

Module Secondary SMT Application Note

LCC/LGA Module Series

Rev. Module_Secondary_SMT_Application_Note_V2.7

Date: 2020-02-25

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, China 200233

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.

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. 2020. All rights reserved.

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

4.2 and Chapter 4.3.

3. Updated the stencil design requirements in Chapter 4.4 and added requirements for: M08-R/L89/UC200T/BG95/BC92/EG18/EC100Y/BG600L-M3.
 4. Updated specifications of reflow soldering and added matters needing attention in Chapter 4.6.
 5. Updated the sub-section and description of Chapter 5.
-

Content

About the Document.....	2
Content	4
Table Index.....	5
Figure Index	6
1 Introduction	7
2 Information about Modules.....	8
2.1. Surface-Mount Packaging Type.....	8
2.2. Packing Methods.....	8
3 Requirements for SMT.....	9
3.1. Chip Mounter.....	9
3.2. Soldering Requirements	9
4 Attentions for Manufacturing	10
4.1. MSL and Moisture-proof Requirement.....	10
4.2. Storage.....	11
4.3. Floor Life	11
4.4. Stencil Design Requirements.....	11
4.5. Mounting Process	22
4.5.1. Load Materials.....	22
4.5.2. Automatic Placement	22
4.6. Reflow Soldering	24
5 Desoldering & Resoldering.....	26
5.1. Matters Needing Attention.....	26
5.2. Desoldering	26
5.3. Recommended Resoldering Procedure.....	28
6 Appendix Reference	30

Table Index

Table 1: Stencil Design Requirements	12
Table 2: Recommended Thermal Profile Parameters.....	24
Table 3: Terms and Abbreviations	30

Figure Index

Figure 1: Vacuum Tape & Reel Packing	8
Figure 2: Humidity Indicator Card	10
Figure 3: Step-up Stencil Area	22
Figure 4: Automatic Placement	23
Figure 5: First Pin and Mounted Picture	23
Figure 6: Ramp-soak-spike Reflow Profile	24
Figure 7: Resoldering Tools	27
Figure 8: Preheating Station.....	27
Figure 9: Quality Inspection for LCC Pins.....	28
Figure 10: Quality Inspection for LGA Pins or BGA Components	29

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 and/or LGA package.

2.2. Packing Methods

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

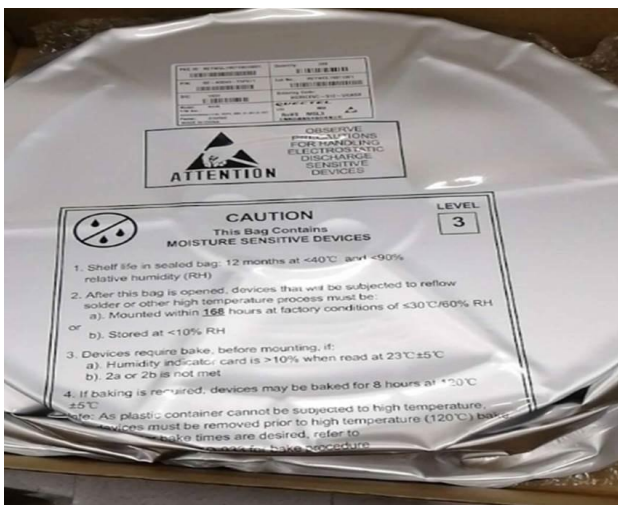


Figure 1: Vacuum Tape & Reel Packing

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.

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 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~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 1.0mm, 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 defined 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.

