# Module Secondary SMT Application Note

**LCC/LGA Module Series**

Version: 2.9

Date: 2021-04-26

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.**
Building 5, Shanghai Business Park Phase III (Area B), No.1016 Tianlin Road, Minhang District, Shanghai 200233, China
Tel: +86 21 5108 6236
Email: 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.

## 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.

## Disclaimer

While Quectel has made efforts to ensure that the functions and features under development are free from errors, it is possible that these functions and features could contain errors, inaccuracies and omissions. Unless otherwise provided by valid agreement, Quectel makes no warranties of any kind, implied or express, with respect to the use of features and functions under development. To the maximum extent permitted by law, Quectel excludes all liability for any loss or damage suffered in connection with the use of the functions and features under development, regardless of whether such loss or damage may have been foreseeable.

## Duty of Confidentiality

The Receiving Party shall keep confidential all documentation and information provided by Quectel, except when the specific permission has been granted by Quectel. The Receiving Party shall not access or use Quectel's documentation and information for any purpose except as expressly provided herein. Furthermore, the Receiving Party shall not disclose any of the Quectel's documentation and information to any third party without the prior written consent by Quectel. For any noncompliance to the above requirements, unauthorized use, or other illegal or malicious use of the documentation and information, Quectel will reserve the right to take legal action.

## Copyright

# About the Document

## Revision History

| Version | 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.<br>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.<br>2. Updated stencil design requirements in Chapter 4.2.<br>3. Optimized the recommended reflow soldering requirements and thermal profile in Chapter 4.4. |
| 2.5 | 2019-03-11 | Alain HUANG | 1. Deleted the tray packing in the packing methods (Figure 2) in Chapter 2.2.<br>2. Updated the soldering requirements in Chapter 3.2.<br>3. Updated stencil design requirements in Chapter 4.2.<br>4. Updated the max temperature of reflow zone in Chapter 4.4. |
| 2.6 | 2019-07-15 | Alain HUANG | Updated the stencil design requirements in Chapter 4.2: updated the specifications of stencil design for SC60/ SC600T/ SC600Y; updated the picture of SC66; added stencil design requirements for RG500Q/ AG520R. |
| 2.7 | 2020-02-25 | Alain HUANG | 1. Updated the soldering requirements in Chapter 3.2. |

| | | | |
|---|---|---|---|
| | | | 2. Added contents about storage and floor life in Chapter 4.2 and Chapter 4.3.<br>3. Updated the stencil design requirements in Chapter 4.4 and added requirements for M08-R/ L89/ UC200T/ BG95/ BC92/ EG18/ EC100Y/ BG600L-M3.<br>4. Updated specifications of reflow soldering and added matters needing attention in Chapter 4.6.<br>5. Updated the sub-section and description of Chapter 5. |
| 2.8 | 2020-07-07 | Alain HUANG | 1. Updated the description of storage note in Chapter 4.3.<br>2. Updated the stencil design requirements in Chapter 4.4 and added the requirements for SC200R/ AF50T/ BG77/ AG550Q/ EG512R-EA/ RG801H and deleted EC100Y.<br>3. Updated the requirements for reflow soldering in Chapter 4.6. |
| 2.9 | 2021-04-26 | Alain HUANG | 1. Added description about the packages in Figure 1.<br>2. Updated the NOTES in Chapter 4.3.<br>3. Updated the stencil design requirements in Chapter 4.4, added the requirements for BC660K-GL/ BG770A-GL, and deleted RG801H.<br>4. Updated the reflow time and max temperature for reflow soldering in Chapter 4.6.<br>5. Added a chapter about ESD protection (Chapter 6). |

# 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.

**NOTE**

This document may be updated at any time in accordance with updates of Quectel products or for other reasons. The contents of this document are for your reference only considering the varying factors in production, such as the different designs and structures of the products and the varying production environment and equipment. The production may not be guaranteed only by following the statements, descriptions and suggestions in this document.

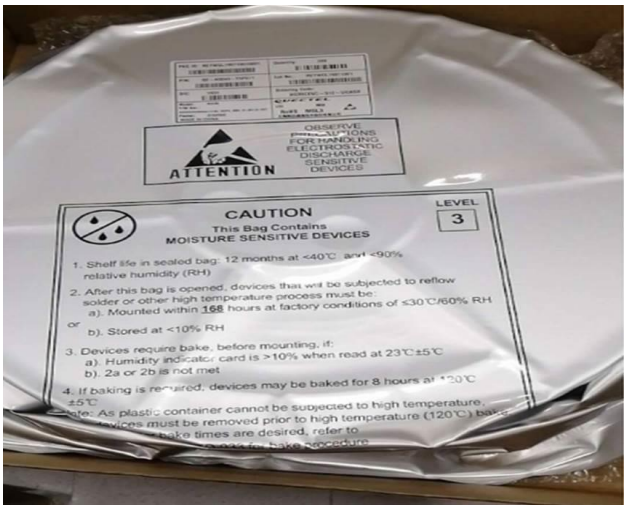
# 2 Information about Modules

## 2.1. Surface-Mount Packaging Type

Quectel modules adopt LCC and/or LGA package.

## 2.2. Packing Methods

Quectel provides vacuum tape and reel pack as shown by the following figure.




Vacuum Pack

Tape & Reel Packing Inside the Vacuum Pack

**Figure 1: Vacuum Tape & Reel Pack**

# 3 Requirements for SMT

## 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 and keep the module stable during the process.

**NOTE**

The recommended diameter of nozzle should be no less than 40 % of the module's shorter side. For example, if the module size is 25.0 mm × 20.0 mm, the nozzle diameter should be 8.0 mm 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 real peak temperature at the temperature measuring point of pads at the bottom of modules should be greater than 238 °C, and the temperature of that with SMT carrier is recommended to be 240 °C to 246 °C to avoid cold solder joints. Further, based on material's heat absorption and size of the carrier, the period in which temperature is over 217 °C should be extended by up to 10 seconds.
3. If thickness of the motherboard is less than 1.0 mm, it is recommended to use carriers when soldering to prevent distortion of the motherboard.

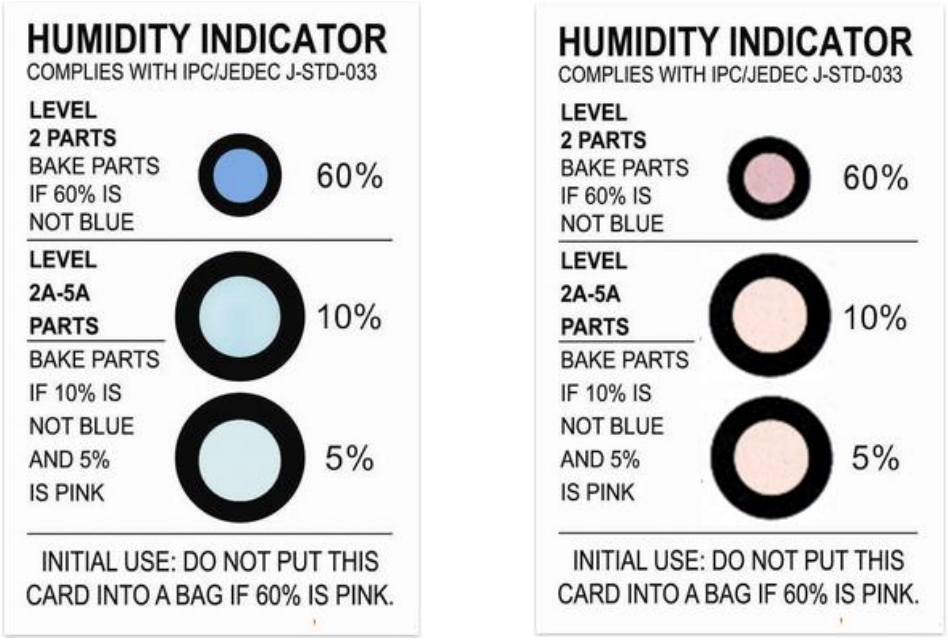
4. For the finishing of pads on the PCB, it is recommended to use the same finishing technique as the module's pin, that is, ENEG (Electroless Nickel Electroless Gold) or ENIG (Electroless Nickel Immersion Gold). And HASL (Hot-Air Solder Leveling) is not recommended.

