of sale, and all sales are made in accordance with Semtech's standard terms and conditions of sale.

SEMTECH PRODUCTS ARE NOT DESIGNED, INTENDED, AUTHORIZED OR WARRANTED TO BE SUITABLE FOR USE IN LIFE-SUPPORT APPLICATIONS, DEVICES OR SYSTEMS, OR IN NUCLEAR APPLICATIONS IN WHICH THE FAILURE COULD BE REASONABLY EXPECTED TO RESULT IN PERSONAL INJURY, LOSS OF LIFE OR SEVERE PROPERTY OR ENVIRONMENTAL DAMAGE. INCLUSION OF SEMTECH PRODUCTS IN SUCH APPLICATIONS IS UNDERSTOOD TO BE UNDERTAKEN SOLELY AT THE CUSTOMER'S OWN RISK. Should a customer purchase or use Semtech products for any such unauthorized application, the customer shall indemnify and hold Semtech and its officers, employees, subsidiaries, affiliates, and distributors harmless against all claims, costs damages and attorney fees which could arise.

The Semtech name and logo are registered trademarks of the Semtech Corporation. All other trademarks and trade names mentioned may be marks and names of Semtech or their respective companies. Semtech reserves the right to make changes to, or discontinue any products described in this document without further notice. Semtech makes no warranty, representation or guarantee, express or implied, regarding the suitability of its products for any particular purpose. All rights reserved.

Wireless Communications

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. The Semtech product 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. Semtech accepts no responsibility for damages of any kind resulting from delays or errors in data transmitted or received using the Semtech product, or for failure of the Semtech product to transmit or receive such data.

Safety

Do not operate the Semtech product in areas where blasting is in progress, where explosive atmospheres may be present, near medical equipment, near life support equipment, or near any equipment which may be susceptible to any form of radio interference. In such areas, the Semtech product should be powered off.

Qualcomm Licenses

Semtech's cellular modules are sold subject to certain notices and restrictions regarding patent licenses from Qualcomm Incorporated. These notices and restrictions are available at www.sierrawireless.com/qualcomm-notices.

Sierra Wireless

Semtech Corporation acquired Sierra Wireless in January 2023. The Sierra Wireless brand is gradually being phased out. During the phase-out period, references to both “Semtech” and “Sierra Wireless” may appear in product documentation.

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

Revision History

Revision number	Release date	Changes
1.0	June 01, 2015	Creation
2.0	September 30, 2016	Updated: <ul style="list-style-type: none"> ▪ 2.2.1 Package Description ▪ 2.3.2 Packing Label Added chapter 5 Rework Guidelines
3.0	June 2, 2017	Added WP76 series Updated: <ul style="list-style-type: none"> ▪ Chapter 1.2 Component Package ▪ Number of reflow cycles (2.1.3 and 4.2) Added: <ul style="list-style-type: none"> ▪ WP Marking example ▪ 4.3 Solder Paste Deposit
4.0	January 23, 2018	Added WP77 Series Updated: <ul style="list-style-type: none"> ▪ Figure 2 WP Marking example ▪ 3.2.2 Footprint ▪ 3.2.3 Layout Recommendations ▪ 4.2 Solder reflow Profile ▪ 4.3 Washing and Potting ▪ 5. Rework Guidelines below
5.0	January 25, 2018	Updated: <ul style="list-style-type: none"> ▪ Figure 7 Recommended Footprint ▪ Figure 8 Recommended Solder Resist and Paste Mask Layout
6.0	October 31, 2019	Updated Figure 3 AirPrime WP76xx Product Marking Example - Laser-etched

Customer Process Guidelines

Revision number	Release date	Changes
7.0	November 03, 2019	Corrected P1 specification for Figure 5 WPx5/WP76/WP77 Series Tape and Reel
8.0	December 16, 2021	Corrected unreel direction
9.0	December 8, 2022	Updated 2.1.1. Storage conditions period
10	February 2026	Updated to Semtech template Updated 4.2. Solder Reflow Profile Updated 4.3 Washing and Potting

Contents

- Important Notice 2
- Wireless Communications..... 2
- Safety..... 2
- Qualcomm Licenses 2
- Sierra Wireless 3
- Contact Information 3
- Revision History 3

