Module Secondary SMT User Guide

LCC/LGA Module Series

Rev. Module_Secondary_SMT_User_Guide_V2.4

Date: 2018-06-02

Status: Released

Our aim is to provide customers with timely and comprehensive service. For any assistance, please contact our company headquarters:

Quectel Wireless Solutions Co., Ltd.
7th Floor, Hongye Building, No.1801 Hongmei Road, Xuhui District, Shanghai 200233, China
Tel: +86 21 5108 6236
Email: info@quectel.com

Or our local office. For more information, please visit:
http://quectel.com/support/sales.htm

For technical support, or to report documentation errors, please visit:
http://quectel.com/support/technical.htm
Or email to: support@quectel.com

GENERAL NOTES

QUECTEL OFFERS THE INFORMATION AS A SERVICE TO ITS CUSTOMERS. THE INFORMATION PROVIDED IS BASED UPON CUSTOMERS' REQUIREMENTS. QUECTEL MAKES EVERY EFFORT TO ENSURE THE QUALITY OF THE INFORMATION IT MAKES AVAILABLE. QUECTEL DOES NOT MAKE ANY WARRANTY AS TO THE INFORMATION CONTAINED HEREIN, AND DOES NOT ACCEPT ANY LIABILITY FOR ANY INJURY, LOSS OR DAMAGE OF ANY KIND INCURRED BY USE OF OR RELIANCE UPON THE INFORMATION. ALL INFORMATION SUPPLIED HEREIN IS SUBJECT TO CHANGE WITHOUT PRIOR NOTICE.

COPYRIGHT

About the Document

History

Revision	Date	Author	Description
1.0	2012-08-28	Gavin HOU	Initial
2.0	2013-08-26	Gavin HOU	Added the description of stencil-making in Chapter 4.2
2.1	2013-12-19	Gavin HOU	Modified Figure 3: Inward Shrinking and Outward Moving
2.2	2015-11-23	Meisy MEI	Added the description of stencil-making on UC/EC/GC series in Chapter 4.2
2.3	2017-03-08	Alain HUANG	1. Added the description of stencil design requirements for M66/M66-DS/MC60/L70-R/ L70-RL/L76-L/L76B/L80-R/L86/L96/EC20 R2.0/ EC21/EC25/EG91/EG95/BG96/FC10/FC20/SC10/ SC20/SG30/AG35 modules in Chapter 4.2. 2. Added desoldering and repair instructions in Chapter 5 and 6.
2.4	2018-06-02	Rowan WANG/ Alain HUANG	1. Updated the MSL rating of Quectel modules into 3. 2. Updated stencil design requirements in Chapter 4.2. 3. Optimized the recommended reflow soldering requirements and thermal profile in Chapter 4.4.

Content

Table Index

Figure Index

1 Introduction

This document describes the process of Quectel modules' secondary SMT and desoldering. It is applicable to all Quectel modules in LCC or LGA form factor.

2 Information about Modules

2.1. Surface-Mount Packaging Type

Quectel modules adopt LCC or LGA package.

2.2. Packing Methods

Quectel provides the following packing types:

- Tray Packing
- Tape and Reel Packing

Figure 1: Tray Packing and Tape & Reel Packing

3 Requirements on Chip Mounter

3.1. Chip Mounter

- Feeder: Support auto tray feeder and auto reel feeder
- Image processing: Optical plummet centering
- Diameter of nozzle: Select the suitable nozzle according to the module size

NOTE

The recommended diameter of nozzle should be not less than 40% of the module's shorter side. For example, if the module size is 25mm×20mm, the nozzle diameter should be 8mm at least.

3.2. Soldering Requirements

1. It is recommended to use reflow soldering equipment with eight zones at least. For Quectel LTE, LPWA, Automotive and Smart series modules, reflow soldering equipment with at least ten zones is recommended.

2. In a lead-free reflow oven, the peak temperature of the actual solder joints on the component side of an LGA module should be greater than 240°C, and the temperature of fixtures is recommended to be 240-245°C to avoid cold solder joints on LGA modules.

