

# Module Secondary SMT User Guide

**LCC/LGA Module Series**

Rev. Module\_Secondary\_SMT\_User\_Guide\_V2.5

Date: 2019-03-11

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.**

7<sup>th</sup> Floor, Hongye Building, No.1801 Hongmei Road, Xuhui District, Shanghai 200233, China

Tel: +86 21 5108 6236

Email: [info@quectel.com](mailto:info@quectel.com)

**Or our local office. For more information, please visit:**

<http://www.quectel.com/support/sales.htm>

**For technical support, or to report documentation errors, please visit:**

<http://www.quectel.com/support/technical.htm>

Or email to: [support@quectel.com](mailto: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**

THE INFORMATION CONTAINED HERE IS PROPRIETARY TECHNICAL INFORMATION OF QUECTEL WIRELESS SOLUTIONS CO., LTD. TRANSMITTING, REPRODUCTION, DISSEMINATION AND EDITING OF THIS DOCUMENT AS WELL AS UTILIZATION OF THE CONTENT ARE FORBIDDEN WITHOUT PERMISSION. OFFENDERS WILL BE HELD LIABLE FOR PAYMENT OF DAMAGES. ALL RIGHTS ARE RESERVED IN THE EVENT OF A PATENT GRANT OR REGISTRATION OF A UTILITY MODEL OR DESIGN.

***Copyright © Quectel Wireless Solutions Co., Ltd. 2019. All rights reserved.***

# 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	<ol style="list-style-type: none"> <li>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.</li> <li>2. Added desoldering and repair instructions in Chapter 5 and 6.</li> </ol>
2.4	2018-06-02	Rowan WANG/ Alain HUANG	<ol style="list-style-type: none"> <li>1. Updated the MSL rating of Quectel modules into 3.</li> <li>2. Updated stencil design requirements in Chapter 4.2.</li> <li>3. Optimized the recommended reflow soldering requirements and thermal profile in Chapter 4.4.</li> </ol>
2.5	2019-03-11	Alain HUANG	<ol style="list-style-type: none"> <li>1. Deleted the tray packing in the packing methods (Figure 2) in Chapter 2.2.</li> <li>2. Updated the soldering requirements in Chapter 3.2.</li> <li>3. Updated stencil design requirements in Chapter 4.2.</li> <li>4. Updated the max temperature of reflow zone in Chapter 4.4.</li> </ol>

## Content

About the Document .....	2
Content .....	3
Table Index .....	4
Figure Index .....	5
<b>1 Introduction .....</b>	<b>6</b>
<b>2 Information about Modules .....</b>	<b>7</b>
2.1. Surface-Mount Packaging Type .....	7
2.2. Packing Methods .....	7
<b>3 Requirements on Chip Mounter .....</b>	<b>8</b>
3.1. Chip Mounter .....	8
3.2. Soldering Requirements .....	8
<b>4 Attentions for Manufacturing .....</b>	<b>9</b>
4.1. MSL and Moisture-proof Requirement .....	9
4.2. Stencil Design Requirements .....	10
4.3. Mounting Process .....	18
4.3.1. Load Materials .....	18
4.3.2. Automatic Placement .....	19
4.4. Reflow Soldering .....	20
<b>5 Desoldering .....</b>	<b>22</b>
<b>6 Repair Instructions .....</b>	<b>24</b>
<b>7 Appendix Reference .....</b>	<b>25</b>

## Table Index

TABLE 1: STENCIL DESIGN REQUIREMENTS .....	10
TABLE 2: RECOMMENDED THERMAL PROFILE PARAMETERS .....	20
TABLE 3: HEAT GUN DESOLDERING REQUIREMENTS.....	22
TABLE 4: TERMS AND ABBREVIATIONS .....	25

## Figure Index

FIGURE 1: TAPE & REEL PACKING .....	7
FIGURE 2: HUMIDITY INDICATOR CARD .....	9
FIGURE 3: STEP-UP STENCIL AREA.....	18
FIGURE 4: AUTOMATIC PLACEMENT .....	19
FIGURE 5: FIRST PIN AND MOUNTED PICTURE .....	19
FIGURE 6: RAMP-SOAK-SPIKE REFLOW PROFILE.....	20
FIGURE 7: REMOVE MODULE .....	23
FIGURE 8: MODULE SOLDERING QUALITY INSPECTION.....	23

# 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 tape and reel pack as shown in the following figure.



Figure 1: 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 238°C, and the temperature of fixtures is recommended to be 240-245°C to avoid cold solder joints on LGA modules.