- 1: About This Document 7**
 - 1.1 Overview 7

- 2: Handling 8**
 - 2.1 Storage and Handling of the WP Series Module 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..... 11
 - 2.3.1 Packing Description 11
 - 2.3.2 Packing Label..... 12

- 3: SMT Assembly Process 13**
 - 3.1 Lead-Free Process 13
 - 3.2 PCB Design Requirements..... 13
 - 3.2.1 PCB Surface Finish..... 13
 - 3.2.2 Footprint 14
 - 3.2.3 Layout Recommendations..... 15
 - 3.3 Solder Mask 15

4: Board Mounting Guidelines	16
4.1 Stencil Design.....	16
4.2 Solder Reflow Profile.....	17
4.3 Washing and Potting.....	19
5: Rework Guidelines	20
5.1 Component Removal.....	20
5.2 Pad Redress.....	21
5.3 Solder Paste Deposit.....	21
5.4 New Component Placement.....	21
5.5 New Component Soldering.....	21

1: About This Document

1.1 Overview

This document presents guidelines for the industrial assembly of the WPx5/WP76xx/WP77xx Series Embedded Module on an application.

2: Handling

2.1 Storage and Handling of the WP Series Module

2.1.1 Storage Condition

WP Series embedded modules can be stored in their sealed, original packages, for up to 18 months.

They can withstand a storage temperature range between -40°C to $+85^{\circ}\text{C}$, nevertheless when packed into T&R the upper storage temperature is decreased to $+40^{\circ}\text{C}$ due to T&R packaging material.

Tip: *For optimal results, the recommended storage temperature is $+20^{\circ}\text{C} \pm 10$ degrees.*

2.1.2 ESD

The WP Series module is ESD-sensitive.

For ESD level information, refer to the corresponding Product Technical Specification of each product as listed in the Reference Documents.

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

- JEDEC standard JESD625-A, 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 WP Series module is sensitive to moisture absorption:

- MSL 3, 245°C , 1 or 2 reflows allowed on customer PCB. See additional details in **Solder Reflow Profile**.

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

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

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

2.2 Component Package

2.2.1 Package Description

The WPx5/WP76/WP77 Series module is a scalable QFN (quad flat no lead) package, 22x23 mm, pitch 0.8 mm, with 239 terminals.

The terminals include:

- 66 inner signal pads
- 91 outer signal pads
- 8 mechanical corner pads
- 9 signal pads for JTAG
- 1 polarity mark
- 64 ground pads

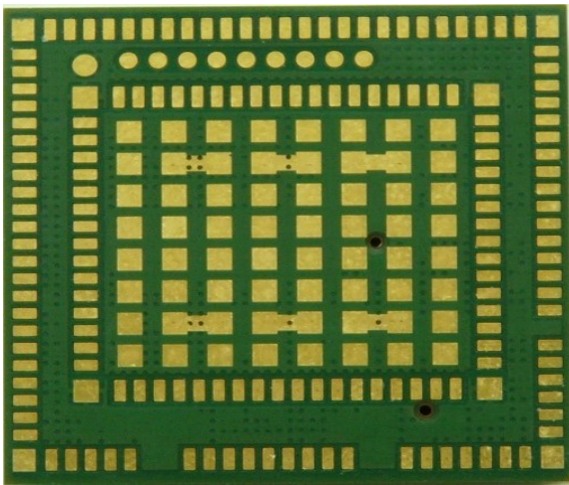


Figure 2-1: WP Series Module - Bottom View

PCB material is FR4. Plating is NiAu ($3\ \mu\text{m} < \text{Nickel} < 8\ \mu\text{m}$ and $0.05\ \mu\text{m} < \text{Gold} < 0.13\ \mu\text{m}$).

Different assembly technologies are used inside the WP Series modules. Some WP modules use Semtech's standard assembly process (2.5 mm module height), while others use a stacked technology (three PCB layers, 4.4 mm module height).

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

2.2.2 Marking Description

Label/marketing contents and methods on the module may differ between each variant of the product family. The marking method can be via paper label or laser-marking.

Common label contents include:

- Model Name, Serial Number and IMEI or MEID Number (both letters and data matrix bar code)
- Fabrication Country
- Relevant regulatory compliance markings and identification codes (this may be a CE logo, FCC ID number, IC ID number, etc.)

- Pin 1 indicator for solder-down modules.

