Module Secondary SMT User Guide

LCC/LGA Module Series

Rev. Module_Secondary_SMT_User_Guide_V2.3

Date: 2017-03-08

www.quectel.com

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

Quectel Wireless Solutions Co., Ltd.
Office 501, Building 13, No.99, Tianzhou Road, Shanghai, China, 200233
Tel: +86 21 5108 6236
Email: info@quectel.com

For technical support, or to report documentation errors, please visit:
http://www.quectel.com/support/techsupport.aspx
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.

Content

Table Index

Figure Index

1 Introduction

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

2 Information about Modules

2.1. Package Type

Quectel modules adopt LCC or LGA package.

2.2. Packing Methods

Quectel provides the following packing types:

- Tray Packing
- Reel Packing

Figure 1: Tray Packing and 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, nozzle diameter should be more than 8mm.

3.2. Soldering Requirements

1. It is recommended to use a reflow soldering equipment with eight zones at least. A module can be reflow soldered for maximally 3 times (including the first time reflow soldering during manufacturing).

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 within 243-246°C to avoid cold solder joints on LGA modules.

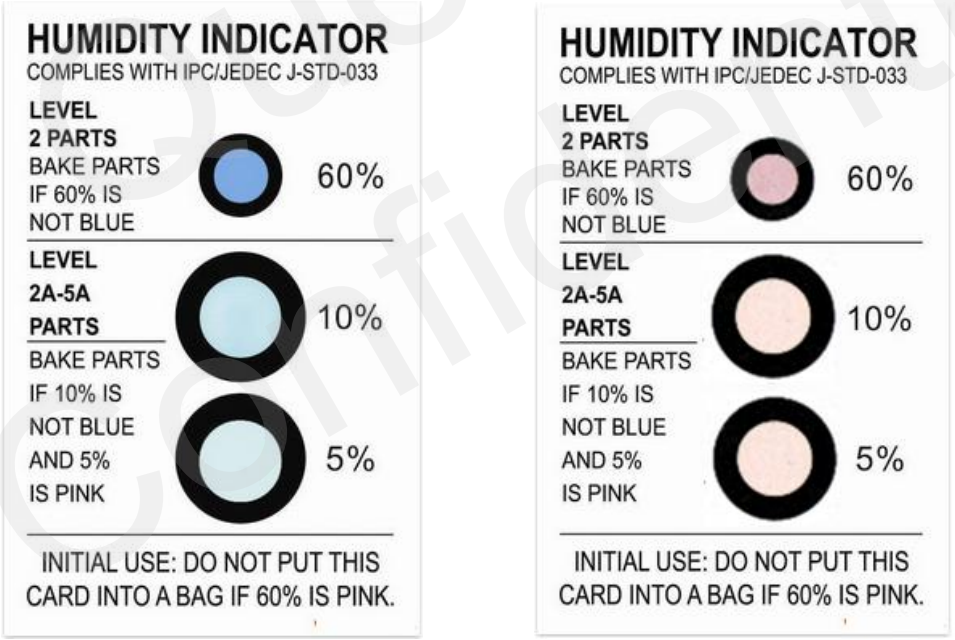
3. When conducting reflow soldering at the bottom of the module, the module is upside down, and components may be dropped because of gravity, so there is a limit on the module's weight. Please refer to the following formula: Allowable Weight (g) = Surface Area of Pin (mm^2) × Number of Pins × 0.665. 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 “4”. 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 72 hours.



Normal **Not blue**

Figure 2: Humidity Indicator Card

NOTES

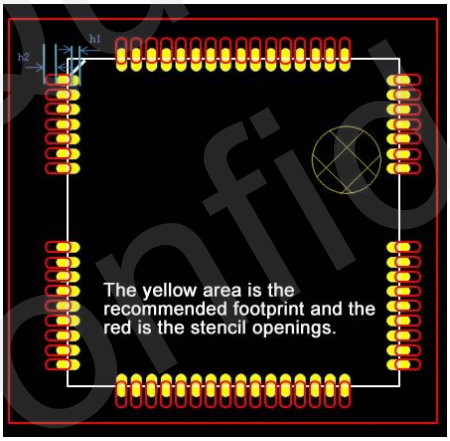
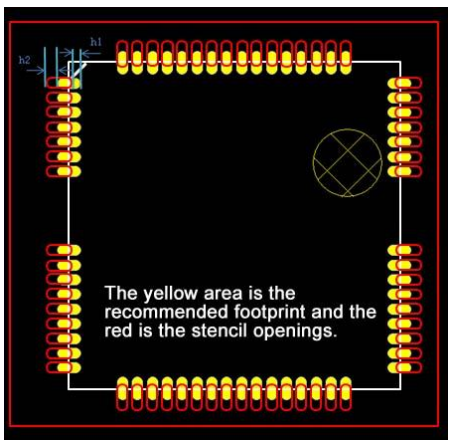
1. If baking is required, the module should be baked for 48 hours at 125ºC±5ºC.
2. Please take out the module from the package and put it on fixtures with high temperature resistance before baking. All modules must be mounted within 24 hours after finishing baking, otherwise put them in the drying oven.