# 4 Attentions for Manufacturing

## 4.1. MSL and Moisture-proof Requirement

Quectel SMD module is sensitive to moisture. According to IPC-JEDEC standard, the moisture sensitive level (MSL) of Quectel SMD modules is rated as “3”. Please make sure the vacuum 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 circumstance below occurs:

- Humidity indicator card: At least one circular indicator is no longer blue.




**Normal** **Not blue**

**Figure 2: Humidity Indicator Card**

## 4.2. Storage

- **Recommended Storage Condition:**
  The temperature should be 23 °C ±5 °C and the relative humidity should be 35–60 %.

- **Storage Life (in Sealed Vacuum Package):**
  12 months in Recommended Storage Condition.

## 4.3. Floor Life and Temperature&Moisture Control

Floor life refers to the allowable time period between removal of the module from a package, dry storage, or dry bake and the solder reflow process. Floor life of MSL-3 products should be 168 hours [1)]. In a plant where the temperature is 23 ±5 °C and relative humidity is below 60%, the module must be processed in solder reflow or other high-temperature operations within 168 hours after the package is removed. Otherwise, the module should be stored in environment where the relative humidity is less than 10 %, such as a dry cabinet.

**NOTES**

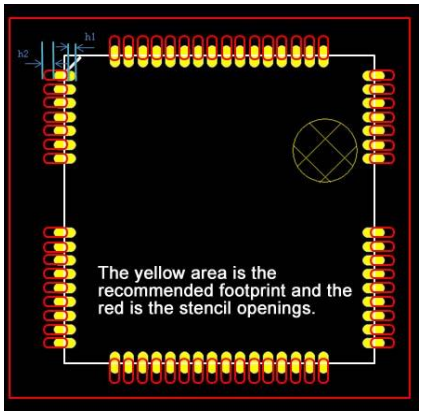
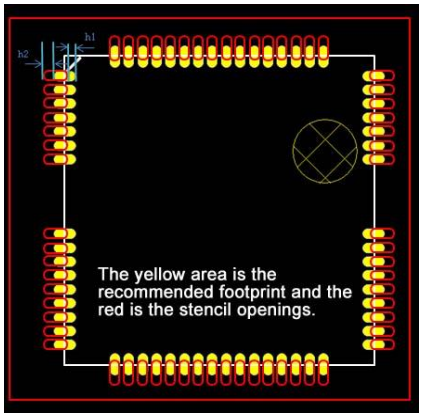
1. [1)] This floor life is only applicable when the environment conforms to *IPC/JEDEC J-STD-033*. It is recommended to start the solder reflow process **within 24 hours** after the package is removed if the temperature and moisture do not conform to, or are not sure to conform to *IPC/JEDEC J-STD-033*. And do not remove the packages of tremendous modules if they are not ready for soldering.
2. To avoid blistering, layer separation and other soldering issues, it is forbidden to expose the modules to the air for a long time.
3. In the case that a violation of moisture-proof requirement or broken vacuum package occurs, or before repairing, baking is required. In this case, the module should be baked for 8 hours at 120 ºC ±5 ºC to avoid blistering, crack and layer separation.
4. Please take out the module from the package and put it on high-temperature resistant fixtures before baking. All modules must be soldered to PCB within 24 hours after the baking, otherwise put them in the drying oven. Please pay attention to ESD protection, such as wearing anti-static gloves, when touching the modules.

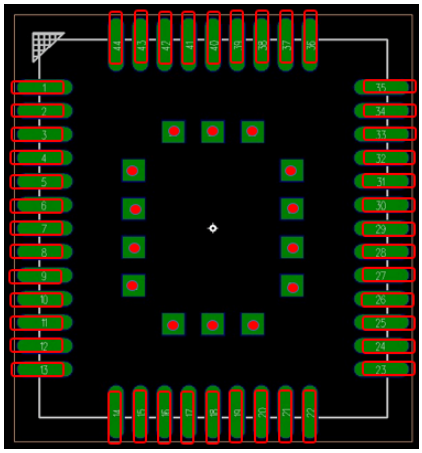
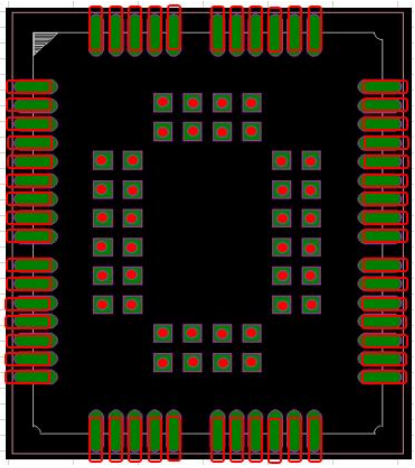
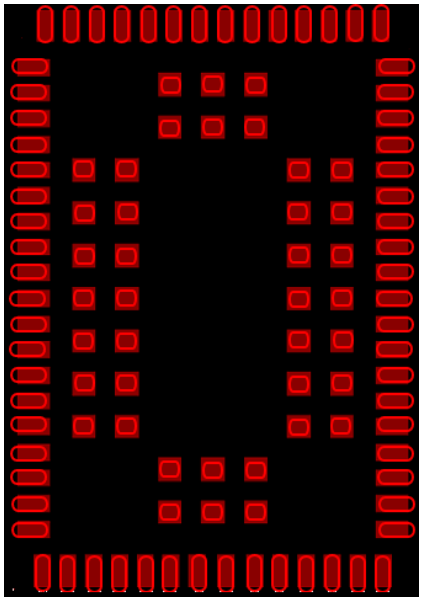
## 4.4. Stencil Design Requirements

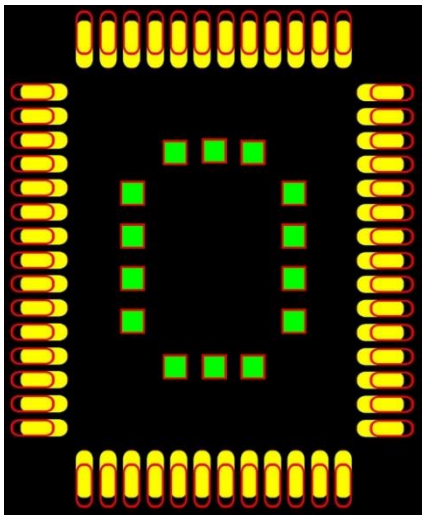
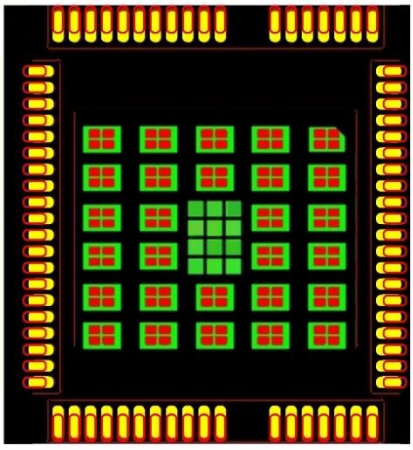
To ensure the solder paste is enough and soldering joints are reliable, the stencil should be partly stepped-up on the top surface. And the stencil aperture for each single pad cannot be larger than 3.0 mm × 4.0 mm and the exceeded part should be divided into smaller apertures with size less than 2.0 mm × 2.0 mm by 0.3–0.5 mm shelves. There is no need of opening for arc-shaped pad near regular pads; if there is any component, a clearance of over 1.0 mm should be left between outward end of the aperture and the component.