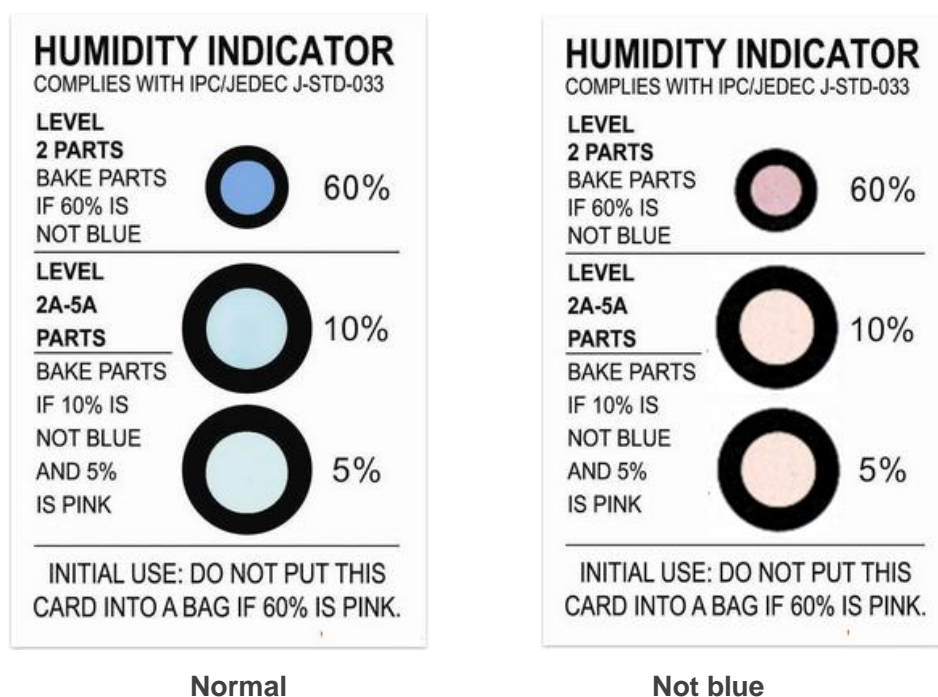


Figure 2: Humidity Indicator Card

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

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 24 hours. In a plant where the temperature is 23°C±5°C and relative humidity is below 60%, the module must be processed in solder reflow or other high-temperature operations within 24 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. 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°C±5°C to avoid blistering, crack and layer separation.
2. 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.

4.4. Stencil Design Requirements

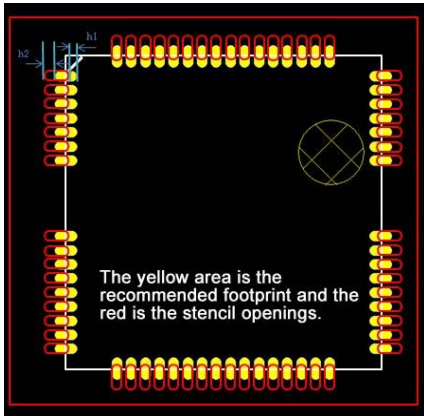
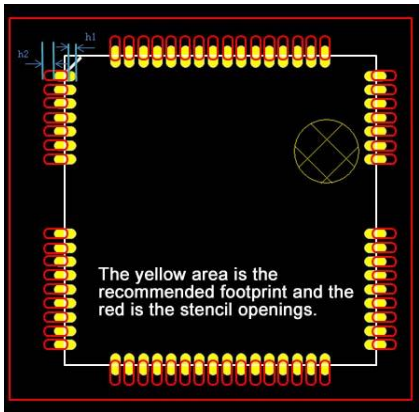
To ensure the solder paste is enough and soldering joints are reliable, the stencil should be partly stepped-up on the top surface. And the stencil aperture for each single pad cannot be larger than 3.0mm×4.0mm and the exceeded part should be divided into smaller apertures with size less than 2.0mm×2.0mm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

NOTES

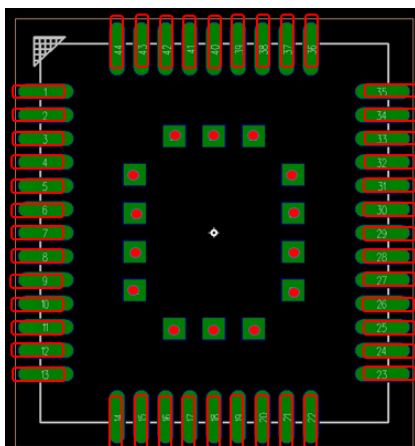
1. It is recommended that no component should be mounted in the area at the backside of the module on PCB, for the convenience of heating and repairing.
2. Please do not design any silkscreen in the area where the module will be mounted to avoid the height that may influence the soldering.
3. When there is a need to step-up the stencil, all 0201, 0.4mm-pitch and 0.5mm-pitch components should be kept over 5.0mm away from the stepped-up area to avoid solder bridging and short circuit that is caused by thicker solder paste.

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

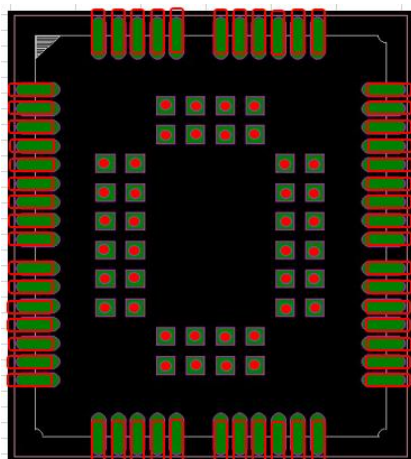
Module	Diagram for Stencil Designs	Requirement Description
M08-R/M10/M12/ M72/M80/M85/ M95/M66/ M66-DS/GC10		<ol style="list-style-type: none"> 1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm. 2. The innermost edge of the aperture for each single pad should be shifted outward by 0.10mm (refer to h1 in the left figure) and the outermost edge should be shifted outward by 0.40mm~0.60mm (refer to h2 in the left figure). 3. The width should be reduced in a ratio of 1:0.9.
L10/L16/L20/L26/ L30/L50/L70/L76/ L80/L70-R/ L70-RL/L76-L/ L76B/L80-R/L86/ L96/L89		<ol style="list-style-type: none"> 1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm. 2. The innermost edge of the aperture for each single pad should be shifted outward by 0.10mm (refer to h1 in the left figure) and the outermost edge should be shifted outward by 0.40mm~0.50mm (refer to h2 in the left figure). 3. The width should be reduced in a ratio of 1:0.9.