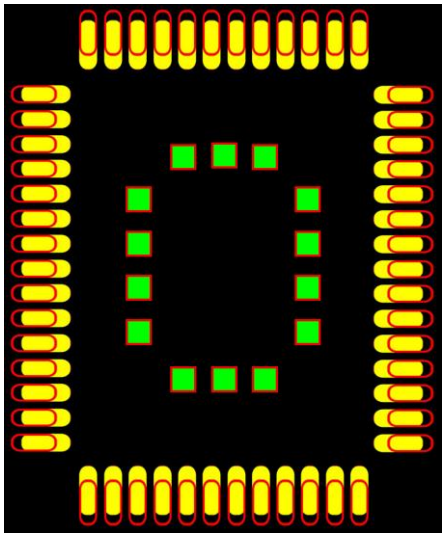
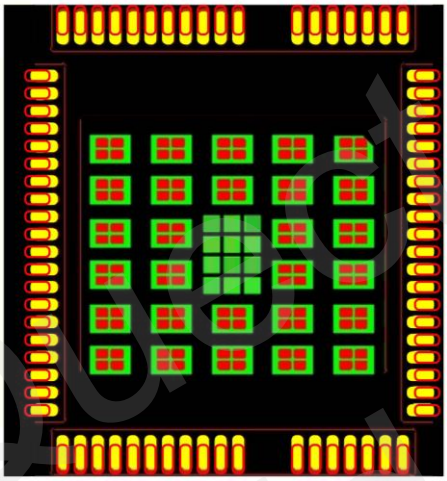
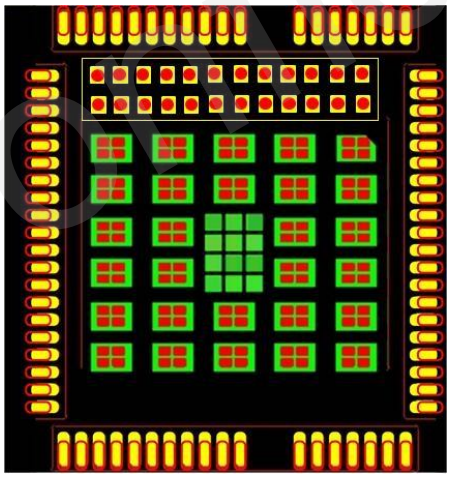
4.2. Stencil Design Requirements

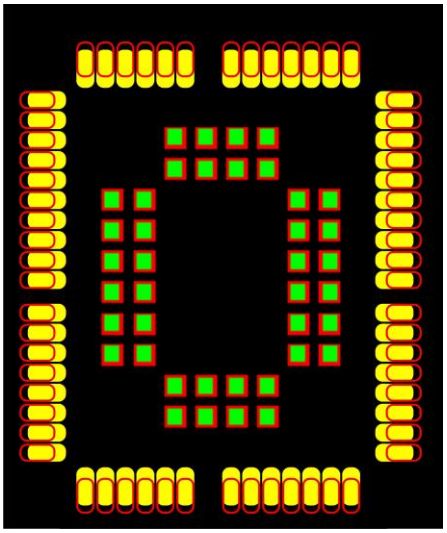
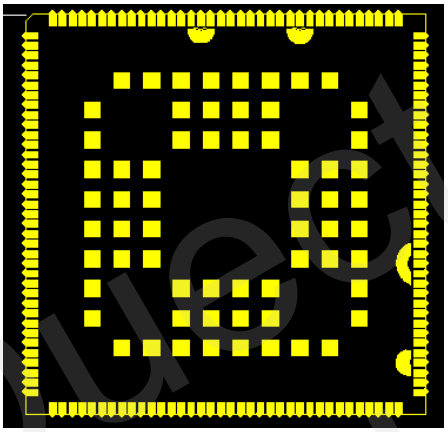
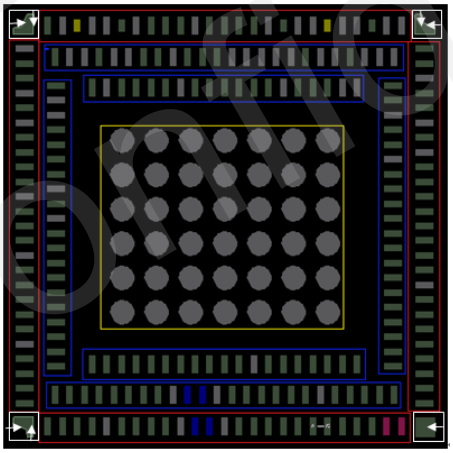
To ensure the solder paste is enough and soldering joints are reliable, the stencil should be partly stepped-up on the top surface. And a single pad cannot be larger than 3.0×4.0mm, otherwise dividing it into several smaller pads less than 2.0×2.0mm by 0.3-0.5mm shelves.