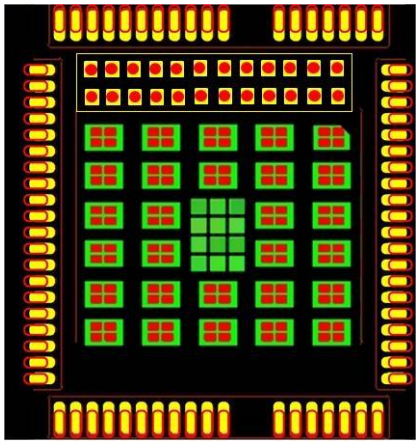
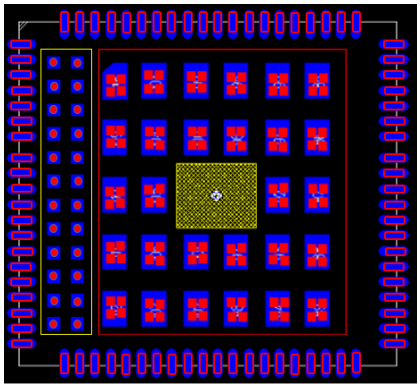
The stencil design requirements for Quectel modules are shown in the table below. Diagrams in the table are only typical examples of corresponding modules. Diagrams of different modules listed in the same row may vary but the stencil requirements for them are identical.

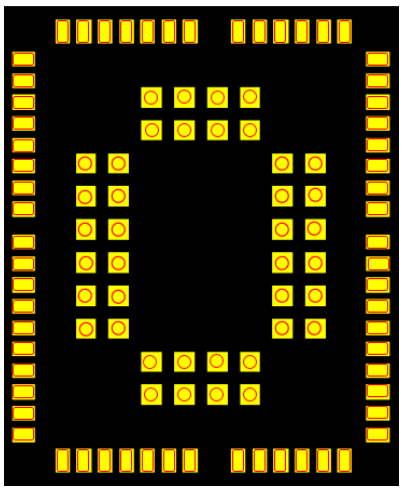
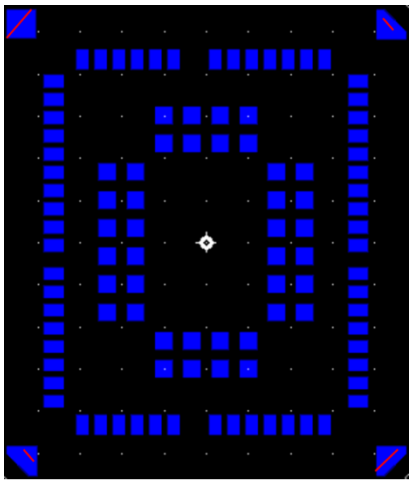
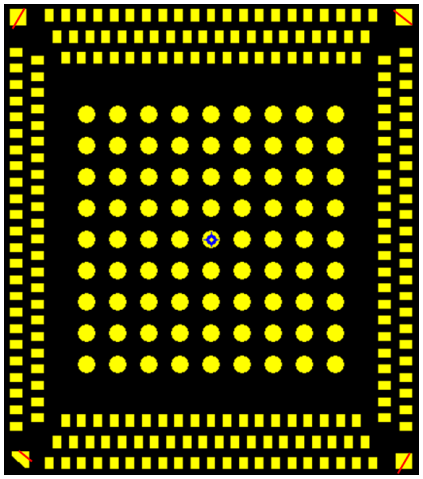
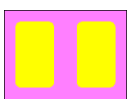
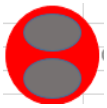
**Table 1: Stencil Design Requirements**

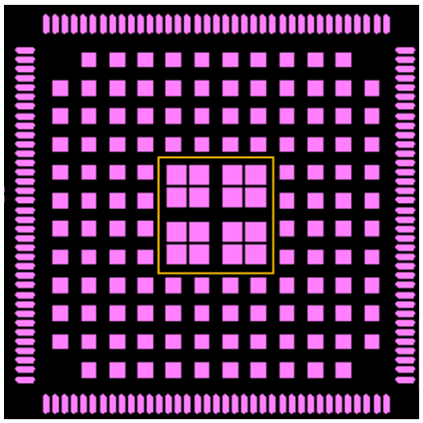
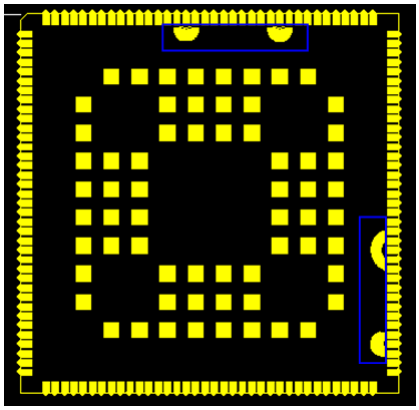
| Module | Diagram for Stencil Designs | Requirement Description |
|---|---|---|
| M08-R/ M10/ M12/ M72/ M80/ M85/ M95/ M66/ M66-DS/ GC10 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.15–0.18 mm.<br>2. The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm (refer to h1 in the left figure) and the outermost edge should be shifted outward by 0.40–0.60 mm (refer to h2 in the left figure).<br>3. The width should be reduced in a ratio of 1:0.8 or 1:0.9. |
| L10/ L16/ L20/ L26/ L30/ L50/ L70/ L76/ L80/ L70-R/ L70-RL/ L76-L/ L76B/ L80-R/ L86/ L96/ L89 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.15–0.18 mm.<br>2. The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm (refer to h1 in the left figure) and the outermost edge should be shifted outward by 0.40–0.60 mm (refer to h2 in the left figure).<br>3. The width should be reduced in a ratio of 1:0.8 or 1:0.9. |