BC66/BC68



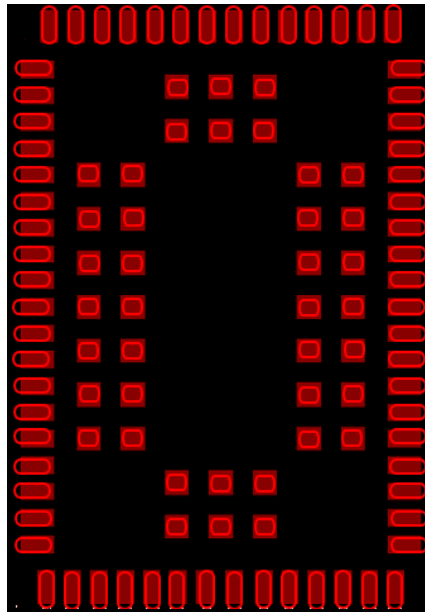
1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm.
2. For pads on four sides:
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.40mm~0.60mm, and the width should 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 65% of that of the corresponding pad and the shape should be round.

BC95-G/BC92



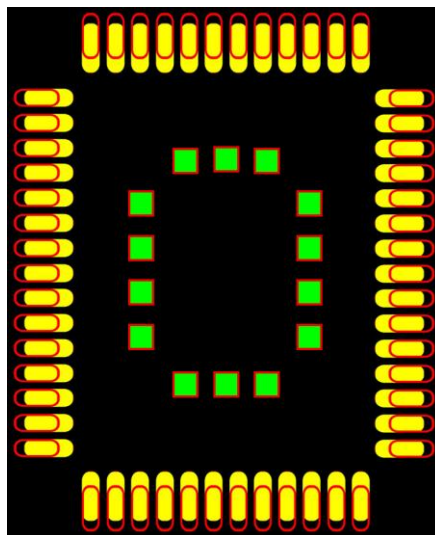
1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm.
2. For pads on four sides:
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.40mm~0.60mm, and the width should 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 65% of that of the corresponding pad and the shape should be round.

M89



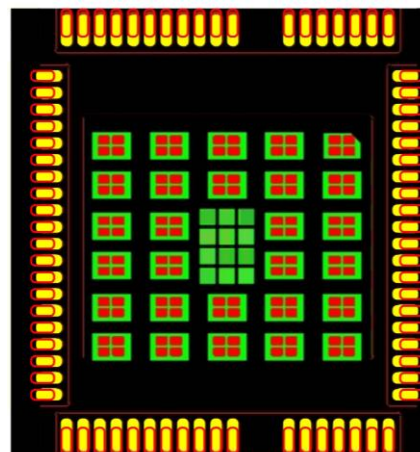
1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm.
2. For pads on four sides:
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and outermost edge should be shifted outward by 0.40mm~0.60mm, and the width should be 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



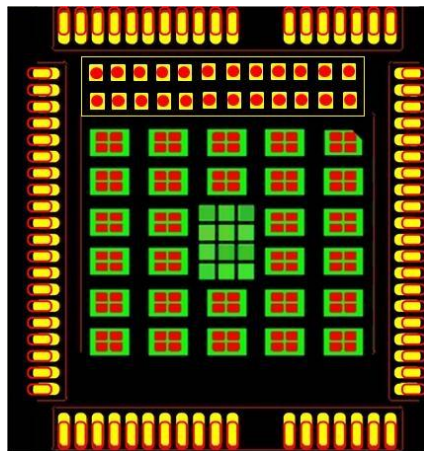
1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm.
2. For pads on four sides:
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and outermost edge should be shifted outward by 0.40mm~0.60mm, while the width should 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 65%. The shape should be square.

UC20/UC15



1. The stencil thickness of the area for the module should be partly stepped-up to 0.18mm~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 0.60mm. 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

EC20/EC20 R2.0/
EC21/EC25/
UC200T/
EC20 R2.1



into 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 0.18mm~0.20mm.

2. For pads on four sides:

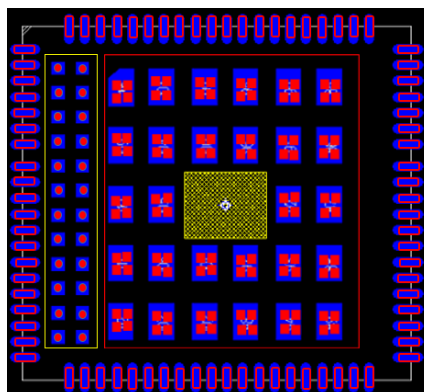
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.60mm. The width should be reduced in a ratio of 1:0.85. And the shape should be rectangle with round chamfers.