# 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.

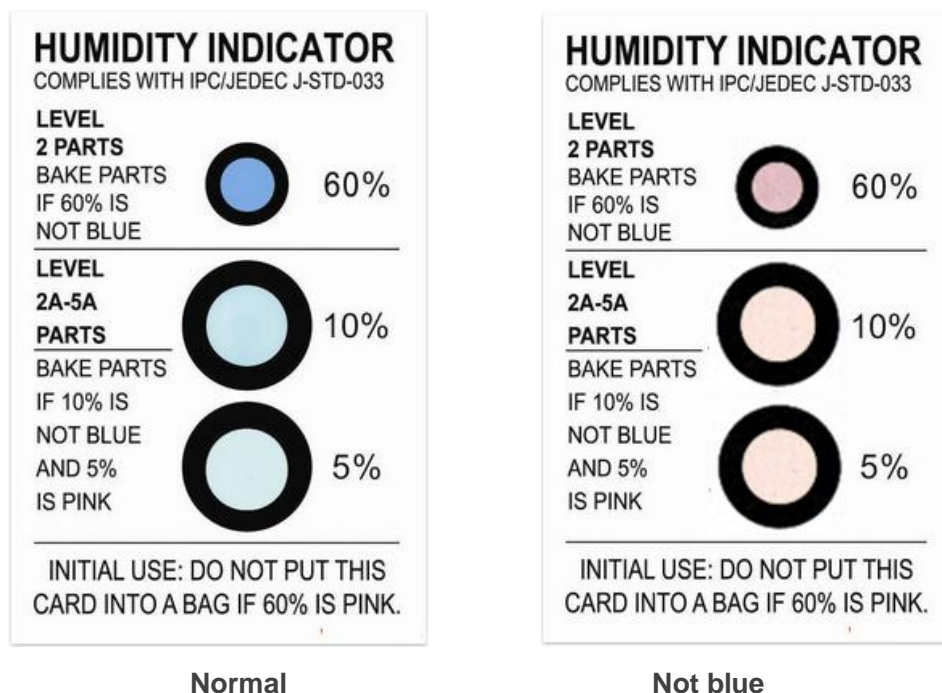


Figure 2: Humidity Indicator Card

## NOTES

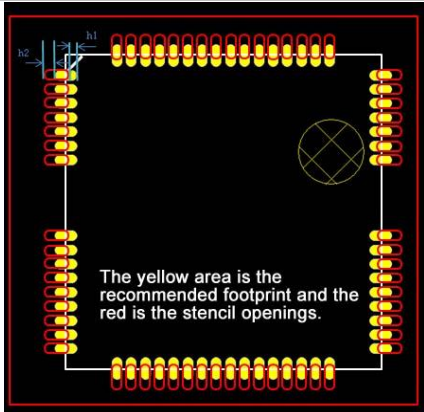
1. If baking is required, the module should be baked for 8 hours at  $120^{\circ}\text{C}\pm 5^{\circ}\text{C}$ .
2. Please take out the module from the package and put it on high-temperature resistant fixtures before baking. All modules must be mounted within 24 hours after finishing baking, otherwise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

## 4.2. Stencil Design Requirements

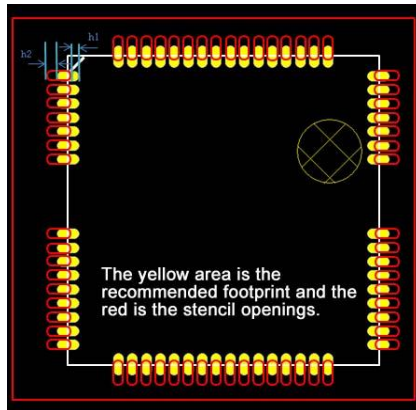
To ensure the solder paste is enough and soldering joints are reliable, the stencil should be partly stepped-up on the top surface. And the stencil aperture for each single pad cannot be larger than  $3.0\text{mm}\times 4.0\text{mm}$  and the exceeded part should be divided into smaller apertures with size less than  $2.0\text{mm}\times 2.0\text{mm}$  by  $0.3\text{-}0.5\text{mm}$  shelves. There is no need of opening for arc-shaped pad near regular pads; if there is any component, a clearance of over  $1\text{mm}$  should be left between outward end of the aperture and the component.

The stencil design requirements for Quectel modules are shown in the table below. Diagrams in the table are only typical examples of corresponding modules. Diagrams of different modules listed in the same row may vary but the stencil requirements for them are identical.

**Table 1: Stencil Design Requirements**

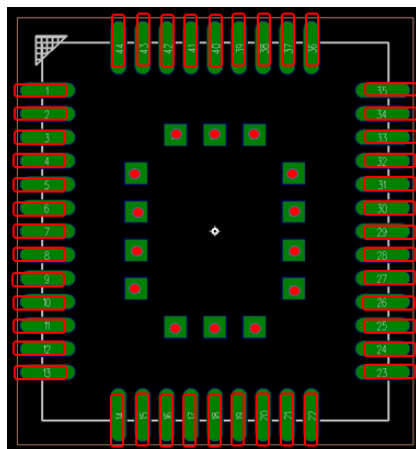
Module	Diagram for Stencil Designs	Requirement Description
M10/M12/M72/ M80/M85/M95/ M66/M66-DS/ GC10	 <p>The yellow area is the recommended footprint and the red is the stencil openings.</p>	<ol style="list-style-type: none"> <li>1. The stencil thickness of the area for the module should be partly stepped-up to <math>0.18\text{-}0.20\text{mm}</math>.</li> <li>2. The innermost edge of the aperture for each single pad should be shifted outward by <math>0.10\text{mm}</math> (refer to <math>h1</math>) and the outermost edge should be shifted outward by <math>0.30\text{-}0.50\text{mm}</math> (refer to <math>h2</math>).</li> </ol>