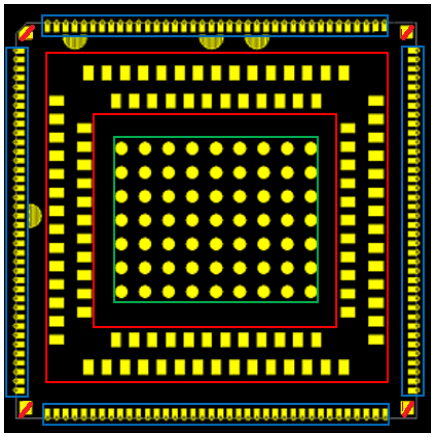
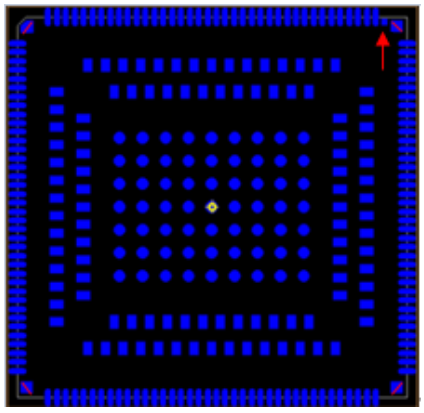
| | | |
|---|---|---|
| BC66/ BC68 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.15–0.18 mm.<br>2. For pads on four sides:<br>The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.40–0.60 mm, and the width should be reduced in a ratio of 1:0.8 or 1:0.9. The shape should be rectangle with round chamfers.<br>3. For pads in the center:<br>The stencil aperture area for each single pad should be 50–70 % of that of the corresponding pad and the shape should be round. |
| BC95-G/ BC92 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.15–0.18 mm.<br>2. For pads on four sides:<br>The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.40–0.60 mm, and the width should be reduced in a ratio of 1:0.8 or 1:0.9. The shape should be rectangle with round chamfers.<br>3. For pads in the center:<br>The stencil aperture area for each single pad should be 50–70 % 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 0.15–0.18 mm.<br>2. For pads on four sides:<br>The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.40–0.60 mm, and the width should be reduced in a ratio of 1:0.8 or 1:0.9. And the shape should be rectangle with rectangle with round chamfer.<br>3. For pads in the center:<br>The stencil aperture for each single pad should be centered with area reduced to 50–70 %, and should be designed with round chamfers. |

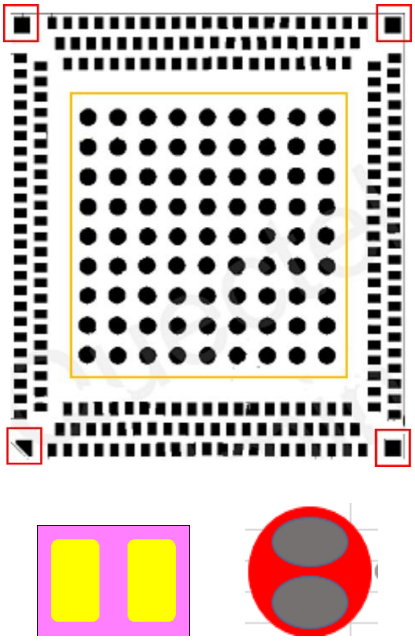
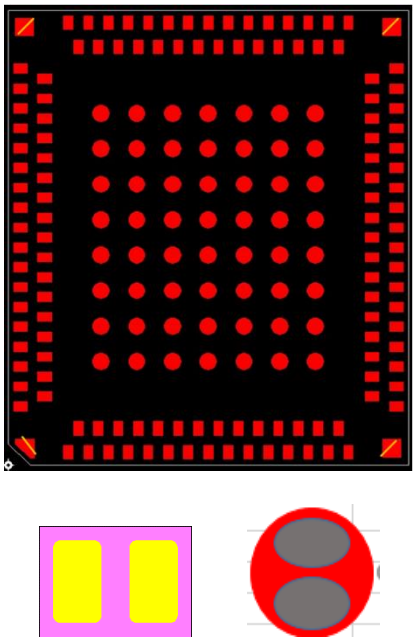
| | | |
|---|---|---|
| MC60/<br>BC660K-GL |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.15–0.18 mm.<br>2. For pads on four sides:<br>The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.40–0.60 mm, while the width should be reduced in a ratio of 1:0.8 or 1:0.9. And the shape should be rectangle with round chamfers.<br>3. For pads in the center:<br>The stencil aperture for each pad should be centered with area reduced to 50–75 %. The shape should be square. |
| UC20/ UC15 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.18–0.20 mm.<br>2. For pads on four sides:<br>The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.40–0.60 mm. The width should be reduced in a ratio of 1:0.8 or 1:0.9. And the shape should be rectangle with round chamfers.<br>3. For GND pads in the center:<br>Design the stencil aperture for each pad into four 1.00 mm × 0.65 mm smaller apertures shaped in 0.05 mm × 0.05 mm square with round chamfers, and with clearance of 0.25 mm in between.<br>4. For the 12 pins in the very center:<br>The stencil aperture for each pad should be centered with area reduced to 50–70 %. The shape should be square. |

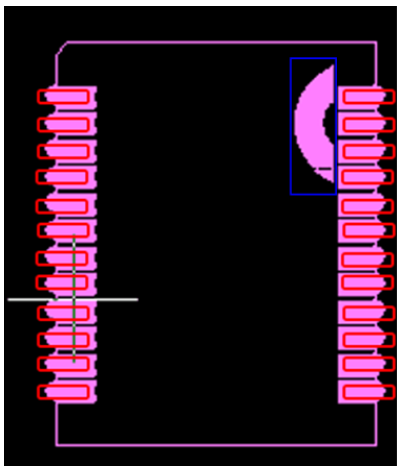
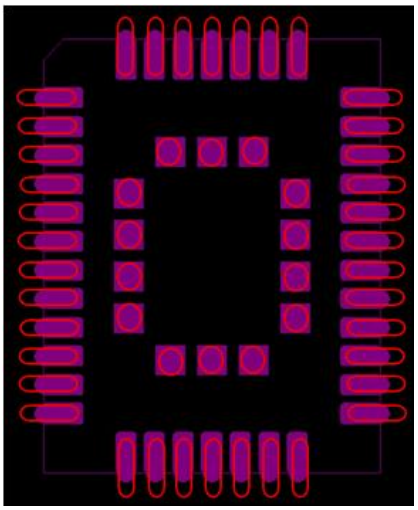
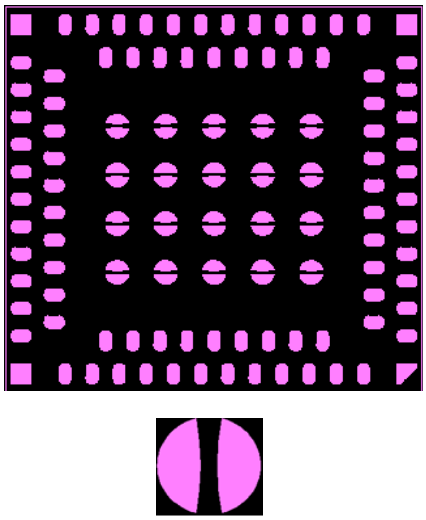
| | | |
|---|---|---|
| EC20/ EC20 R2.0/ EC21/ EC25/ UC200T/ EC20 R2.1 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.18–0.20 mm.<br>2. For pads on four sides:<br>The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.60 mm. The width should be reduced in a ratio of 1:0.8 or 1:0.9. And the shape should be rectangle with round chamfers.<br>3. For GND pads in the center:<br>Design the stencil aperture for each pad into four smaller 1.00 mm × 0.65 mm apertures shaped in 0.05 mm × 0.05 mm square with round chamfers, and with clearance of 0.25 mm in between.<br>4. Design apertures of round chamfers with diameter of 0.70 mm for the pads in the yellow frame.<br>5. For the 12 pins in the very center:<br>The stencil aperture for each pad should be centered with area reduced to 50–70 %. The shape should be square. |
| EG25-G |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.13–0.15 mm.<br>2. For pads on four sides:<br>The stencil aperture should be centered with area reduced to 65–85 % and the shape should be rectangle with round chamfers.<br>3. For pads in the center (in the red frame):<br>Design four square apertures centered in each pad of which the total area is 50–75 % of that of the pad.<br>4. The apertures for pads in the yellow frame should be centered with area reduced to 50–75 % and the shape should be rectangle with round chamfers.<br>5. The yellow block in the very center should be kept intact. |