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 apertures of round chamfers with diameter of 0.75mm 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 0.13mm~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):

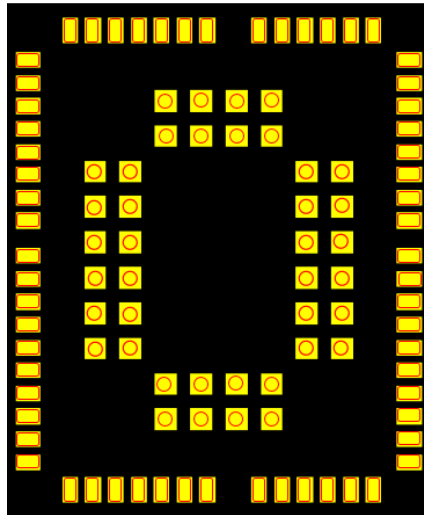
Design four square apertures centered in each pad of which the total area is 70% of that of the pad.

4. The apertures for pads in the yellow box should be centered with area reduced to

75% and the shape should be rectangle with round chamfers.

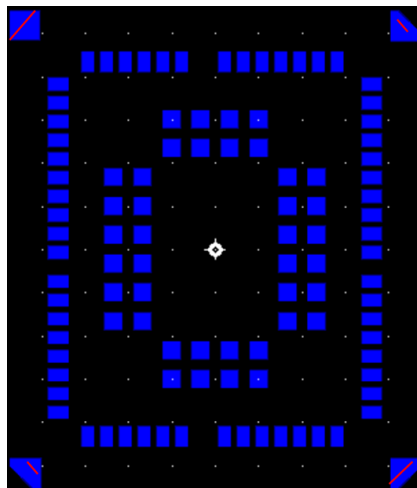
5. The yellow block in the very center should be kept intact.

UG95/UG96/
BG96/BG95



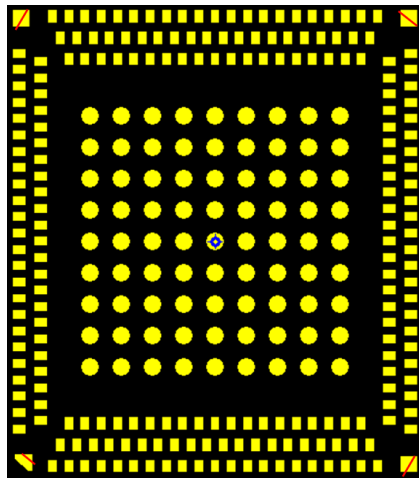
1. The stencil thickness of the area for the module should be partly stepped-up to 0.13mm~0.15mm.
2. For pads on four sides:
The aperture for each single pad should be centered with area reduced to 75%~85%. And the shape should be rectangle with round chamfers.
3. For square pads in the center:
The aperture for each single pad should be designed into round and centered with area reduced to 65%.

EG91/EG95



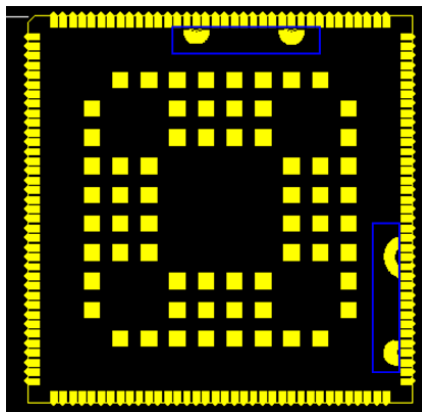
1. The stencil thickness of the area for the module should be partly stepped-up to 0.13mm~0.15mm.
2. For pads on four sides:
The aperture for each single pad should be centered with area reduced to 75%~85%. And the shape should be rectangle with round chamfers.
3. For square pads in the center:
The aperture for each single pad should be designed into round and centered with area reduced to 65%.
4. For pads at four corners:
The stencil aperture should be designed into diagonal-patterned lines with 60% area of the corresponding pad.

EG06/EG12/
EG18



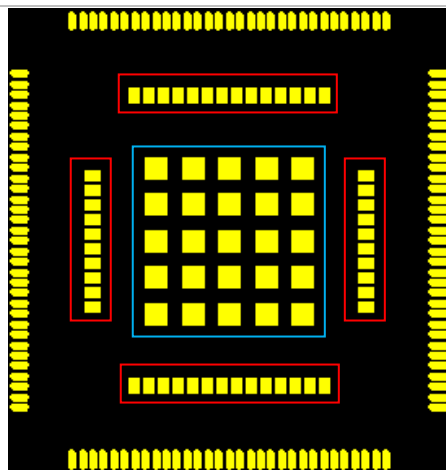
1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm.
2. For pads on four sides:
The aperture for each single pad should be centered with area reduced to 75%~85%. And the shape should be rectangle with round chamfers.
3. For round pads in the center:
The aperture should be centered with area reduced to 65%. And the shape should be round.
4. For pads at four corners:
The stencil aperture should be designed into diagonal-patterned lines with 60% area of the corresponding pad.