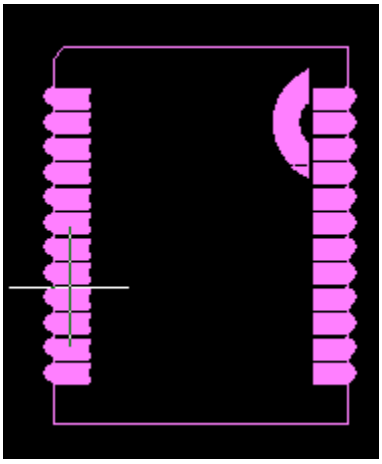
The stencil design requirements are shown in the table below.

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 0.20mm. 2. The stencil openings 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 openings should be shrunken inward by 0.10mm (refer to h1) and moved outward by 0.30-0.50mm (refer to h2).

MC60		1. The thickness of stencil should be stepped-up to 0.20mm. 2. The stencil openings should be shrunken inward by 0.10mm and moved outward by 0.40mm. 3. Cut four 0.55×0.55mm openings with 0.05mm square chamfer on the pads in the center.
UC20/UC15		1. The thickness of stencil should be stepped-up to 0.18mm. 2. The stencil openings should be shrunken inward by 0.30mm and moved outward by 0.40mm. 3. Cut four 1.00×0.65mm openings with 0.05mm square chamfer on each grounding pads, and with 0.25m space in between. 4. The 12 pads in the center are used for R&D test and recommended to be kept intact.
EC20/EC20 R2.0/ EC21/EC25		1. The thickness of stencil should be stepped-up to 0.20mm. 2. The stencil openings should be shrunken inward by 0.20mm and moved outward by 0.40mm. 3. Cut four 1.00×0.65mm openings with 0.05mm square chamfer on each grounding pads, and with 0.25m space in between. 4. Cut a round opening with a diameter of 0.70mm for the pads in the yellow box. 5. The 12 pads in the center are used for R&D test and recommended to be kept intact.

UG95/UG96/BC95/ EG91*/EG95*/BG96*		1. The thickness of stencil should be stepped-up to 0.13mm for UG95/UG96, 0.15mm for BC95 and 0.18mm for EG91/EG95/ BG96. 2. The stencil openings should be shrunken inward by 0.30mm and moved outward by 0.40mm. 3. Cut four 1.00×1.00mm openings with 0.05mm square chamfer on the pads in the center.
SC20		1. The thickness of stencil should be stepped-up to 0.18mm. 2. The stencil openings should be shrunken inward by 0.10mm and moved outward by 0.40mm. 3. Cut four 0.55×0.55mm openings with 0.05mm square chamfer on the pads in the center.
SG30*/AG35*	 	1. The thickness of stencil should be stepped-up to 0.15-0.18mm. 2. Pads in the red boxes should be shrunken inward by 0.05mm on the both sides of the width direction, and moved outward by 0.10mm along the direction of length. 3. Pads in the blue boxes should be shrunken inward by 0.05mm on the both sides of the width direction. 4. Grounding circular pads in the yellow box should be made as the figure shown at the bottom (the section in grey is the stencil opening with an area of about 60% of the pad. 5. The other stencil openings should be 100% cut.

FC10/FC20		1. The thickness of stencil should be stepped-up to 0.18mm. 2. The stencil openings should be shrunken inward by 0.20mm, and moved outward by 0.40mm along the direction of length, and shrunken inward by 0.05mm on the both sides of the width direction, and cut with 0.05mm square chamfer.