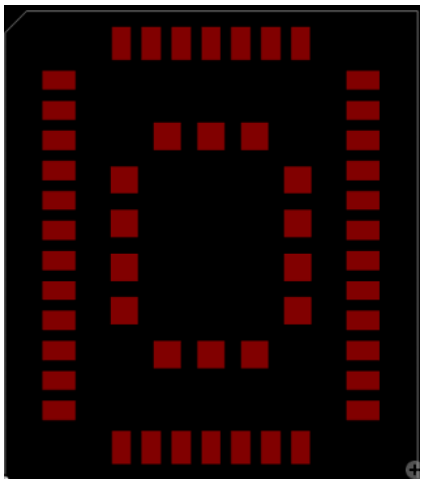
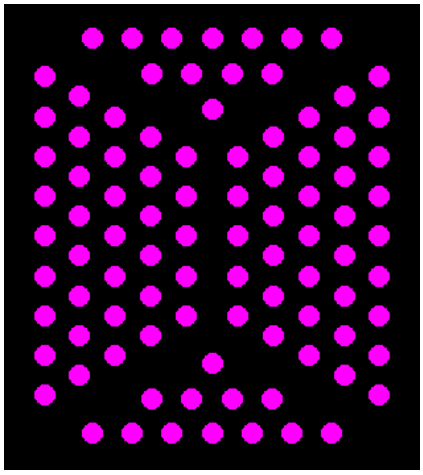
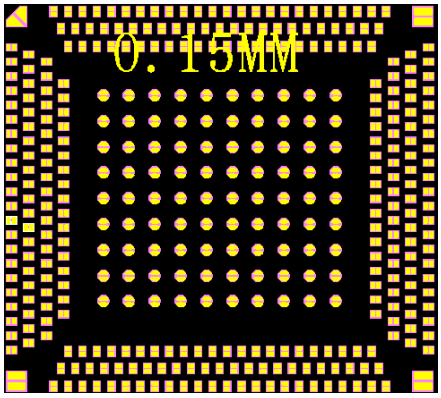
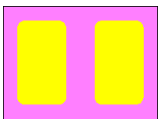
| | | |
|---|---|---|
| UG95/ UG96/ BG96/ BG95 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.13–0.15 mm.<br>2. For pads on four sides:<br>The aperture for each single pad should be centered with area reduced to 70–85 %. And the shape should be rectangle with round chamfers.<br>3. For square pads in the center:<br>The aperture for each single pad should be designed into round and centered with area reduced to 50–75 %. |
| EG91/ EG95 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.13–0.15 mm.<br>2. For pads on four sides:<br>The aperture for each single pad should be centered with area reduced to 75–85 %. And the shape should be rectangle with round chamfers.<br>3. For square pads in the center:<br>The aperture for each single pad should be designed into round and centered with area reduced to 50–75 %.<br>4. For pads at four corners:<br>The stencil aperture should be designed into diagonal-patterned lines with 50–60 % area of the corresponding pad. |
| EG06/ EG12/ EG18 |  <br>   | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.13–0.18 mm.<br>2. For pads on four sides:<br>The aperture for each single pad should be centered with area reduced to 75–90 %. And the shape should be rectangle with round chamfers as shown by the bottom left figure (yellow areas indicate the apertures for each pad, width of the bridge in between is 0.20 mm).<br>3. For pads at four corners:<br>The stencil aperture should be designed into diagonal-patterned lines with 50–60 % area of the corresponding pad.<br>4. For round pads in the center:<br>The aperture should be should be designed as shown by the bottom right figure (grey areas |

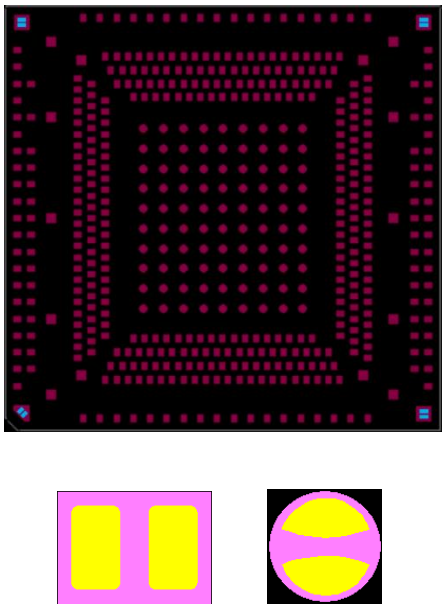
| | | |
|---|---|---|
| | | refer to the aperture shape, with 60–80 % area of the pad). |
| SC200R |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.18–0.20 mm.<br>2. For pads on four sides:<br>The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.60–0.80 mm, while width should be reduced in a ratio of 1:0.8 or 1:0.9. And should be shaped into rectangle with round chamfers.<br>3. For square pads in the center, the stencil aperture should be centered with area reduced to 40–75 % of that of the corresponding pad.<br>4. For the bigger square pads in the center (within the yellow frame), the aperture should be divided into four smaller squares and centered with area reduced to 40–60 % of that of the corresponding pad. |
| SC20 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.18–0.20 mm.<br>2. For pads on four sides:<br>The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.60–0.80 mm, while width should be reduced in a ratio of 1:0.8 or 1:0.9. And should be shaped into rectangle with round chamfers.<br>3. For square pads in the center, the stencil aperture area should be 50–75 % of that of the corresponding pad.<br>4. There is no need to design stencil aperture for the arc-shaped pad in the blue frames. |

| | | |
|---|---|---|
| SC600T/ SC600Y |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.18–0.20 mm.<br>2. For pads on four sides (in blue frames), the innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.60–0.80 mm while the width should be reduced in a ratio of 1:0.8 or 1:0.9. And the shape should be rectangle with round chamfers.<br>3. For square pads between the two red frames, the stencil aperture should be centered with area reduced to 75–90 % and designed with round chamfers.<br>4. For round pads in the center (in the green frame), the stencil aperture should be centered with area reduced to 50–60 %.<br>5. For pads at the four corners of the module, the stencil aperture should be designed into diagonal-patterned lines with 50–70 % area of the corresponding pad.<br>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 0.18–0.20 mm.<br>2. For pads on four sides:<br>The innermost edge of aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.60–0.80 mm, while the width should be reduced in a ratio of 1:0.8 or 1:0.9. And the shape should be rectangle with round chamfers.<br>3. For square pads in the center:<br>the stencil aperture should be centered with area reduced to 75–90 % and designed with round chamfers.<br>4. For round pads in the very center:<br>The aperture should be centered with area reduced to 50–60 %.<br>5. For pads at four corners:<br>the stencil aperture should be designed into diagonal-patterned lines with 50–60 % area of the corresponding pad. |