The examples below are not contractual and do not show the exact contents of the product. Label contents may also be rearranged to fit any additional specific needs to a customer or market segment, and can change without notice at the sole discretion of Semtech.

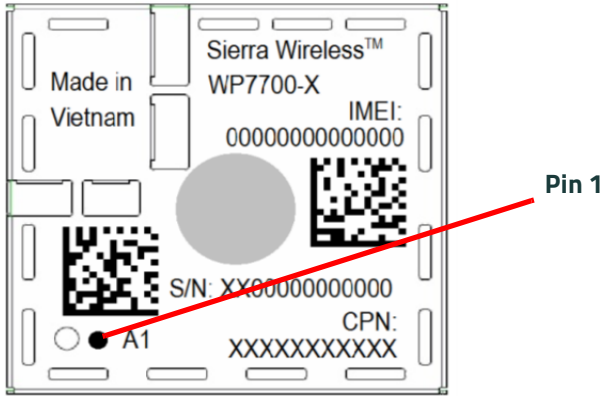
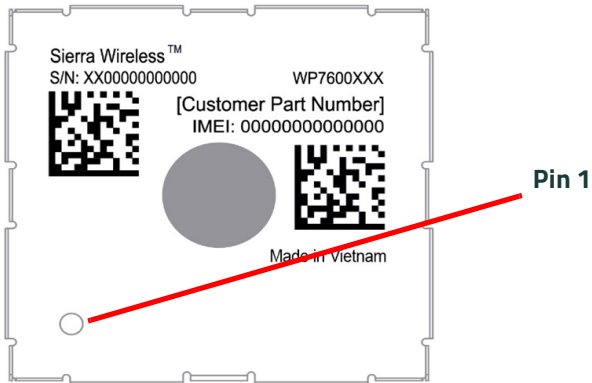


Figure 2-2: WP77xx Product Marking Example – Laser-etched



Note: The WP76xx marking example above is for all WP76xx modules except WP7601 / WP7603 / WP7611.

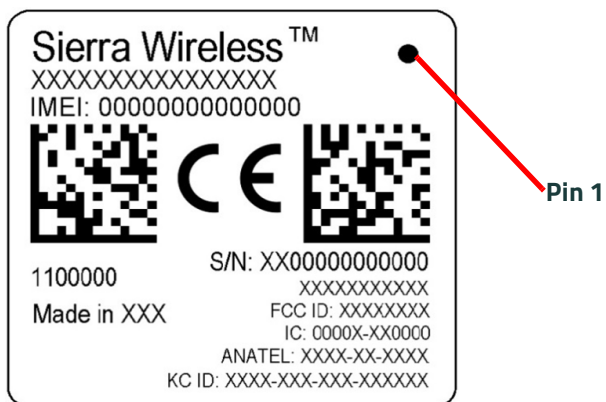


Figure 2-3: WPx5xx Product Marking Example – Label

2.3 Component Packing

2.3.1 Packing Description

The WP Series module is delivered in tape and reel.

Each tape and reel contains:

- 4.4 mm modules: 300 per reel
- 2.5 mm modules: 500 per reel

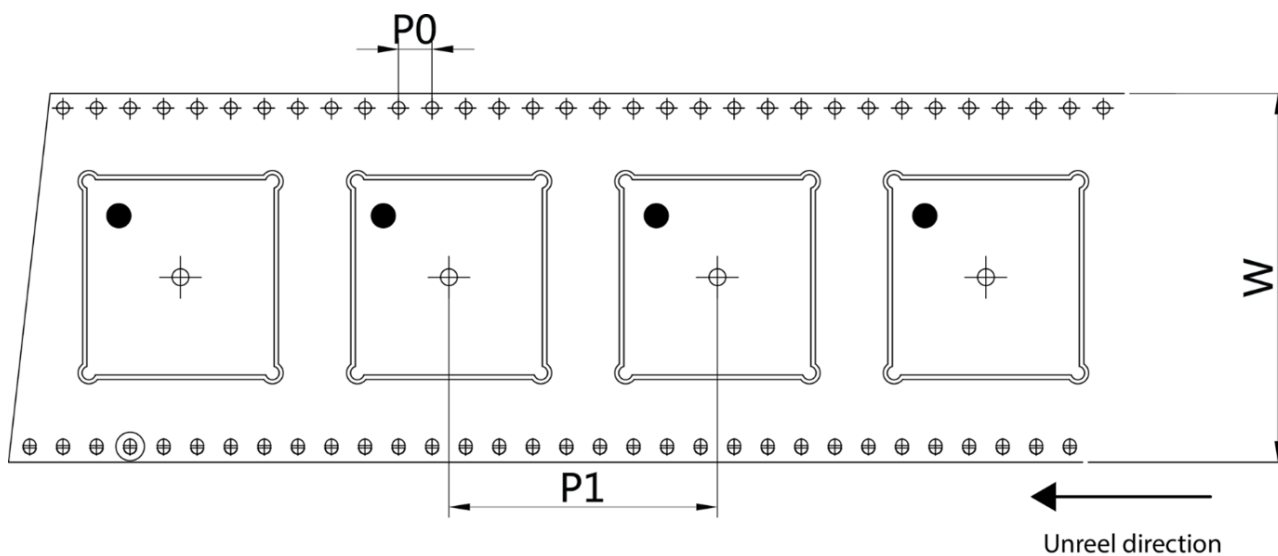


Figure 2-4: WPx5/WP76/WP77 Series Tape and Reel

P1	P0	W
28.0 mm	4.0 mm	44.0 mm

2.3.2 Packing Label

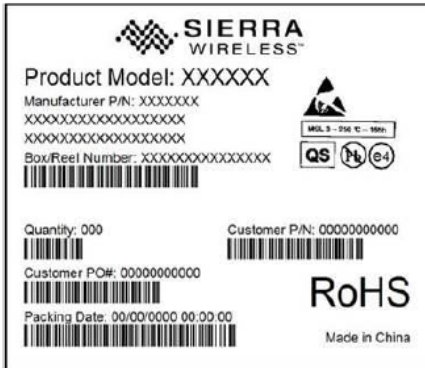


Figure 2-5: Packing Label

Note that Customer PO # may not be on the packing label, but may be attached to the box on a separate label.

3: SMT Assembly Process

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

Note: The WP Series products should be assembled by reflow process.

3.1 Lead-Free Process

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

Table 3-1: List of 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%

WP Series modules are manufactured with RoHS-compliant components and processes.

3.2 PCB Design Requirements

3.2.1 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. The process does not give consistent solder volumes on each pad because of poor pad flatness.*

3.2.2 Footprint

In order to produce high assembly yields and a reliable solder joint, the footprint on the customer application board should match **Figure 7 on page 14**.

Footprint notes:

- The 64 central pads and the 8 corner pads are ground pads.
- Manufacturing tolerance for copper pads: 30 µm