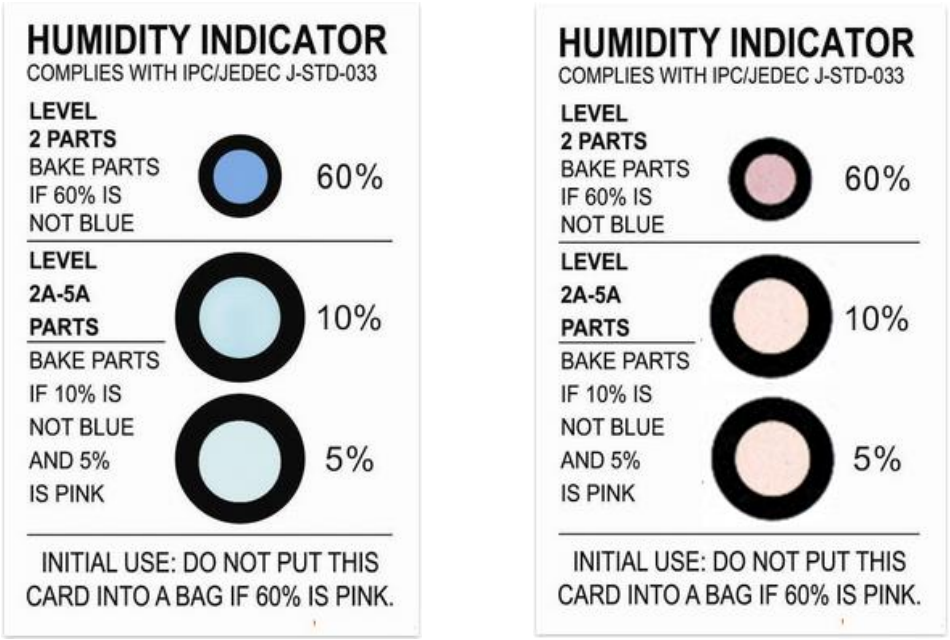
3. When conducting reflow soldering at the bottom side of the module, the module is upside down, and components may be dropped because of gravity, so there is a limit on the module's weight. Please refer to the following formula: Allowable Module Weight (g) = Surface Area of Pin (mm^2) × Number of Pins × 0.665. If the module exceeds the allowable weight, please reduce the temperature of the bottom side by 5-8°C or use a fixture to hold the module.

4 Attentions for Manufacturing

4.1. MSL and Moisture-proof Requirement

Quectel SMD module is sensitive to moisture absorption. According to IPC-JEDEC standard, the moisture sensitive level (MSL) of Quectel SMD modules is defined as “3”. Please make sure the package is intact before using. After opening the package, please confirm the status of humidity indicator card in the vacuum-sealed package. To prevent the module from permanent damage, baking before reflow soldering is required if any circumstance below occurs:

- Humidity indicator card: At least one circular indicator is no longer blue.
- The seal is open and the module is exposed to air for more than 168 hours.



Normal **Not blue**

Figure 2: Humidity Indicator Card

NOTES

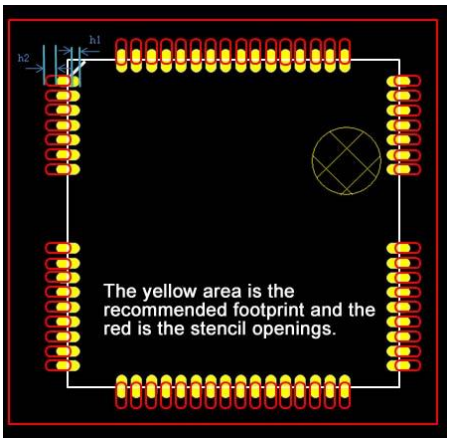
1. If baking is required, the module should be baked for 8 hours at 120ºC±5ºC.
2. Please take out the module from the package and put it on high-temperature resistant fixtures before baking. All modules must be mounted within 24 hours after finishing baking, otherwise put them in the drying oven.