L10/L16/L20/L26/  
L30/L50/L70/L76/  
L80/L70-R/  
L70-RL/L76-L/  
L76B/L80-R/L86/  
L96



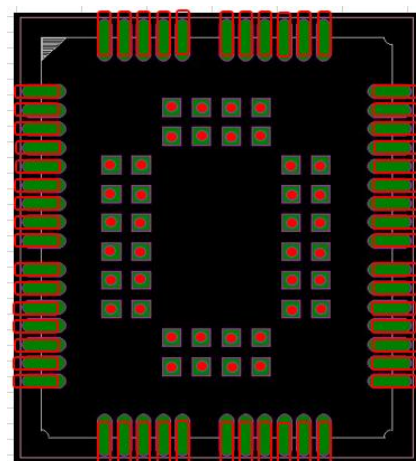
1. The stencil thickness of the area for the module should be partly stepped-up to 0.13-0.15mm.
2. The innermost edge of the aperture for each single pad should be shifted outward by 0.10mm (refer to h1) and the outermost edge should be shifted outward by 0.30-0.50mm (refer to h2).

BC66/BC68



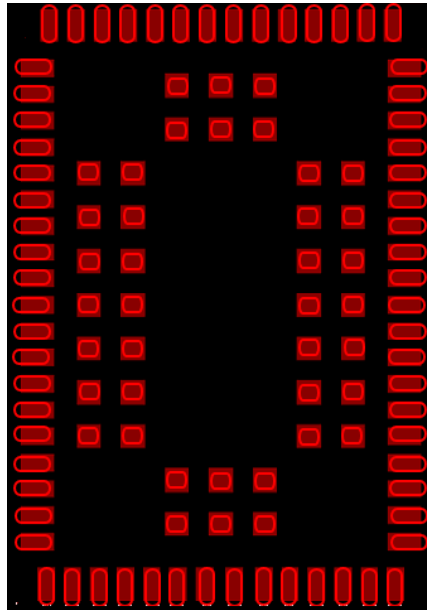
1. The stencil thickness of the area for the module should be partly stepped-up to 0.18-0.20mm.
2. For pads on four sides:  
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.40-0.60mm, and the width should be reduced in a ratio of 1:0.85. The shape should be rectangle with round chamfers.
3. For pads in the center:  
The stencil aperture area for each single pad should be 70% of that of the corresponding pad and the shape should be round.

BC95/BC95-G



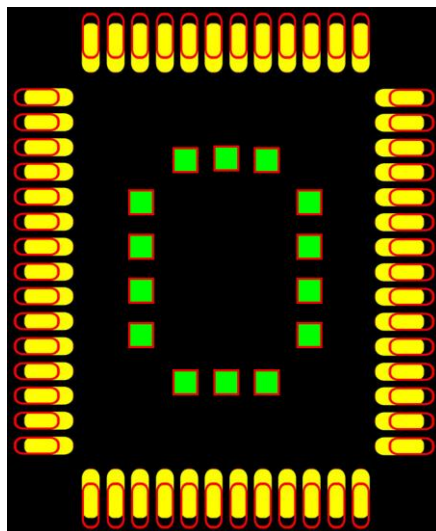
1. The stencil thickness of the area for the module should be partly stepped-up to 0.18-0.20mm.
2. For pads on four sides:  
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.40-0.60mm, and the width should be reduced in a ratio of 1:0.85. The shape should be rectangle with round chamfers.
3. For pads in the center:  
The stencil aperture area for each single pad should be 70% of that of the corresponding pad and the shape should be round.

M89



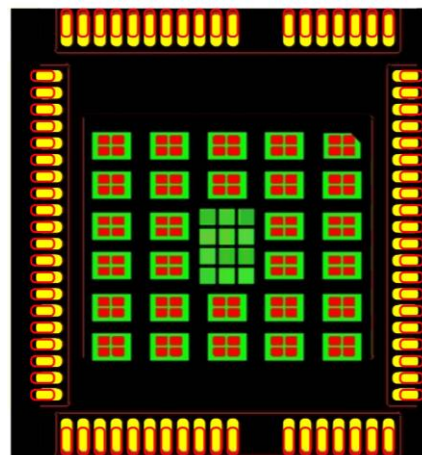
1. The stencil thickness of the area for the module should be partly stepped-up to 0.18-0.20mm.
2. For pads on four sides:  
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and outermost edge should be shifted outward by 0.40-0.60mm, and the width should be reduced in a ratio of 1:0.85. And the shape should be rectangle with rectangle with round chamfer.
3. For pads in the center:  
The stencil aperture for each single pad should be centered with area reduced to 70%, and should be designed with round chamfers.