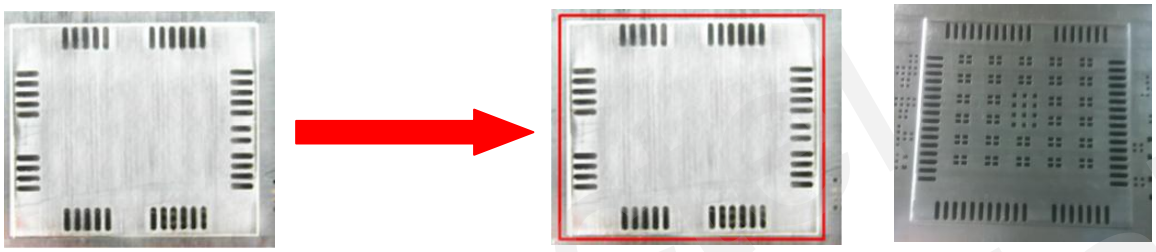


Figure 3: Step-up Stencil Area

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 packing, to ensue mounting accuracy, dedicated tray fixture is recommended when placing the module.

4.3.2. Automatic Placement

Select the 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. Module's pad should be put on the solder paste of motherboard's pad after module is mounted. The triangle mark on the module indicates the 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 reflow profile in the following figure.

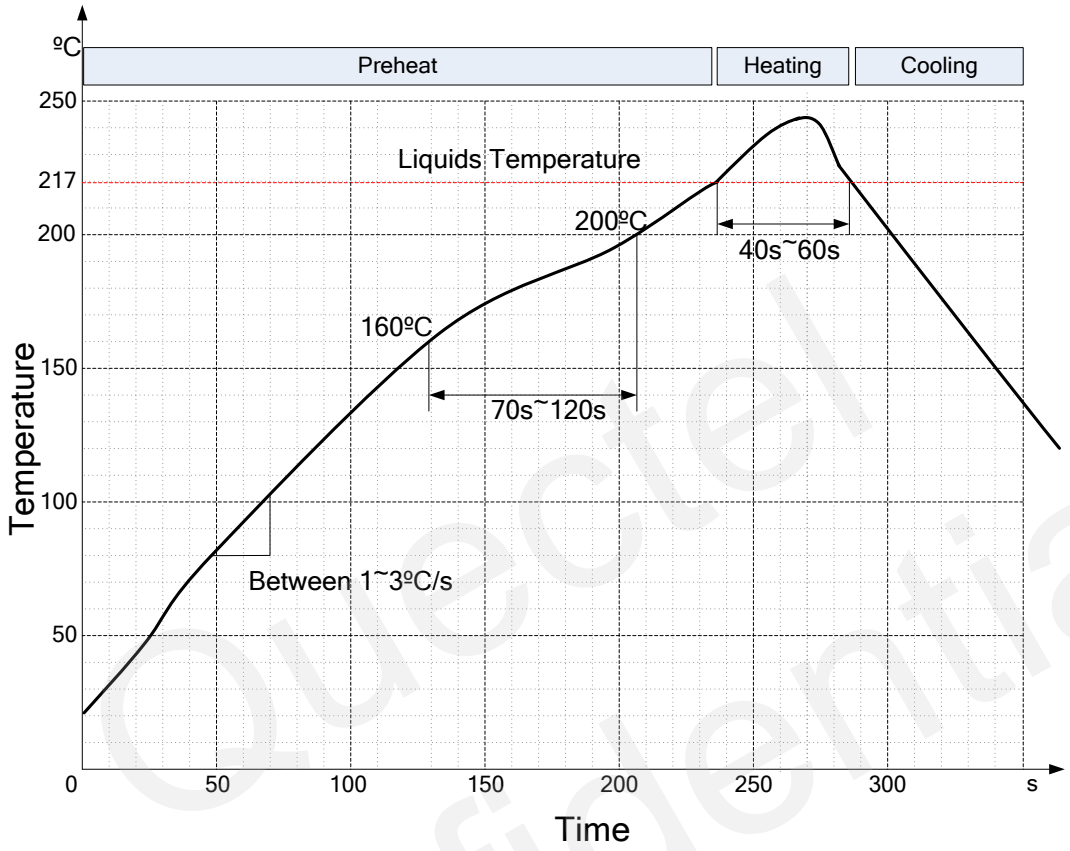


Figure 6: Ramp-soak-spike Reflow Profile

NOTE

1. You can optimize ramp-soak-spike reflow profile based on the actual situation.
2. 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.