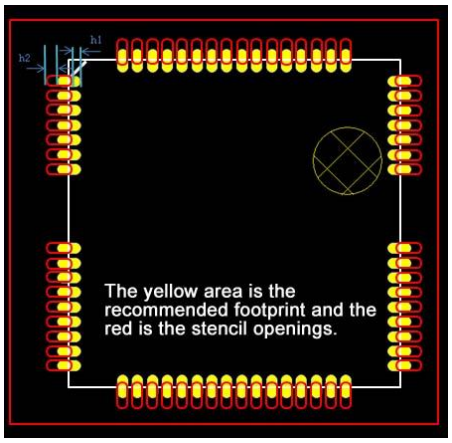
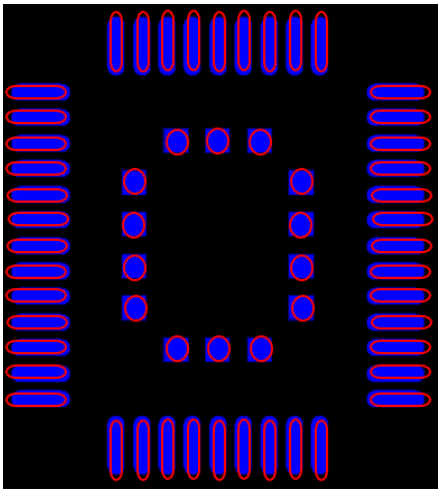
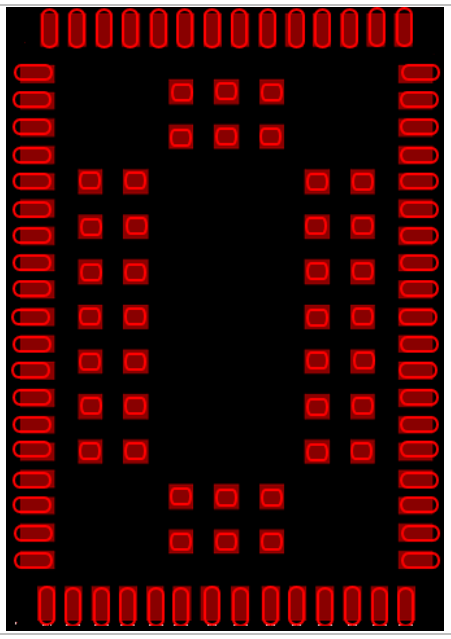
4.2. Stencil Design Requirements

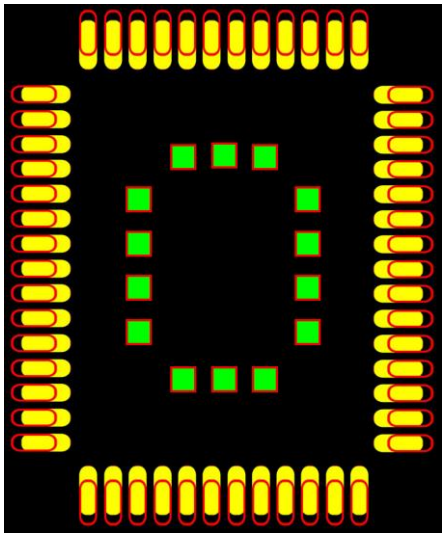
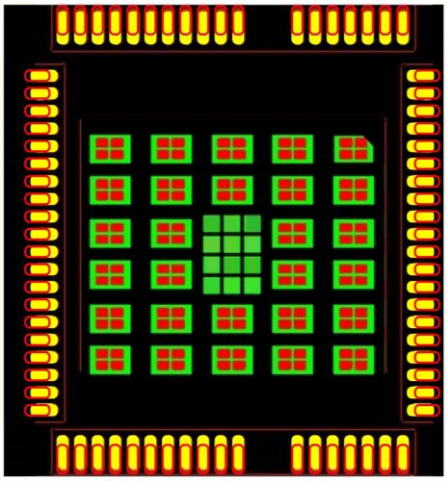
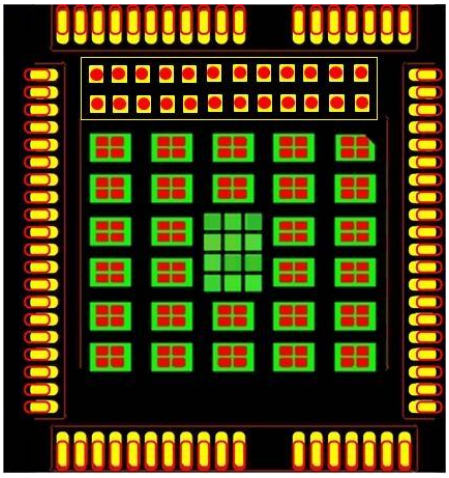
To ensure the solder paste is enough and soldering joints are reliable, the stencil should be partially stepped-up on the top surface. And the stencil opening for each single pin cannot be larger than 3.0mm×4.0mm. If the size is exceeded, divide it into smaller openings with size less than 2.0mm×2.0mm by 0.3-0.5mm shelves.