SC20



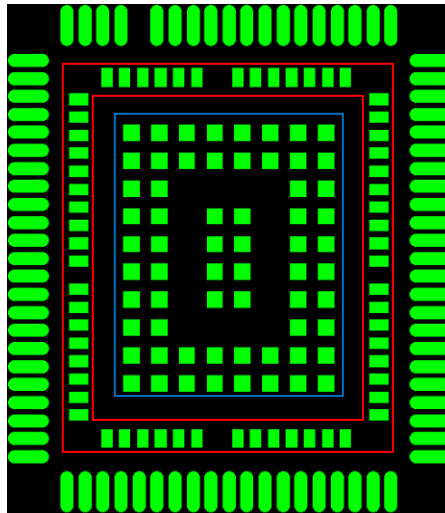
1. The stencil thickness of the area for the module should be partly stepped-up to 0.18mm~0.20mm.
2. For pads on four sides:
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.60mm~0.80mm, while width should 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 65% of that of the corresponding pad.
4. There is no need to design stencil aperture for the arc-shaped pad in blue box.

SC300P



1. The stencil thickness of the area for the module should be partly stepped-up to 0.18mm~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 0.80mm, while width should be reduced in a ratio of 1:0.85. And should be shaped into rectangle with round chamfers.
3. For the pads marked with red frame, the

EC100Y



stencil aperture area should be 85% of that of the corresponding pad.

4. For the pads marked with blue frame, the stencil aperture area should be 65% of that of the corresponding pad.

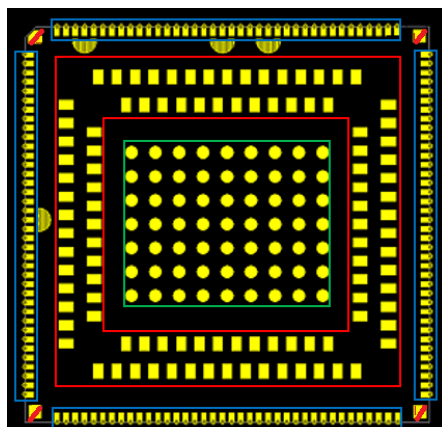
1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm.

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 0.60mm~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 the pads marked with red frame, the stencil aperture area should be 85% of that of the corresponding pad and the shape should be rectangle with round chamfers.
4. For the pads marked with blue frame, the stencil aperture area should be 65% of that of the corresponding pad and the shape should be rectangle with round chamfers.

SC600T/SC600Y



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

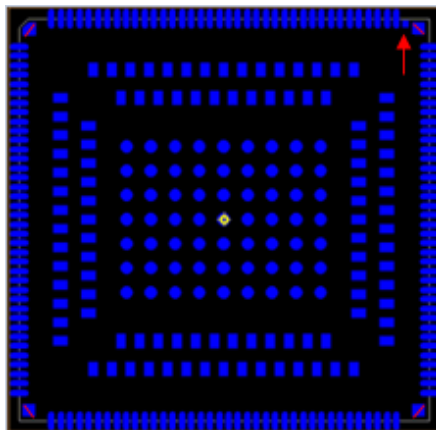
2. For pads on four sides (in blue boxes), the innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.80mm while the width should be reduced in a ratio of 1:0.90. And the shape should be rectangle with round chamfers.

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 60%.

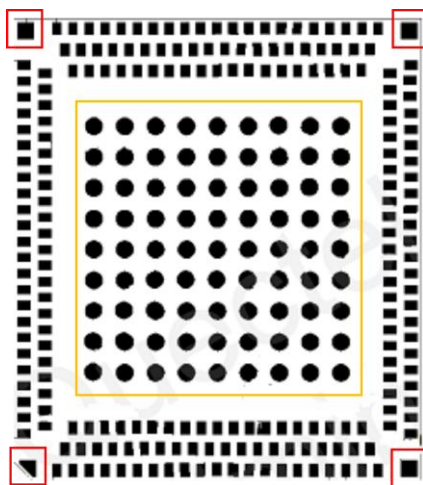
5. For pads at the four corners of the module, the stencil aperture should be designed

SC66

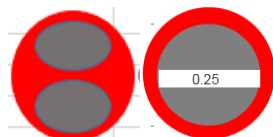


- into diagonal-patterned lines with 70% area of the corresponding pad.
 6. There is no need to design stencil aperture for the four arc-shaped pads.
1. The stencil thickness of the area for the module should be partly stepped-up to 0.18mm~0.20mm.
 2. For pads on four sides:
The innermost edge of aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.60mm~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 60%.
 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 85%.