MC60



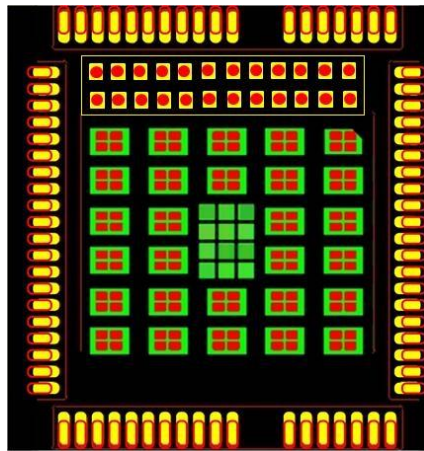
1. The stencil thickness of the area for the module should be partly stepped-up to 0.18-0.20mm.
2. For pads on four sides:  
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and outermost edge should be shifted outward by 0.40-0.60mm, while the width should be reduced in a ratio of 1:0.85. And the shape should be rectangle with round chamfers.
3. For pads in the center:  
The stencil aperture for each pad should be centered with area reduced to 70%. The shape should be square.

UC20/UC15



1. The stencil thickness of the area for the module should be partly stepped-up to 0.18-0.20mm.
2. For pads on four sides:  
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.40-0.60mm. The width should be reduced in a ratio of 1:0.85. And the shape should be rectangle with round chamfers.
3. For GND pads in the center:  
Design the stencil aperture for each pad into

EC20/EC20 R2.0/  
EC21/EC25/  
EC20 R2.1



four 1.00mm×0.65mm smaller apertures shaped in 0.05mm square with round chamfers, and with clearance of 0.25mm in between.

4. The 12 pins in the very center are used for R&D test and recommended to be kept intact.

1. The stencil thickness of the area for the module should be partly stepped-up to 0.18-0.20mm.

2. For pads on four sides:

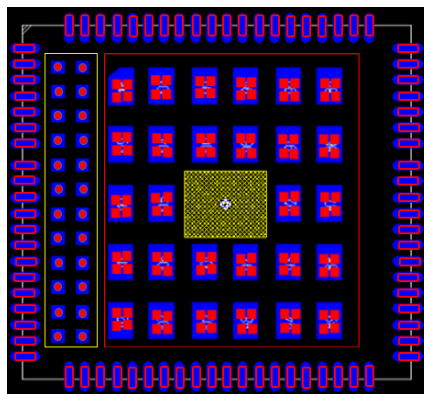
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.40-0.60mm. The width should be reduced in a ratio of 1:0.85. And the shape should be rectangle with round chamfers.

3. For GND pads in the center:

Design the stencil aperture for each pad into four smaller 1.00mm×0.65mm apertures shaped in 0.05mm square with round chamfers, and with clearance of 0.25mm in between.

4. Design round apertures with diameter of 0.70mm for the pads in the yellow box.
5. The 12 pins in the very center are used for R&D test and recommended to be kept intact.

EG25-G



1. The stencil thickness of the area for the module should be partly stepped-up to 0.13-0.15mm.

2. For pads on four sides:

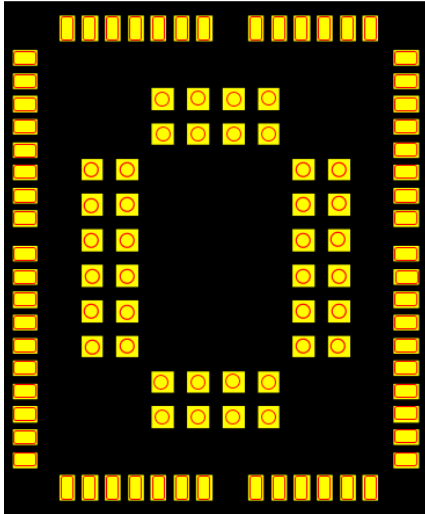
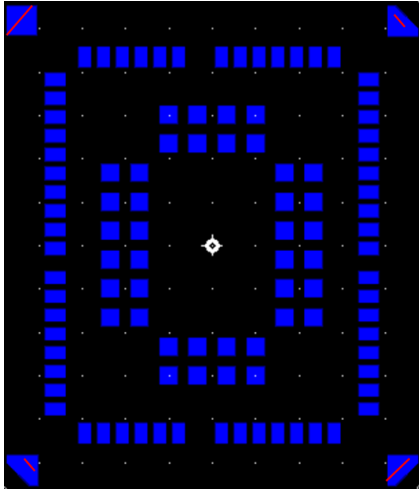
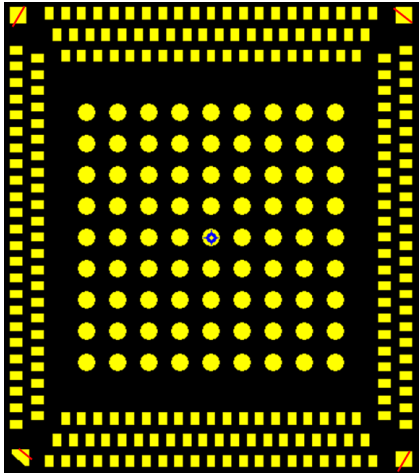
The stencil aperture should be centered with area reduced to 75%-85% and the shape should be rectangle with round chamfers.