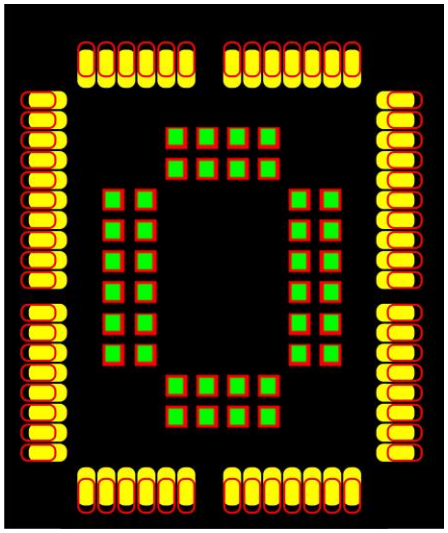
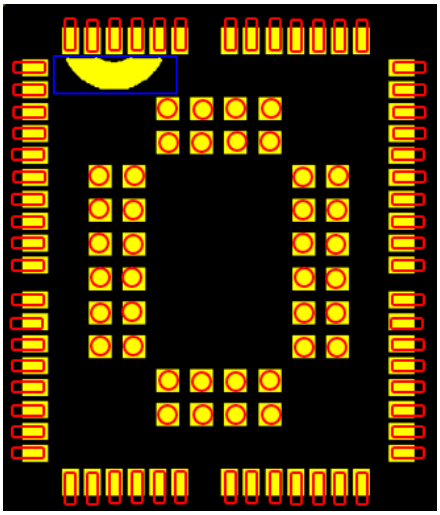
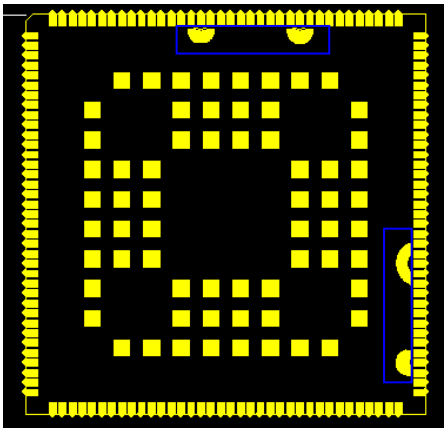
The stencil design requirements for Quectel modules are shown in the table below.

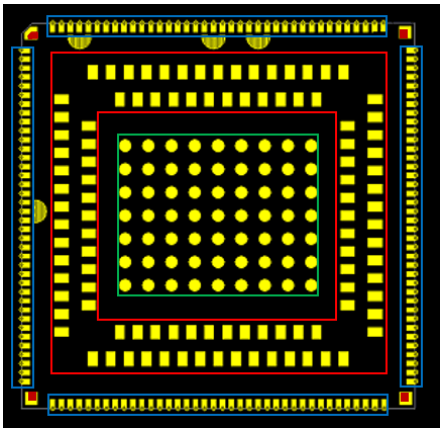
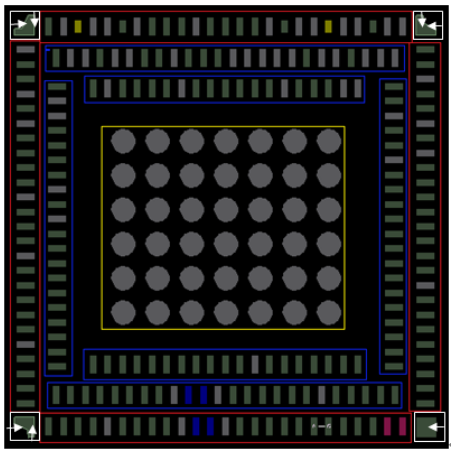
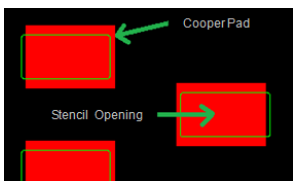
Table 1: Stencil Design Requirements

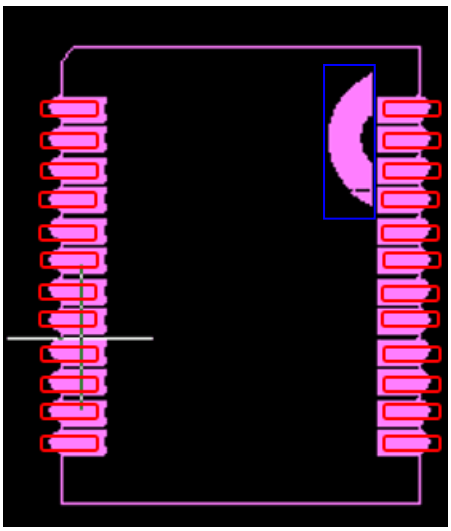
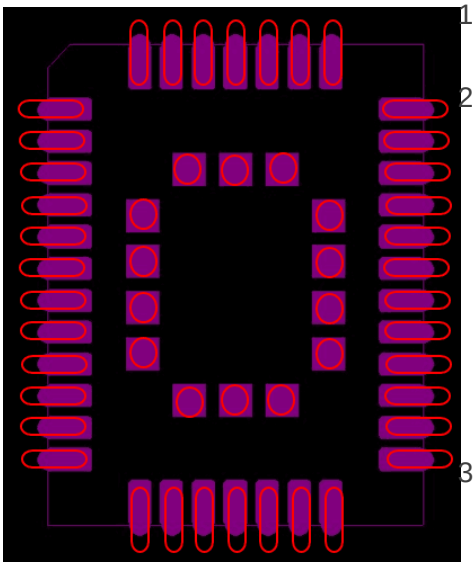
Module	Diagram for Stencil Design Requirements	Description
M10/M12/M72/M80/M85/ M95/GC10/M66/M66-DS		1. The thickness of stencil should be stepped-up to 0.18-0.20mm. 2. The stencil opening for each single pin should be shrunken inward by 0.10mm (refer to h1) and moved outward by 0.20-0.30mm (refer to h2).