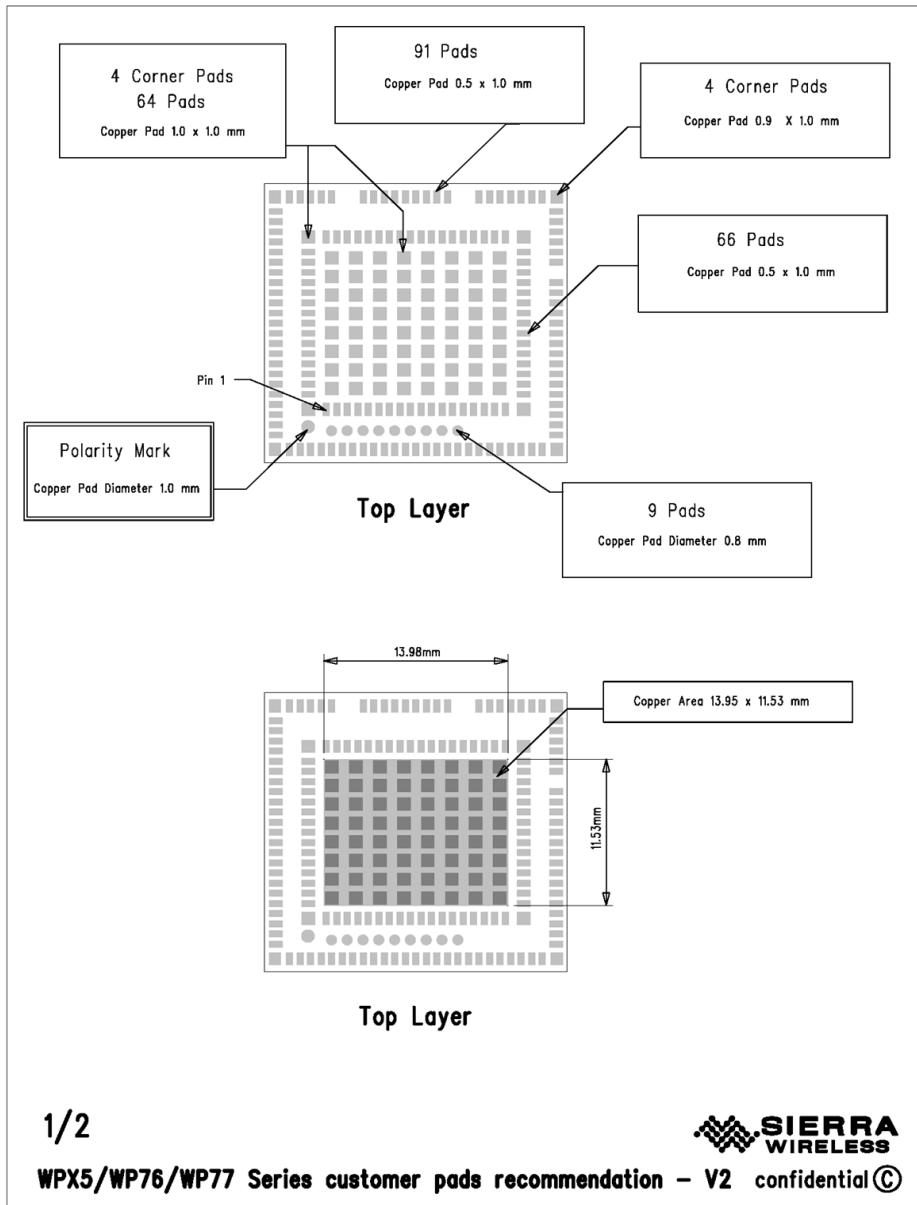


Figure 3-1: Recommended Footprint

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

3.2.3 Layout Recommendations

Semtech layout recommendations include:

- A GROUND area under the WP Series 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 between each of the 64 ground pads (under the solder resist).
- There should not be any SIGNAL trace or hole / micro-via under the module.
- Customers should place a copper pad under the Polarity mark to avoid any risk of short circuits between it and the customer layout. This mark eases visual confirmation of the module's correct orientation.
- The Polarity mark should not be soldered (there is no stencil aperture for the Polarity mark).
- 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.
- Leave a component-free area of 2 mm around the WP Series module for accessing the surrounding components.

3.3 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 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.

For the external pads, the solder mask opening should be 100 µm to 150 µm larger than the pad, resulting in 50 µm to 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 the 64 ground pads, SMD pads should be used if a copper ground area is under the WPx5/WP76/WP77 Series module, as described in [Layout Recommendations](#).

Recommended solder mask thickness on top copper is 10 µm to 30 µm.

Refer to **Figure 8. Recommended Solder Resist and Paste Mask Layout** for a recommended solder resist and paste mask layout.

4: Board Mounting Guidelines

4.1 Stencil Design

The recommended stencil thickness is 125 μm .

The proposed stencil design is presented in **Figure 8.Recommended Solder Resist and Paste Mask Layout**