| | | |
|---|---|---|
| | | 6. For the pad marked with red arrow:<br>The aperture should be centered with area reduced to 75–90 %. |
| AG35 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.13–0.15 mm.<br>2. The stencil aperture area for the pads on four sides should be centered with 75–90 % area of corresponding pads, and should be designed with round chamfers as shown by the bottom left figure (yellow areas indicate the apertures for each pad, width of the bridge in between is 0.20 mm).<br>3. For pads at four corners (marked with red frames), the stencil aperture should be reduced inward in the shape of diagonal-patterned lines with 50–60 % area of the corresponding pad.<br>4. The stencil aperture for round GND pads in the yellow frame should be designed as shown by the bottom right figure (grey areas refer to the aperture shape, with 60–80 % area of the pad). |
| AG15 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.13–0.15 mm.<br>2. The stencil aperture area for the pads on four sides should be centered with 75–90 % area of corresponding pads, and should be designed with round chamfers as shown by the bottom left figure (yellow areas indicate the apertures for each pad, width of the bridge in between is 0.20 mm).<br>3. For pads at four corners, the aperture should be centered and designed into diagonal-patterned lines with 50–60 % area of the corresponding pad.<br>4. The aperture for round GND pads in the center should be designed as shown by the bottom right figure (grey areas refer to the aperture shape, with 60–80 % area of the pad). |

| | | |
|---|---|---|
| FC10 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.15–0.18 mm.<br>2. The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.40–0.60 mm, and the width should be reduced in a ratio of 1:0.8 or 1:0.9, and should be designed with round chamfers.<br>3. There is no need to design stencil apertures for the arc-shaped pad in the blue frame. |
| FC20 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.15–0.18 mm.<br>2. For pads on four sides:<br>The innermost edge of the aperture for each single pad should be shifted outward by 0.10–0.20 mm and the outermost edge should be shifted outward by 0.40–0.60 mm while the width should be reduced in a ratio of 1:0.8 or 1:0.9, and should be designed in rectangle with round chamfers.<br>3. For pads in the center:<br>The stencil aperture should be designed into round with area reduced to 50–75 %. |
| AF50T |  <br> | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.13–0.15 mm.<br>2. For pads on four sides:<br>The stencil aperture for each pad should be centered with area reduced in a ratio of 1:0.8 or 1:0.9 and the shape should be rectangle with round chamfers.<br>3. For round pads in the center, the aperture should be opened in the shape illustrated by the bottom figure (pink areas refer to the aperture area) and centered with area reduced to 50–75 %. |

| | | |
|---|---|---|
| AF20/BG600L-M3 |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.13–0.15 mm.<br>2. For pads on four sides:<br>The stencil aperture for each pad should be centered with area reduced to 75–90 % and the shape should be rectangle with round chamfers.<br>3. For square pads in the center:<br>The stencil aperture should be designed into round with area reduced to 50–70 %. |
| BG77/ BG770A-GL |  | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.10–0.13 mm.<br>2. For the round pads of the module, the stencil aperture should be centered with area reduced in a ratio of 1:0.8 or 1:0.9. |
| RG500Q/ AG520R/ EG512R-EA |  <br>   | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.15–0.18 mm.<br>2. For each pad on four sides: the aperture of each pad should be centered with area reduced in a ratio of 1:0.7 or 1:0.85, and should be designed with round chamfers as shown by the bottom left figure (yellow areas indicate the apertures for each pad, width of the bridge in between is 0.20 mm).<br>3. For round pads in the center, the aperture should be in the shape shown by the bottom right figure (yellow areas indicate the aperture area) and centered with area reduced to 50–75 %.<br>4. The stencil aperture for the four pads at four corners should be centered with area reduced in a ratio of 1:0.7 or 1:0.8 and a 0.3-mm-width bridge should be left in the middle. |

| | | |
|---|---|---|
| AG550Q |    | 1. The stencil thickness of the area for the module should be partly stepped-up to 0.15–0.18 mm.<br>2. For each pad on four sides: the aperture should be centered with area reduced in a ratio of 1:0.7 or 1:0.85, and should be designed with round chamfers as shown by the bottom left figure (yellow areas indicate the apertures for each pad, width of the bridge in between is 0.20 mm).<br>3. For round pads in the center, the aperture should be in the shape illustrated by the bottom right figure (pink areas refer to the aperture area) and centered with area reduced to 50–75 %.<br>4. The stencil aperture for the four pads at four corners (shown by the blue blocks) should be centered with area reduced in a ratio of 1:0.7 or 1:0.8 and a 0.3-mm-width bridge should be left in the middle. |

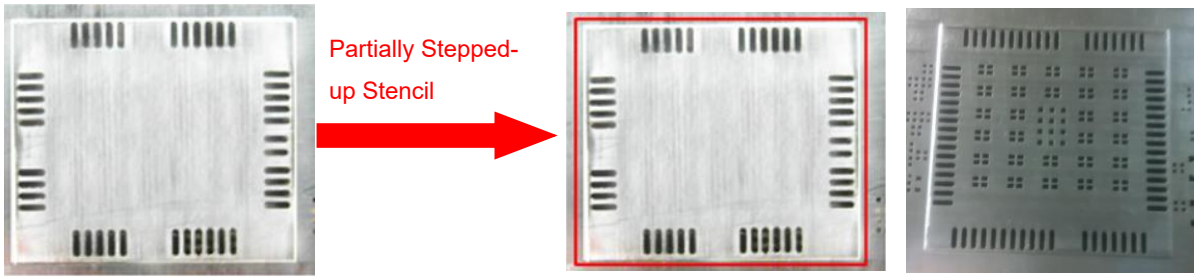



**Figure 3: Stepped-up Stencil Area**

**NOTES**

1. Quectel modules listed above may include multiple models. Please refer to the corresponding module specifications for details.
2. It is recommended that no component should be mounted in the area at the backside of the module on PCB, for the convenience of heating and repairing.
3. Please do not design any silkscreen in the area where the module will be mounted to avoid the height that may influence the soldering.
4. Stencil apertures for the components pads within 1.0 mm from the partially thickened area of the stencil should be reduced by 10 % to 30 % compared with regular apertures, since the thickening will increase the volume of solder paste. When there is a need to step-up the stencil, all 0201 components (with package measuring 0.024 in by 0.012 in) and components with 0.40 mm or 0.50

mm pin-pitch should be kept over 5.0 mm away from the stepped-up area to avoid solder bridging and short circuit that is caused by thicker solder paste.
5. Please optimize the stencil design depending on the actual situation.
6. 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.

## 4.5. Mounting Process

### 4.5.1. 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.

### 4.5.2. 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 pins 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**

## 4.6. Reflow Soldering

It is recommended to test the temperature based on real modules. Thermocouple temperature test points should be applied to both pins on four sides and that at the bottom, to guarantee required soldering temperature. Please refer to the recommended ramp-soak-spike thermal profile for lead-free reflow soldering in the following figure.