L10/L16/L20/L26/L30/L50/L70/L76/L80/L70-R/L70-RL/L76-L/L76B/L80-R/L86/L96		1. The thickness of stencil should be stepped-up to 0.13mm. 2. The stencil opening for each single pin should be shrunken inward by 0.10mm (refer to h1) and moved outward by 0.30-0.50mm (refer to h2).
BC66/BC68		1. The thickness of stencil should be stepped-up to 0.15-0.18mm. 2. For pins on four sides: The stencil opening for each single pin should be moved outward by 0.20-0.30mm in length direction, and shrunken inward by 10% in width direction. 3. For pins in the center: The stencil opening for each single pin should be in an area of 80% of corresponding pin.
M89		1. The thickness of stencil should be stepped-up to 0.18-0.20mm. 2. For pins on four sides: The stencil opening for each single pin should be shrunken inward by 0.30mm and moved outward by 0.40mm. 3. For pins in the center: The stencil opening for each single pin should be in an area of 80% of corresponding pin, and should be designed into square shapes with round chamfers.

MC60		1. The thickness of stencil should be stepped-up to 0.18-0.20mm. 2. For pins on four sides: The stencil opening for each single pin should be shrunken inward by 0.10mm and moved outward by 0.30-0.40mm. 3. For pins in the center: Design the stencil opening for each pin into four 0.55mm×0.55mm smaller openings with 0.05mm square chamfers.
UC20/UC15		1. The thickness of stencil should be stepped-up to 0.18-0.20mm. 2. For pins on four sides: The stencil opening for each single pin should be shrunken inward by 0.30mm and moved outward by 0.40mm. 3. For GND pins in the center: Design the stencil opening for each pin into four 1.00mm×0.65mm smaller openings with 0.05mm square chamfers, and with 0.25m space in between. 4. The 12 pins in the very center are used for R&D test and recommended to be kept intact.
EC20/EC20 R2.0/ EC21/EC25/ EC20 R2.1		1. The thickness of stencil should be stepped-up to 0.18-0.20mm. 2. For pins on four sides: The stencil opening for each single pin should be shrunken inward by 0.20mm and moved outward by 0.40mm. 3. For GND pins in the center: Design the stencil opening for each pin into four 1.00mm×0.65mm smaller openings with 0.05mm square chamfers, and with 0.25m space in between. 4. Design a round opening with a diameter of 0.70mm for the pins in

		the yellow box. 5. The 12 pins in the very center are used for R&D test and recommended to be kept intact.
UG95/UG96/BC95/ BC95-G/EG91/EG95		1. The thickness of stencil should be stepped-up to 0.18-0.20mm. 2. For pins on four sides: The stencil opening for each single pin should be shrunken inward by 0.30mm and moved outward by 0.40mm. 3. For pins in the center: Design the stencil opening for each pin into four 0.75mm×0.75mm smaller openings with 0.05mm square chamfers.
BG96		1. The thickness of stencil should be stepped-up to 0.18-0.20mm. 2. For pins on four sides: The stencil opening for each single pin should be shrunken inward by 0.10mm and moved outward by 0.30mm, and that for adjacent pins should keep a spacing of 0.4mm. 3. For pins in the center: Design a round opening with a diameter of 1.0mm for these pins. 4. There is no need to design stencil opening for the arc shaped pin in blue box.
SC20		1. The thickness of stencil should be stepped-up to 0.18-0.20mm. 2. For pins on four sides: The stencil opening for each single pin should be shrunken inward by 0.10mm and moved outward by 0.50-0.60mm. 3. For square pins in the center, the stencil opening size should be 70% to 80% of corresponding pin size. 4. There is no need to design stencil opening for the arc shaped pin in