3. For pads in the center (in the red box):

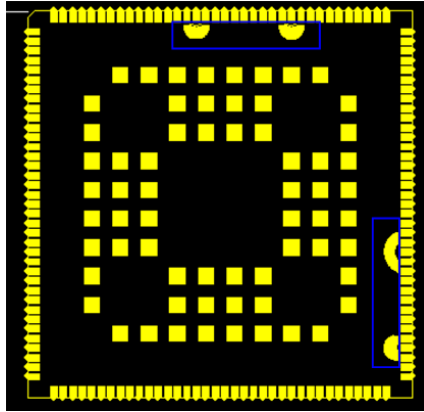
Design four square apertures centered in each pad of which the total area is 70% of that of the pad.

4. The apertures for pads in the yellow box should be centered with area reduced to 70% and the shape should be round.



	<p>5. The yellow block in the very center should be kept intact.</p>
<p>UG95/UG96/ BG96/BG95</p>	 <ol style="list-style-type: none"> <li>1. The stencil thickness of the area for the module should be partly stepped-up to 0.13-0.15mm.</li> <li>2. For pads on four sides: The aperture for each single pad should be centered with area reduced to 75%-85%. And the shape should be rectangle with round chamfers.</li> <li>3. For square pads in the center: The aperture for each single pad should be designed into round and centered with area reduced to 70%.</li> </ol>
<p>EG91/EG95</p>	 <ol style="list-style-type: none"> <li>1. The stencil thickness of the area for the module should be partly stepped-up to 0.13-0.15mm.</li> <li>2. For pads on four sides: The aperture for each single pad should be centered with area reduced to 75%-85%. And the shape should be rectangle with round chamfers.</li> <li>3. For square pads in the center: The aperture for each single pad should be designed into round and centered with area reduced to 70%.</li> <li>4. For pads at four corners: The stencil aperture should be designed into diagonal-patterned lines with 60% area of the corresponding pad.</li> </ol>
<p>EG06/EG12</p>	 <ol style="list-style-type: none"> <li>1. The stencil thickness of the area for the module should be partly stepped-up to 0.13-0.15mm.</li> <li>2. For pads on four sides: The aperture for each single pad should be centered with area reduced to 75%-85%. And the shape should be rectangle with round chamfers.</li> <li>3. For round pads in the center: The aperture should be centered with area reduced to 70%. And the shape should be round.</li> <li>4. For pads at four corners:</li> </ol>

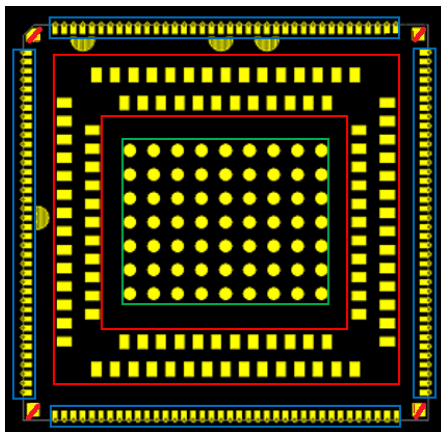
SC20



The stencil aperture should be designed into diagonal-patterned lines with 60% area of the corresponding pad.

1. The stencil thickness of the area for the module should be partly stepped-up to 0.18-0.20mm.
2. For pads on four sides:  
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.60-0.80mm, while width should be reduced in a ratio of 1:0.85. And should be shaped into rectangle with round chamfers.
3. For square pads in the center, the stencil aperture area should be 70% of that of the corresponding pad.
4. There is no need to design stencil aperture for the arc-shaped pad in blue box.

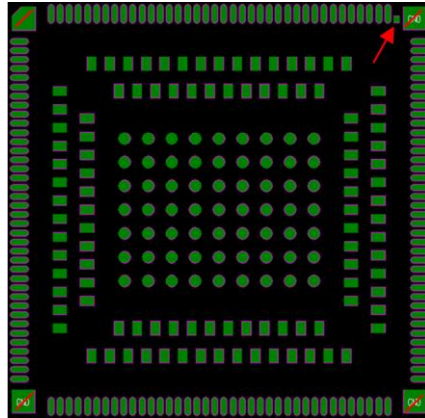
SC60/SC600T/  
SC600Y



1. The stencil thickness of the area for the module should be partly stepped-up to 0.18-0.20mm.
2. For pads on four sides (in blue boxes), the innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.60-0.80mm while the width should be reduced in a ratio of 1:0.85. And the shape should be rectangle with round chamfers.
3. For square pads between the two red boxes, the stencil aperture area should be centered with area reduced to 85% and designed with round chamfers.
4. For round pads in the center (in the green box), the stencil aperture should be centered with area reduced to 70%.
5. For pads at the four corners of the module, the stencil aperture should be designed into diagonal-patterned lines with 60% area of the corresponding pad.
6. There is no need to design stencil aperture for the four arc-shaped pads.