AG35

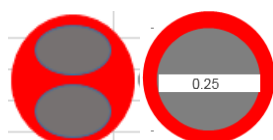
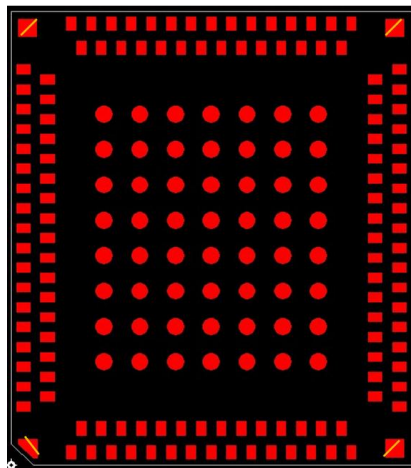


1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm.
2. The stencil aperture area for the pads on four sides should be centered with 75%~85% area of corresponding pads, and should be designed with round chamfers.
3. For pads at four corners (marked with red 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



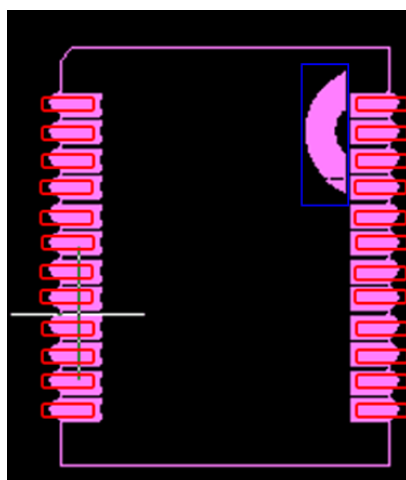
the yellow frame should be designed into either of the two references below the figure of the module (grey areas refer to the aperture shape, with 60%~70% area of the pad).

AG15



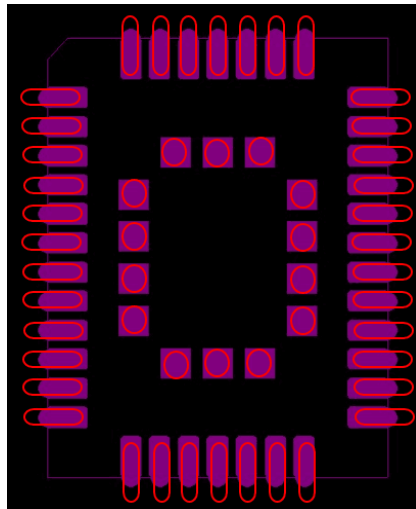
1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm.
2. The stencil aperture area for the pads on four sides should be centered with area reduced to 75%~85%, and should be designed with round chamfers.
3. For pads at four corners, the aperture should be centered 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 either of the two references below the figure of the module (grey areas refer to the aperture shape, with 60%~70% area of the pad).

FC10



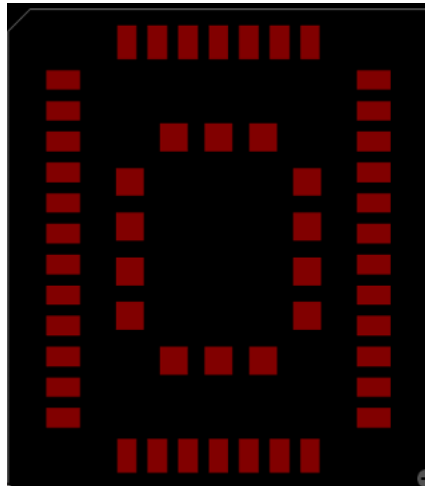
1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm.
2. The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.40mm~0.60mm, and the width should be reduced in a ratio 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



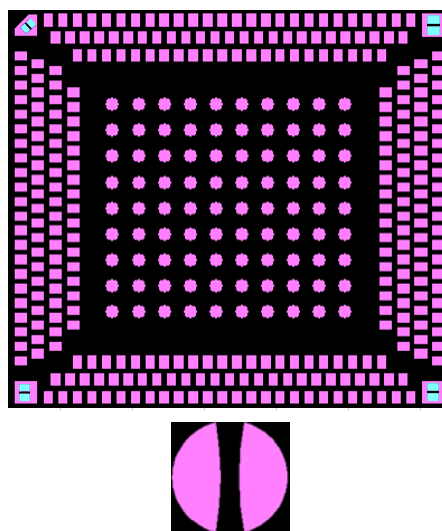
1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm.
2. For pads on four sides:
The innermost edge of the aperture for each single pad should be shifted outward by 0.20mm and the outermost edge should be shifted outward by 0.40mm~0.60mm while the width should 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/BG600L-M3



1. The stencil thickness of the area for the module should be partly stepped-up to 0.13mm~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 65%.