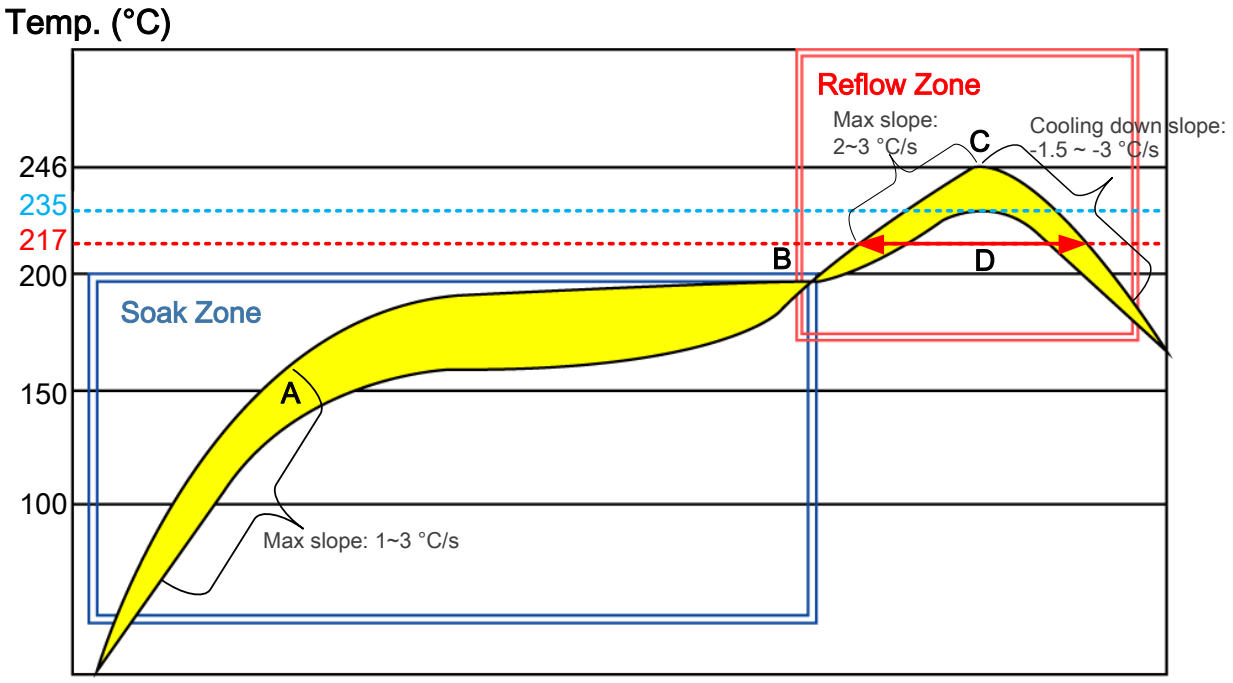



**Figure 6: Ramp-soak-spike Reflow Profile**

**Table 2: Recommended Thermal Profile Parameters**

| Factor | Recommendation |
|---|---|
| **Soak Zone** | |
| Max slope | 1–3 °C/s |
| Soak time (between A and B: 150 °C and 200 °C) | 70–120 s |
| **Reflow Zone** | |
| Max slope | 2–3 °C/s |
| Reflow time (D: over 217°C) | 40–70 s |
| Max temperature | 235 °C to 246 °C |

| | |
|---|---|
| Cooling down slope | -1.5 to -3 °C/s |
| **Reflow Cycle** | |
| Max reflow cycle | 1 |

The following aspects should be noticed:

1. Temperature:

   The real soldering temperature is affected by factors such as carrier, solder paste, size and thickness of PCB substrate, thermal durability of components, and the PCB design, etc. If the recommended specifications cannot be reached, please contact Quectel technical support, or the module may be damaged during reflow soldering.

2. Carrier:

   For motherboard with thickness of less than 1.0 mm, it is recommended to use high Tg material or to use board carrier during reflow soldering, to prevent the motherboard from distorting due to heat.

   For 5G modules or modules of 40.0 mm × 40.0 mm or larger size, it is recommended to use carriers, to reduce the defect (such as pseudo soldering or cold joint) rate caused by thermal imbalance resulting from Tg value difference between the module the motherboard.

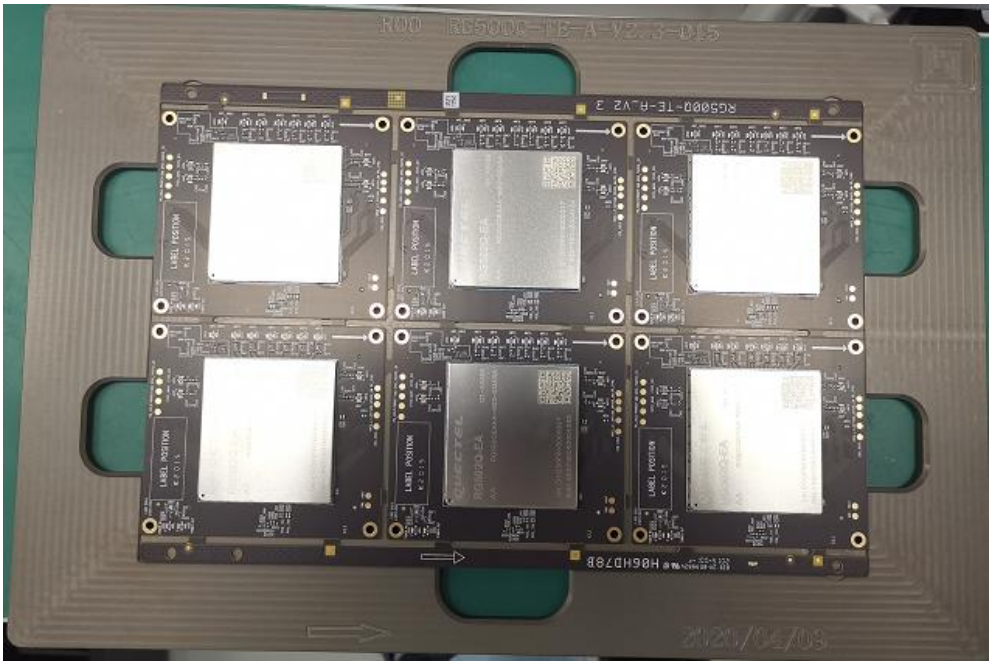


**Figure 7: Carrier for 5G Modules**

3. Cooling:

   By controlling the cooling rate can the soldering defect (fragile solder joint) and negative effects on mechanical stress be avoided. Proper cooling rate will help to make the solder joint lustrous with lower solder fillet. The proper cooling rate should be -3 °C/s.

4. Visual Inspection:

   Please implement inspections on the soldering quality with X-ray or other optical methods such as using a magnifier after reflow soldering. For relevant standard, please refer to *IPC-A-610F*, in which 3 classes are proposed. Please perform the inspection following different classes according to practical applications and scenarios. For automotive applications, Class 3 is recommended.

**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 & Resoldering

## 5.1. Matters Needing Attention

Please pay attention to the following factors for heating and desoldering:

1. BGA soldering should be inspected through X-ray in advance to avoid bridging and displacement caused by improper temperature or method. It is recommended to contact technical support of the supplier in advance.

2. The motherboard should be baked at 120 °C for 8 hours to prevent damp PCBs from being damaged after direct heating. It is important to ensure that all components on the motherboard stand baking at 120 °C.

3. Do not disassemble and desolder the module itself, or the warranty will terminate immediately.

**NOTE**

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

## 5.2. Desoldering

Please use a heat gun to heat the solder joints from both sides of the motherboard to remove the module. ESD protection must be implemented during the desoldering.

- The temperature of the heat gun should be about 320 °C to 350 °C to release enough heat. The wind speed and distance should be adjusted according to actual situation.
- If the motherboard has been exposed to the air for exceeding 48 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 2.0–3.5 cm.