		blue box.
SC60		1. The thickness of stencil should be stepped-up to 0.18-0.20mm. 2. For pins on four sides (in blue boxes), the stencil opening for each pin should be in a size of 1.8mm×0.6mm, and the opening for adjacent pins should keep a spacing of 0.4mm. 3. For square pins between the two red boxes, the stencil opening size should be the same as the pin size, but should be designed with round chamfers. 4. For round pins in the center (in green box), the stencil opening size should be 80% of corresponding pin size. 5. For pins at the four corners of the module, the stencil opening should be designed into an area of about 60% of corresponding pin, as indicated in red blocks in the figure. 6. There is no need to design stencil opening for the four arc shaped pins.
SG30/AG35/ EG06/EG12*	  	1. The thickness of stencil should be stepped-up to 0.15-0.18mm. 2. The stencil opening size for the square pins should be 1.0mm×0.5mm. 3. For pins at the four corners of the module (marked with white boxes), the stencil opening should be shrunken inward as per the directions marked in arrows, and with an area of about 60% of corresponding pin. 4. The stencil opening for circular GND pins in the yellow box should be designed into the one shown in the bottom right figure (the section in grey is the stencil opening with an area of about 80% of the pin).

FC10		1. The thickness of stencil should be stepped-up to 0.15-0.18mm. 2. The stencil opening for each single pin should be shrunken inward by 0.10mm and moved outward by 0.40mm in length direction, and shrunken inward by 0.05mm in width direction, and additionally should be designed with 0.05mm square chamfers. 3. There is no need to design stencil opening for the arc shaped pin in blue box.
FC20		1. The thickness of stencil should be stepped-up to 0.15-0.18mm. 2. For pins on four sides: The stencil opening for each single pin should be shrunken inward by 0.10mm and moved outward by 0.40mm in length direction, and shrunken inward by 0.05mm in width direction, and additionally should be designed with 0.05mm square chamfers. 3. For pins in the center: The stencil opening for each single pin should be in an area of 80% of corresponding pin, and should be designed into circular shapes.

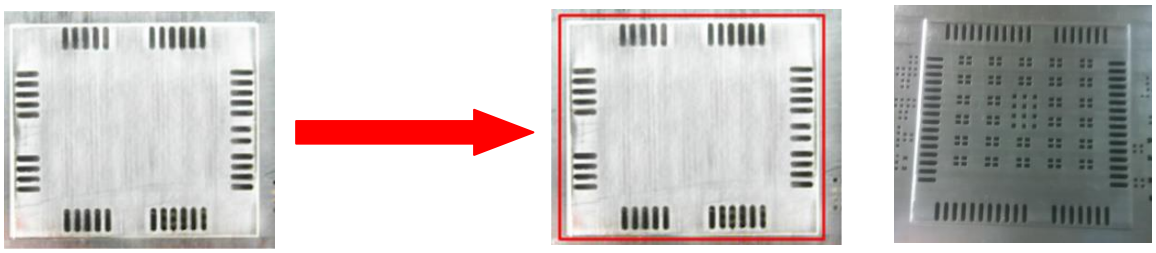


Figure 3: Step-up Stencil Area

NOTES

1. "*" means under development.
2. The openings of stencil's components, which have a distance about 5mm away from the edge of the

module, should be shrunken by 10%~30% of the actual opening size. For components with 0.5mm pitch (or smaller) or 0201 components, please keep at least 3mm space in between, otherwise the module will be at the risk of short circuit.

3. You can optimize stencil-making depending on the actual situation.
4. Inward shrinking and outward moving are relative to the host PCB footprint of the module. For details of the recommended footprint, please refer to the hardware designs of the corresponding modules.

4.3. Mounting Process

4.3.1. Load Materials

For tray packed modules, in order to ensure mounting accuracy, it is recommended to use dedicated tray/fixture for module loading.

For tape and reel packed modules, there is a need to set the feeding spacing according to actual conditions.

4.3.2. Automatic Placement

Select a suitable nozzle according to the module size. To keep module's stability, please ensure that the nozzle is placed in the center of gravity, image detection and recognition are 100% passed, and keep a medium speed when mounting the module. After the module is placed onto the motherboard, the module pads should be in alignment with the corresponding solder paste on the motherboard's pads. The triangle mark on the module indicates its first pin, which should correspond to the mark on PCB.

Figure 4: Automatic Placement

Figure 5: First Pin and Mounted Picture

4.4. Reflow Soldering

Please refer to the recommended ramp-soak-spike thermal profile for lead-free reflow soldering in the following figure.