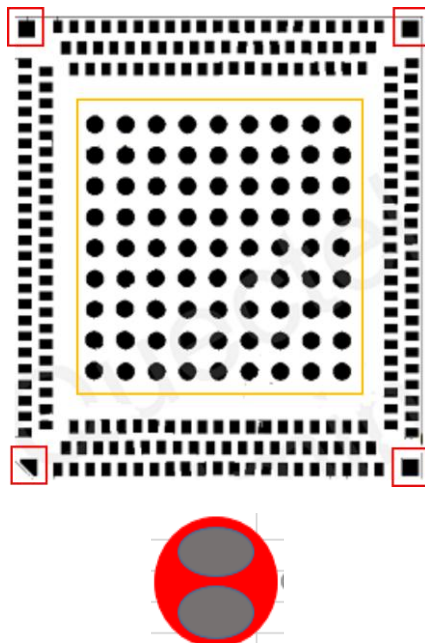


SC66



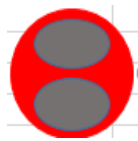
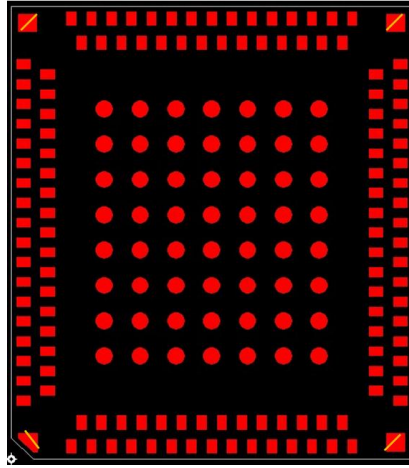
1. The stencil thickness of the area for the module should be partly stepped-up to 0.18-0.20mm.
2. For pads on four sides:  
The innermost edge of aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.60-0.80mm, while the width should be reduced in a ratio of 1:0.85. And the shape should be rectangle with round chamfers.
3. For square pads in the center:  
the stencil aperture area should be centered with area reduced to 85% and designed with round chamfers.
4. For round pads in the very center:  
The aperture should be centered with area reduced to 70%.
5. For pads at four corners:  
the stencil aperture should be designed into diagonal-patterned lines with 60% area of the corresponding pad.
6. For the pad marked with red arrow:  
The aperture should be centered with area reduced to 70%-80%.

AG35



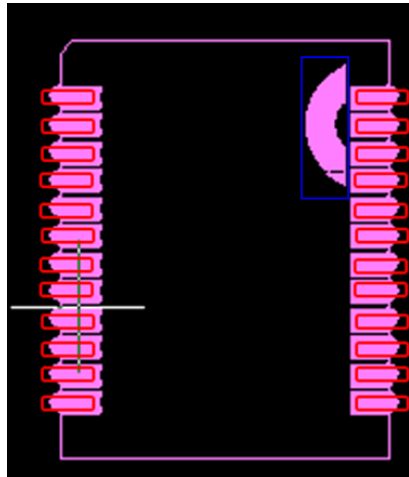
1. The stencil thickness of the area for the module should be partly stepped-up to 0.13-0.15mm.
2. The stencil aperture area for the pads on four sides should be centered with 75%-85% area of corresponding pads, and should be designed with round chamfers.
3. For pads at four corners (marked with red boxes), the stencil aperture should be reduced inward as marked by the white arrows in the shape of diagonal-patterned lines with 60% area of the corresponding pad.
4. The stencil aperture for round GND pads in the yellow box should be designed into 2 ovals with 60%-70% area of the pad, as shown by the 2 grey parts in the bottom right figure.

AG15



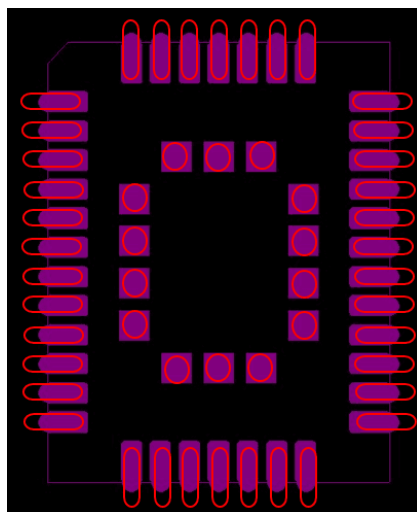
1. The stencil thickness of the area for the module should be partly stepped-up to 0.13-0.15mm.
2. The stencil aperture area for the pads on four sides should be centered with area reduced to 75%-85%, and should be designed with round chamfers.
3. For pads at four corners, the aperture should be centered and designed into diagonal-patterned lines with 60% area of the corresponding pad.
4. The aperture for round GND pads in the center should be designed into 2 ovals with 60%-70% area of the pad, as shown by the 2 grey parts in the figure at the bottom.