- Move the nozzle along the edge of the module at a constant speed. When all of the solder joints are melted, quickly take off the module along the diagonal direction with tweezers and lay it on a flat cooling platform for cooling.

For the module larger than 30.0 mm × 30.0 mm, a BGA rework station or heat gun (with larger outlet nozzle) can be used to desolder components. To prevent separation between pad and circuit as well as PCB blistering caused by long-time 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.


**Figure 8: Resoldering Tools**

For single-sided motherboard with no component mounted at the bottom, a preheating station can be used with a heat gun. Heating from both sides helps to melt the solder paste fast and to avoid damage caused by layer separation or blistering.

One of the preheating stations is shown as below. The temperature for reference is 265 °C to 280 °C (provided on the basis of factory experience and the temperature should be set according to real situation)


| | |
|---|---|
| Model | ANSAI 946C |
| Input Voltage | 220V±10% AC |
| Heating Area | 200mm*200mm |
| Power | 800W |
| Temperature | 50℃~350℃ |
| Stability | ±1.5~℃ |
| Size | 218mm*152mm*218mm |
| Weight | 5.4KG |

**Figure 9: Preheating Station**

## 5.3. Recommended Resoldering Procedure

1. Remove superfluous solder paste on the pads of motherboard with electric soldering iron and keep the pads flat.

2. Pre-apply solder paste on the pads with electric soldering iron, keeping the solder paste moderate in amount and equally distributed.

3. Equally distribute moderate solder flux for the pads.

4. Mount the module precisely on the motherboard (pay attention to the pins and direction of the module). Then, heat the area where the module is mounted on the motherboard from both top and bottom side with heat gun kit or BGA rework station. The resoldering will be finished after solder paste on all pads are melted.

5. After the soldering, cool the motherboard fast with fan.

6. After the temperature returns to normal, inspect the soldering quality for LCC pins on four sides of the module to ensure that there is no defect such as bridging or insufficient solder paste.


**Figure 10: Quality Inspection for LCC Pins**

7. Finally, inspect the soldering quality of BGA components and pins of the module at the bottom of modules with X-ray.

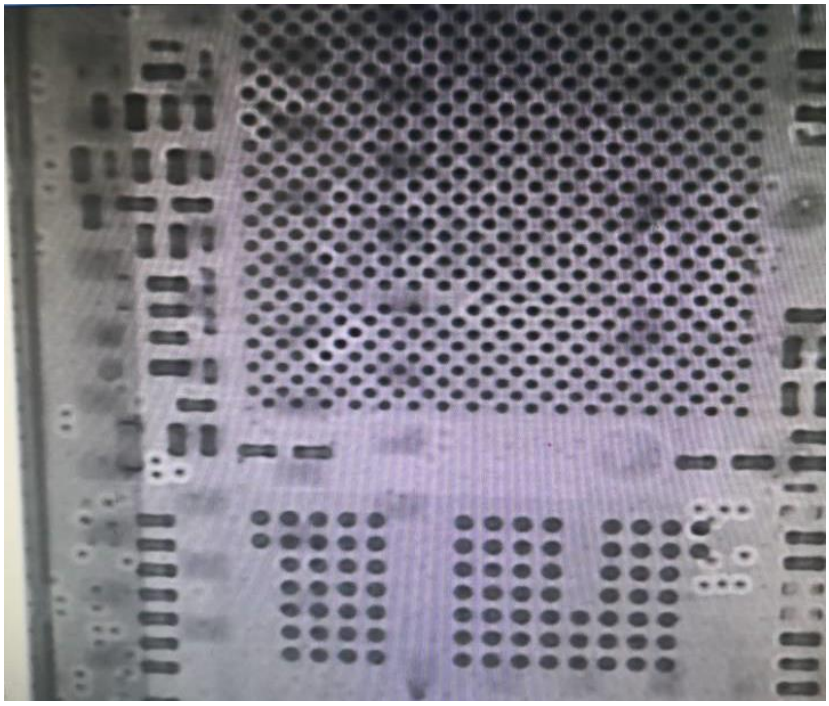


**Figure 11: Quality Inspection for LGA Pins or BGA Components**

# 6 ESD Protection

Static electricity may cause intermittent or permanent circuit damage, which is extremely harmful to electronic devices. Most of the damages, as analyzed, are obvious ESD or EOS burning, as shown in the following figure:

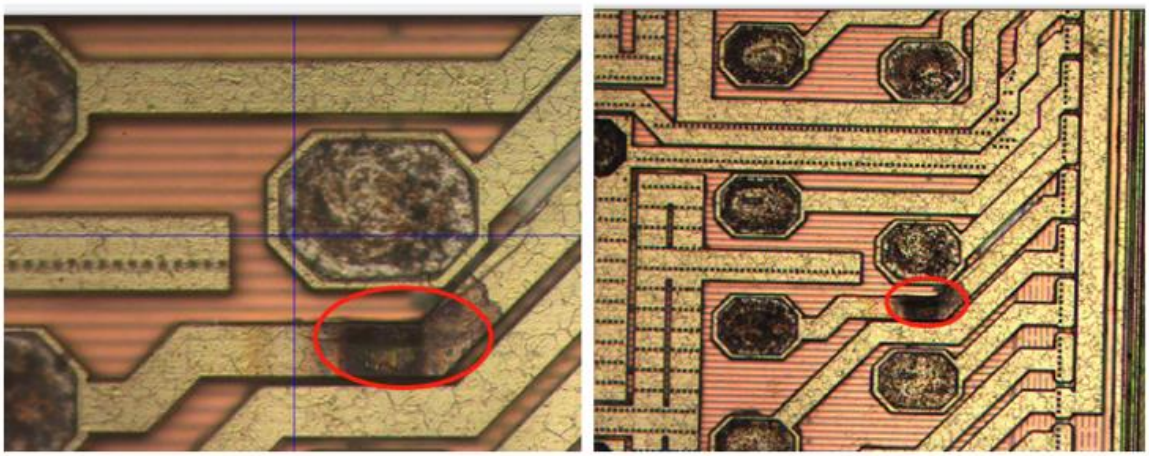


**Figure 12: ESD or EOS Burning**

Thus, ESD protection is vital to the modules. Please strictly observe the following conditions during production and transportation:

- Avoid touching the module with bare hand, especially for the area of module pins.
- Ground the housing of related equipment such as chip mounters, work bench, soldering equipment.
- ESD protection methods such as anti-static wrist band with grounding wire (cordless anti-static wrist strap are not allowed, while anti-static gloves are recommended) should be adopted for operators.
- Qualified anti-static materials must be used for packaging and PCBA boards.

# 7 Appendix Reference

**Table 3: Terms and Abbreviations**

| Abbreviation | Description |
|---|---|
| BGA | Ball Grid Array |
| ENEG | Electroless Nickel Electroless Gold |
| ENIG | Electroless Nickel Immersion Gold |
| EOS | Electrical OverStress |
| ESD | Electro-Static Discharge |
| LCC | Leadless Chip Carriers |
| LGA | Land Grid Array |
| MSL | Moisture Sensitivity Level |
| PCB | Printed Circuit Board |
| SMD | Surface Mount Device |
| SMT | Surface Mount Technology |
| Tg | Glass Transition Temperature |