5 Desoldering

Please use a heat gun to heat the pads of the module in 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.

When the motherboard is heated, the distance between the motherboard and the wind outlet should be from 1.0cm to 3.5cm. Move the wind outlet along the edge of the module in uniform rotation. When all of the solder are melted, 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 33.0×33.0mm, a BGA workbench or heat gun can be used to desolder components; to prevent separation between pad and circuit caused by heating on a single surface, pre-heating is needed at the bottom of the module; if PCB is blistered, it is recommended to inspect the bottom of the module by X-rays.

Table 2: Desoldering Requirements

Parameters	Requirements
The maximum temperature on the surface of PCB	260°C
Removal or soldering time limit	40s-120s
Temperature measurement and calibration	Use tempature measurement devices in caliberation period to measure the tempature (the heat gun tempature must be set according to the actually soldering requirements). The tempature must not exceed 350°C. Check point must be kept 5mm away from the outlet of the heat gun. Nozzle must be placed vertically down when measuring. Devices inconsistent with temperature requirements are prohibited to be used. Heat gun should be detected with grounding.
Nozzle shape and dimensions	Select an appropriate nozzle according to electronic components.
Fixture	Use dedicated fixtures for the module to keep the components still when dissembling.

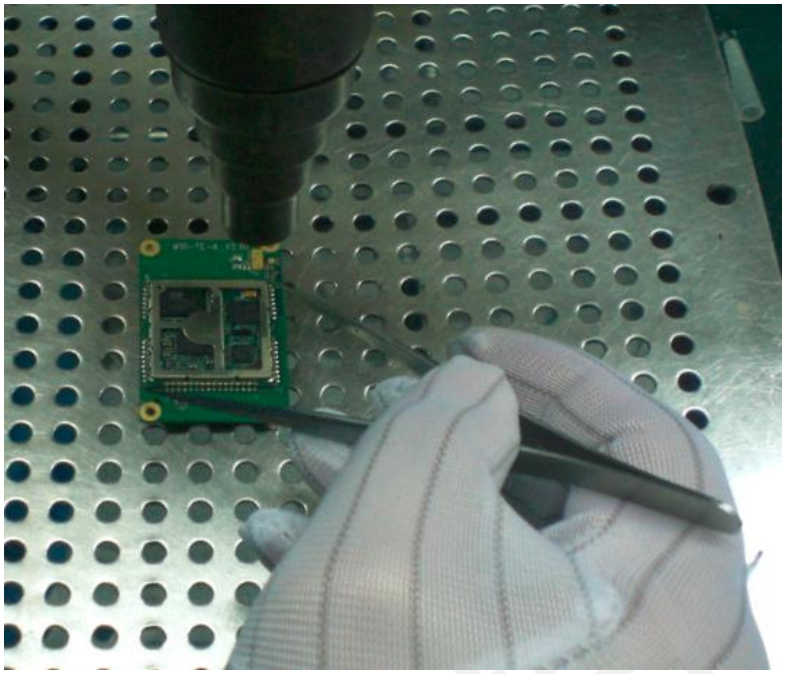


Figure 7: Remove Module

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.

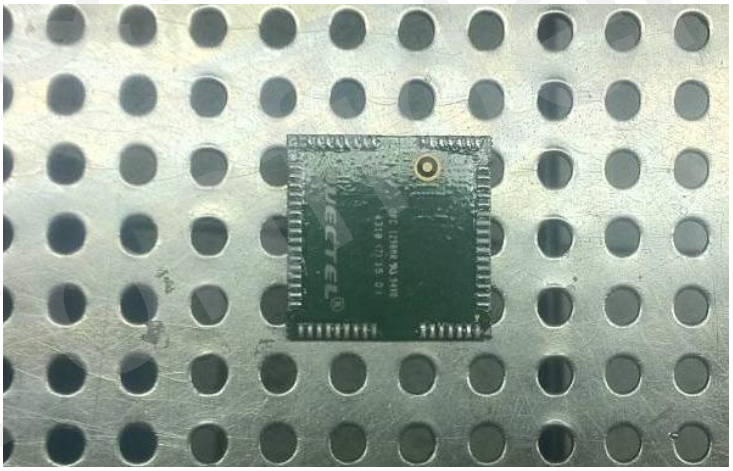


Figure 8: Module Soldering

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 board 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 board will be heated twice for every repair (desoldering and soldering), and the maximum repair time for each PCBA board is 3. If the module is not restored after three times, it is recommended to be scrapped.

7 Appendix Reference

Table 3: 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