FC10



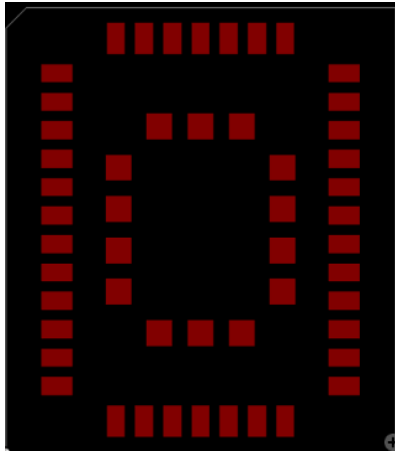
1. The stencil thickness of the area for the module should be partly stepped-up to 0.15-0.18mm.
2. The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.40mm, and the width should be reduced in a ratio of 1:0.85, and should be designed with round chamfers.
3. There is no need to design stencil apertures for the arc-shaped pad in blue box.

FC20



1. The stencil thickness of the area for the module should be partly stepped-up to 0.15-0.18mm.
2. For pads on four sides:  
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.40-0.60mm while the width should be reduced in a ratio of 1:0.85, and should be designed in rectangle with round chamfers.
3. For pads in the center:  
The stencil aperture should be designed into round with area reduced to 70%.

AF20



1. The stencil thickness of the area for the module should be partly stepped-up to 0.13-0.15mm.
2. For pads on four sides:  
The stencil aperture for each pad should be centered with area reduced to 75%-85% and the shape should be rectangle with round chamfers.
3. For square pads in the center:  
The stencil aperture should be designed into round with area reduced to 70%.

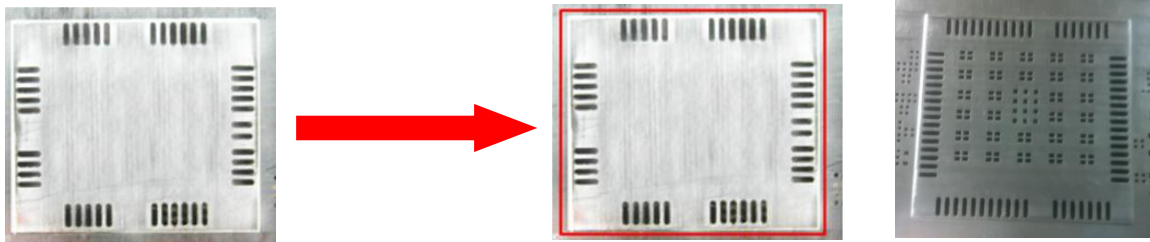


Figure 3: Step-up Stencil Area

#### NOTES

1. “\*” means under development.
2. Area of the stencil apertures for components within 1mm from the thickened stencil part should be reduced by 10%-30% compared with regular apertures, since the thickening will increase the volume of solder paste. For components with 0.5mm pitch (or smaller) or 0201 components, please keep at least 3mm space in between, otherwise there may be a risk of solder bridging that will cause short circuit of the component.
3. You can optimize the stencil design depending on the actual situation.
4. Inward shrinking and outward extending are relative to the host PCB footprint of the module. For details of the recommended footprint, please refer to the hardware designs of corresponding modules.

## 4.3. Mounting Process

### 4.3.1. Load Materials

In order to ensure mounting accuracy, for single modules that have been unpacked for baking, it is recommended for the customers to use a 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.