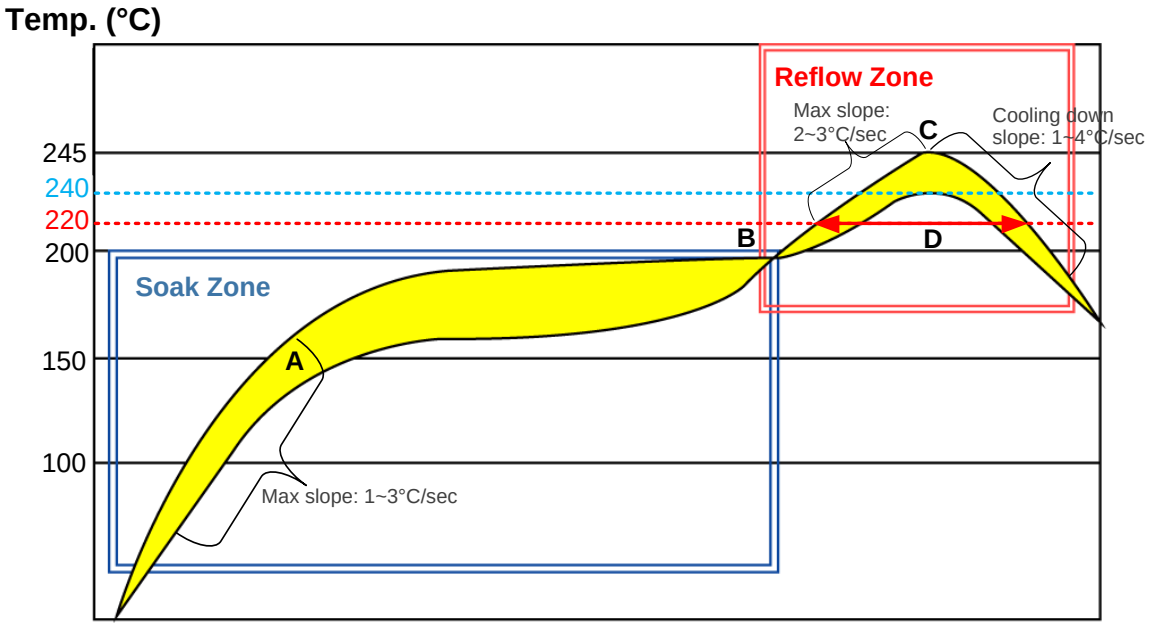


Figure 6: Ramp-soak-spike Reflow Profile

Table 2: Recommended Thermal Profile Parameters

Factor	Recommendation
Soak Zone	
Max slope	1 to 3°C/sec
Soak time (between A and B: 150°C and 200°C)	60 to 120 sec
Reflow Zone	
Max slope	2 to 3°C/sec
Reflow time (D: over 220°C)	40 to 60 sec
Max temperature	240°C ~ 245°C
Cooling down slope	1 to 4°C/sec
Reflow Cycle	
Max reflow cycle	1

NOTES

1. For modules with paper labels:
   - During manufacturing and soldering, or any other processes that may contact the module directly, NEVER wipe the module label with organic solvents, such as acetone, ethyl alcohol, isopropyl alcohol, trichloroethylene, etc. Otherwise, the label information may become unclear.
2. For modules with Cupro-Nickel shields and laser engraved labels:
   - During manufacturing and soldering, or any other processes that may contact the module directly, NEVER wipe the module's shielding can with organic solvents, such as acetone, ethyl alcohol, isopropyl alcohol, trichloroethylene, etc. Otherwise, the shielding can may become rusted.
   - The shielding can for the module is made of Cupro-Nickel base material. It is tested that after 12 hours' Neutral Salt Spray test, the laser engraved label information on the shielding can is still clearly identifiable and the QR code is still readable, although white rust may be found.

5 Desoldering

Please use a heat gun to heat the solder joints so as to remove the module from the motherboard.

- The temperature of the heat gun should be about 350ºC in order to release enough heat. The wind speed should be adjusted according to actual situation.
- If the motherboard has been exposed to the air for exceeding 72 hours, then it should be baked before desoldering.
- During heating, the motherboard should be laid flat and fixed to avoid movement, and the distance between the motherboard and the nozzle should be from 1.0cm to 3.5cm.
- Move the nozzle along the edge of the module at a uniform speed. When all of the solder joints are melted off, take off the module along the diagonal direction with tweezers. The time of the whole process should be no more than 120 seconds.