RG500Q/
AG520R



1. The stencil thickness of the area for the module should be partly stepped-up to 0.15mm~0.18mm.
2. The stencil aperture for each pad on four sides, the aperture should be centered with area reduced in a ratio of 1:0.90.
3. For round pads in the center, the aperture should be round and centered with area reduced to 65%.
4. The stencil aperture for the four pads at four corners should be centered with area reduced in a ratio of 1:0.70.

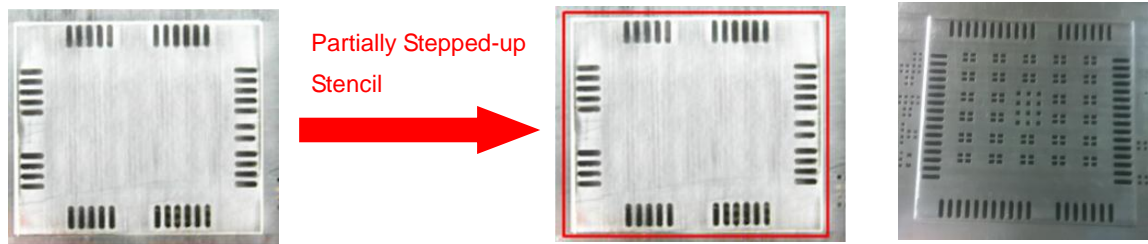


Figure 3: Step-up Stencil Area

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 5.0mm space in between, otherwise there may be a risk of solder bridging that will cause short circuit of the component.
3. Please 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.

4.5. 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



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.

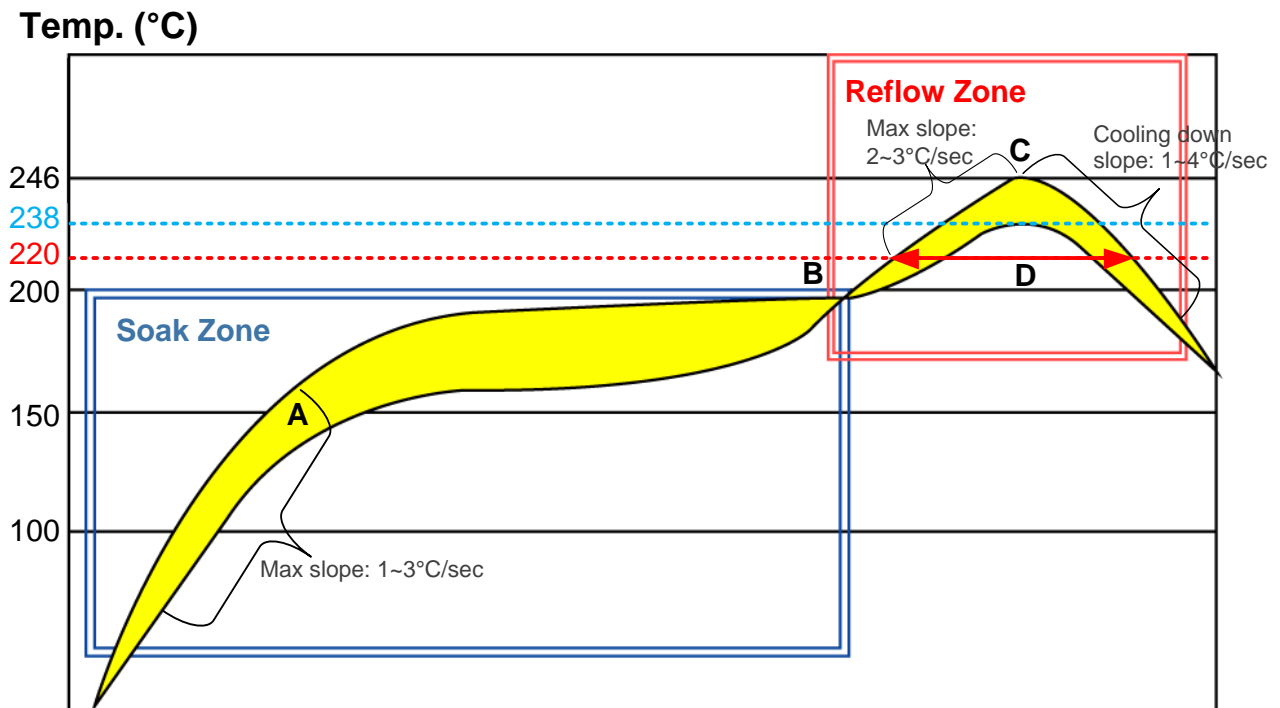


Figure 6: Ramp-soak-spike Reflow Profile

Table 2: Recommended Thermal Profile Parameters

Factor	Recommendation
Soak Zone	
Max slope	1°C/sec ~ 3°C/sec
Soak time (between A and B: 150°C and 200°C)	70 sec ~ 120 sec
Reflow Zone	
Max slope	2°C/sec ~ 3°C/sec
Reflow time (D: over 220°C)	45 sec ~ 70 sec
Max temperature	238°C ~ 246°C

Cooling down slope	-1°C/sec ~ -4°C/sec
--------------------	---------------------

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

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/sec.