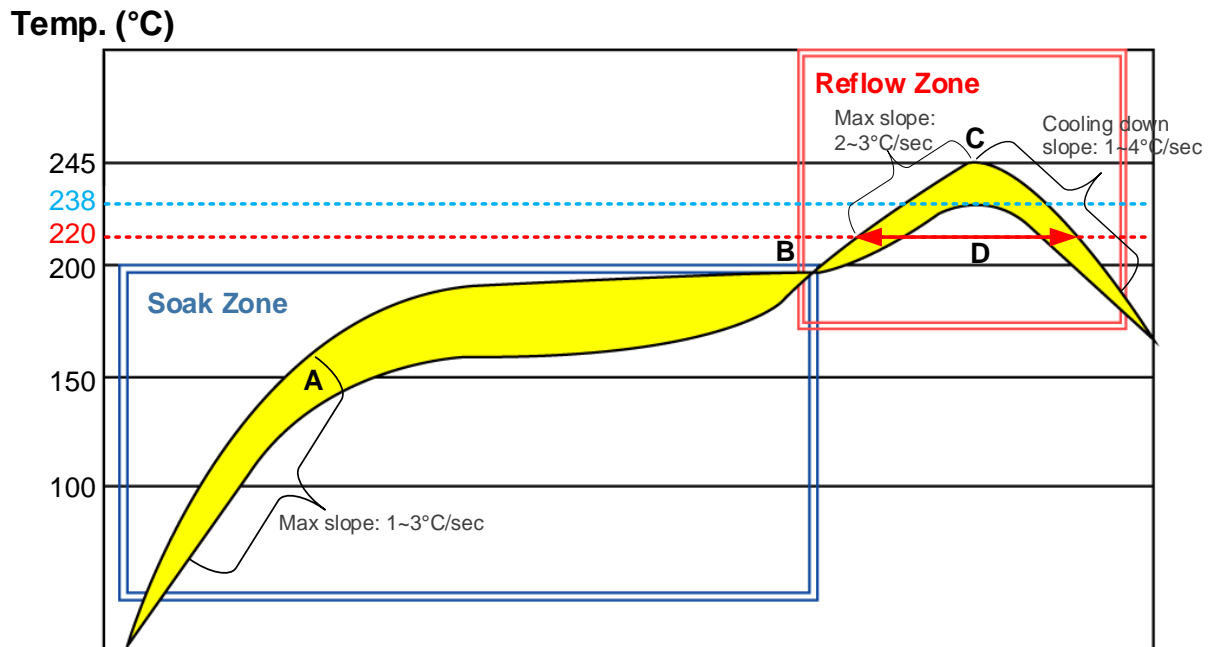


Figure 6: Ramp-soak-spike Reflow Profile

Table 2: Recommended Thermal Profile Parameters

Factor	Recommendation
<b>Soak Zone</b>	
Max slope	1 to 3°C/sec
Soak time (between A and B: 150°C and 200°C)	60 to 120 sec
<b>Reflow Zone</b>	
Max slope	2 to 3°C/sec
Reflow time (D: over 220°C)	40 to 60 sec
Max temperature	238°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 350°C 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 constant 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 as well as PCB blistering 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. 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	<ul style="list-style-type: none"> <li>● Use temperature measurement devices in calibration period to measure the temperature (the heat gun temperature must be set according to the actually soldering requirements).</li> <li>● The temperature must not exceed 350°C.</li> <li>● Temperature check point must be 5mm away from the nozzle of the heat gun, and the nozzle must be placed vertically down when measuring.</li> <li>● Heat guns that cannot meet the temperature requirements are prohibited to be used.</li> <li>● Heat gun should be detected with grounding.</li> </ul>
Nozzle shape and dimensions	Select an appropriate nozzle according to the type of electronic components.

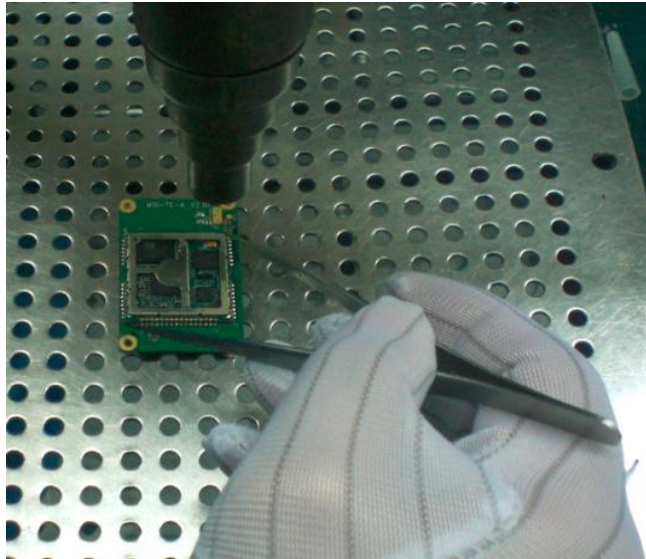


---

Fixture

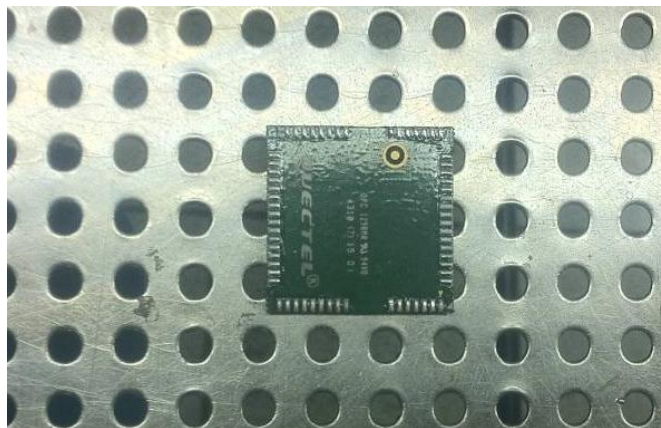
Use dedicated fixtures to hold and fix the motherboard so as to keep it stay still during components removal.

---



**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