For the module larger than 33.0mm×33.0mm, a BGA workbench or heat gun can be used to desolder components. To prevent separation between pad and circuit caused by long-term heating on a single side, pre-heating is needed at the bottom side of the module when heat gun is used for desoldering. If PCB is blistered, it is recommended to inspect soldering quality of modules by X-rays.

Table 3: Heat Gun Desoldering Requirements

Parameters	Requirements
The maximum temperature on the surface of PCB	260°C
Desoldering or soldering time limit	40s-120s
Temperature measurement and calibration	● Use temperature measurement devices in calibration period to measure the temperature (the heat gun temperature must be set according to the actually soldering requirements). ● The temperature must not exceed 350°C. ● Temperature check point must be 5mm away from the nozzle of the heat gun, and the nozzle must be placed vertically down when measuring. ● Heat guns that cannot meet the temperature requirements are prohibited to be used. ● Heat gun should be detected with grounding.
Nozzle shape and dimensions	Select an appropriate nozzle according to the type of electronic components.

Fixture	Use dedicated fixtures to hold and fix the motherboard so as to keep it stay still during compoennts removal.

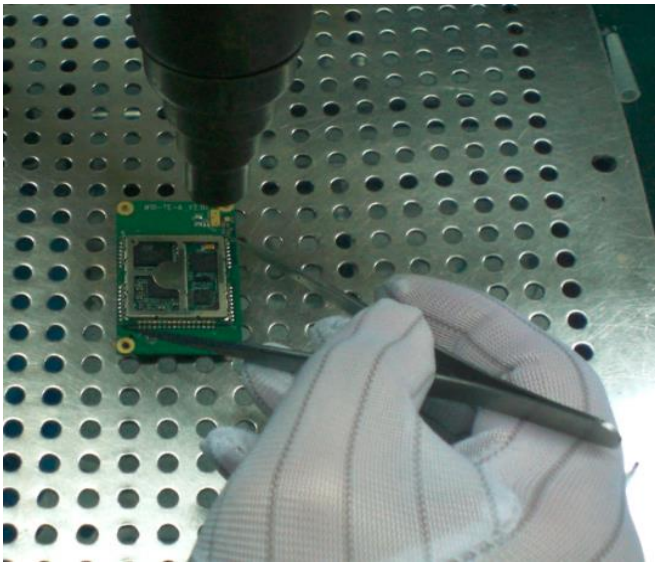


Figure 7: Remove Module

After desoldering, wait a moment until the module and the motherboard cool down. When the module has been removed, please guarantee that the solder paste on the motherboard must be smooth and there is no short circuit between two pins.

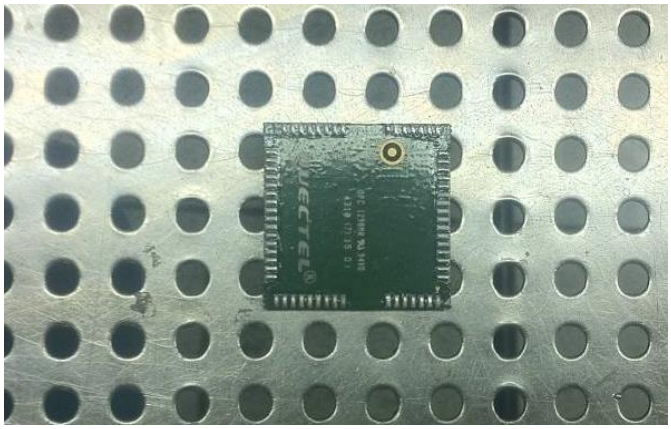


Figure 8: Module Soldering Quality Inspection

NOTE

For the rework requirements of Quectel AG35 module, please refer to *Quectel_AG35_Secondary_SMT_Guidelines*.

6 Repair Instructions

If the temperature of part of the module exceeds the PCB glass transition temperature (140-150°C), then it will be regarded as one repair. The PCBA can be repaired 6 times maximally. Re-soldering or spot soldering by soldering iron is not regarded as one repair, and soldering by heat gun will be defined as one repair. Normally, PCBA will be heated twice for every repair (desoldering and soldering), and thus the maximum repair time for each PCBA is 3. If the module is not restored after three times' repairing, it is recommended to be scrapped.

NOTE

For the rework requirements of Quectel AG35 module, please refer to *Quectel_AG35_Secondary_SMT_Guidelines*.

7 Appendix Reference

Table 4: Terms and Abbreviations

Abbreviation	Description
LCC	Leadless Chip Carriers
LGA	Land Grid Array
MSL	Moisture Sensitivity Level
PCB	Printed Circuit Board
SMD	Surface Mount Device
SMT	Surface Mount Technology