4. Visual Inspection:

Please implement inspections on the soldering quality with X-ray or other optical methods after reflow soldering. For relevant standard, please refer to *IPC-A-610F*.

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~350°C in order 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 2.0cm to 3.5cm.

- Move the nozzle along the edge of the module at a constant speed. When all of the solder joints are melted, 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 30.0mm×30.0mm, 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 7: 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~280°C (provided on the basis of factory experience and the temperature should be set according to real situation)

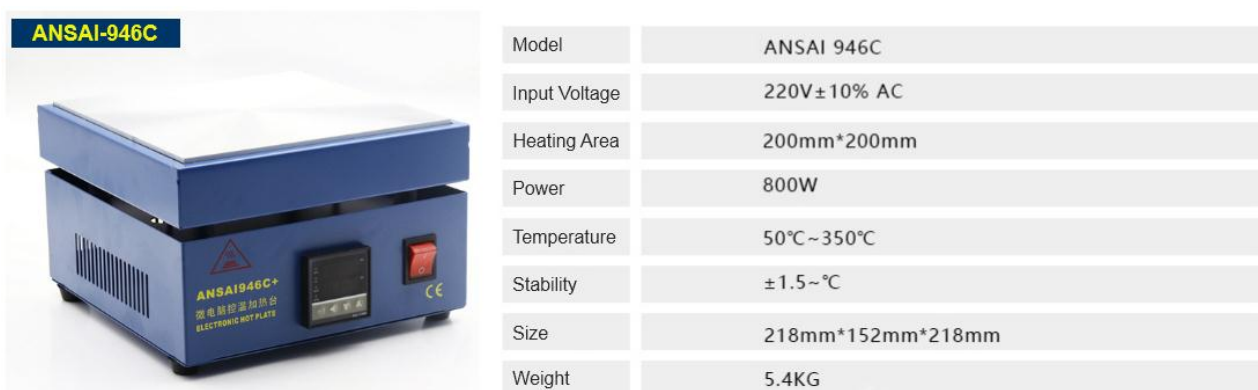


Figure 8: Preheating Station

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 9: 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.

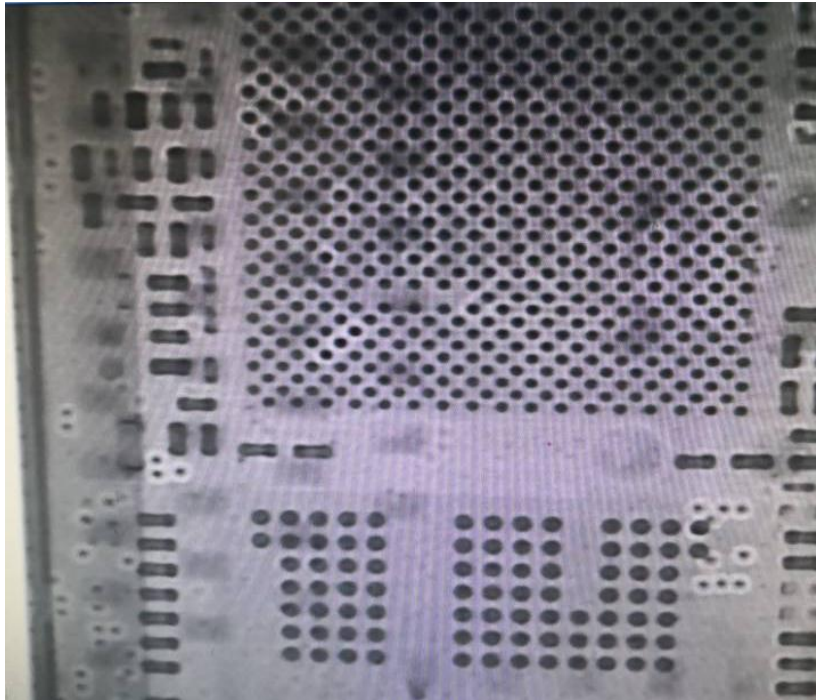


Figure 10: Quality Inspection for LGA Pins or BGA Components

6 Appendix Reference

Table 3: Terms and Abbreviations

Abbreviation	Description
BGA	Ball Grid Array
ENEG	Electroless Nickel Electroless Gold
ENIG	Electroless Nickel Immersion Gold
LCC	Leadless Chip Carriers
LGA	Land Grid Array
MSL	Moisture Sensitivity Level
PCB	Printed Circuit Board
SMD	Surface Mount Device
SMT	Surface Mount Technology