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

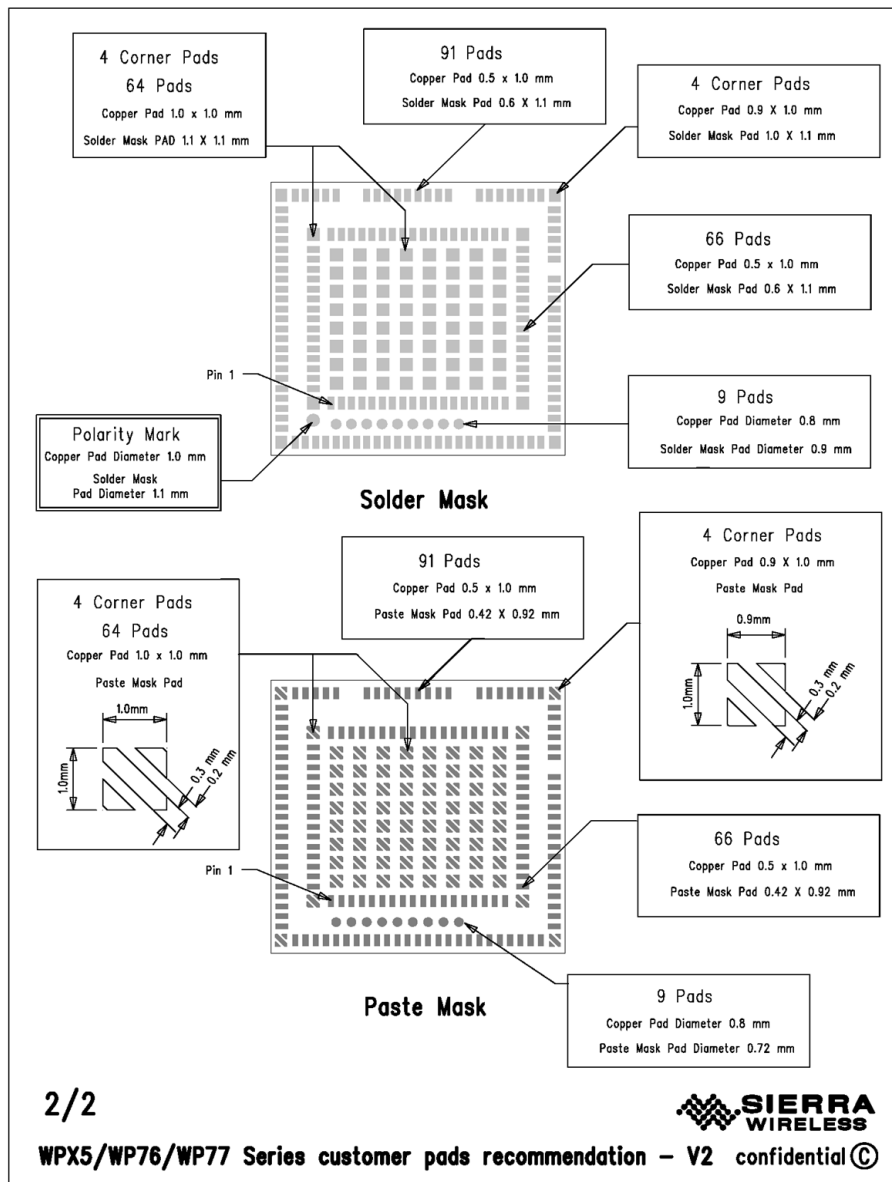


Figure 4-1: Recommended Solder Resist and Paste Mask Layout

4.2 Solder Reflow Profile

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

Table 4-1 is a mandatory requirement to ensure reliable assembly.

Table 4-1: Solder Reflow Profile

Peak Temperature	245°C max
Number of reflow cycles – 2.5 mm module	2
Number of reflow cycles – 4.4 mm module	1

Important: *The initial customer manufacturing process should limit 4.4 mm WP modules to one reflow.*

Caution: *Even for 2.5 mm WP modules, 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 WP module, care must be taken to ensure the module is not reflowed.

When customer PCBA assembly requires dual-sided reflow cycles, Semtech strongly recommends mounting the module during the second reflow cycle to ensure optimal solder joint reliability and component integrity.

Figure 4-2 is an example of reflow profile.

Example of reflow profile:

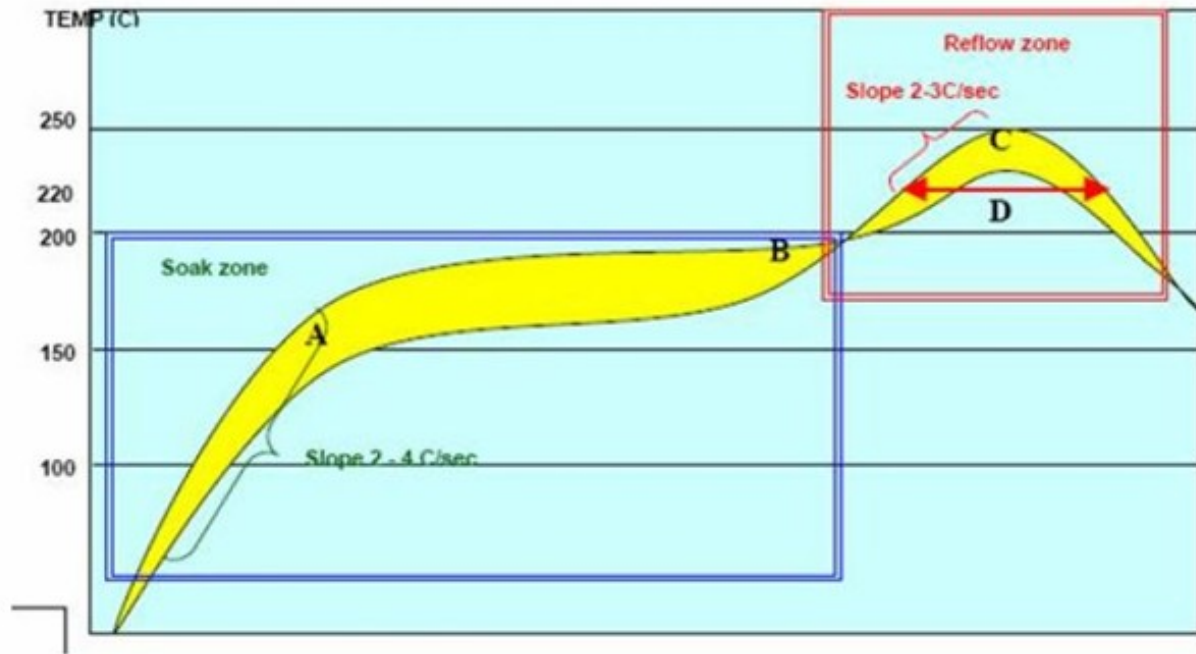


Figure 4-2: Recommended Reflow Profile

Additional recommendations are presented in the table below for consideration.

Factor	Recommendation
Max slope at soak zone	2 to 4 °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 under nitrogen atmosphere.

4.3 Washing and Potting

Semtech has not performed potting/coating qualification tests on the WP modules. Customers should pay attention to RF tracks since the ϵ_{air} (epsilon-air) will be different below the potting/coating compared to an open-air design. After potting/coating a module, RF performance should be re-checked to guarantee that no degradation compared to nominal values occurred. Semtech also recommends that potting/coating material not be allowed to ingress under the shield.

As the potting/coating process varies depending on a customer's need, it is the customer's responsibility to ensure the selected material and process parameters are compatible with the WP module. If a customer's potting/coating process is found to have contributed to a failure, the unit will not be covered under warranty. Units returned for failure analysis must have all potting/coating material removed and the FSN marking must be visible.

The shield cannot be removed and no heat treatment should be used during the potting/coating material removal.

Water wash is not recommended with shielded Semtech embedded modules due to difficulty in ensuring proper drying under the shield.

Use of ultrasonic process should be avoided as it can damage the quartz crystal components.

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 component floor life has been exceeded, it is highly recommended to pre-bake the PCB to remove moisture from the assembly. (See JEDEC J-STD-033 paragraph 6 - Board rework. If possible for the PCB and the other components of the board, apply the same baking as per [Moisture Sensitivity](#).) The pre-baking process will prevent damage to any component due to moisture vapor pressures caused during reflow.

Prior to removing the component, the shielding and the module's PCB layers (1 or 3 depending on module series) should be glued together using glue able to withstand reflow profile. Glue should not touch the customer board; usage of adhesive tape (PI) on the PCB helps to avoid glue drop on the PCB.



Figure 5-1: Glue deposit on WP module.

5.1 Component 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 as long as the reflow is complete.

In the removal process, it is recommended that the board should be heated from the bottom side using convective heaters and hot gas, or hot air or IR should be used on the top side of the component. Special nozzles or IR lens should be used to direct the heating in the component area and heating of adjacent components should be minimized.

Excessive hot airflow should also be avoided, as this causes the component to overheat.

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

Once the component has been removed, the site and pads need to be cleaned properly. It is better to use the combination of a blade style conductive tool and a fluxed desoldering braid.

Once the residual solder has been removed, the land pads should be cleaned with a solvent. The solvent is usually specific to the type of solder paste used in the original assembly and 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 component itself or on the customer PCB) with a mini-stencil that has the same thickness and apertures as the stencil used for original attachment.

5.4 New Component Placement

A slip-beam optical system should be used to align the component 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 Component Soldering

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