AG35 Secondary SMT Guidelines

LTE Module Series

Rev. AG35_Secondary_SMT_Guidelines_V1.0

Date: 2018-06-02

Status: Released

www.quectel.com

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

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

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

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

GENERAL NOTES

COPYRIGHT

About the Document

History

Revision	Date	Author	Description
1.0	2018-06-02	Rowan WANG/ Alain HUANG	Initial

Contents

Figure Index

1 Introduction

This document presents guidelines of stencil design and rework for Quectel AG35 module during secondary SMT process.

2 Stencil Design Guidelines

This chapter describes stencil design requirements during PCB mounting.

2.1. Stencil Design

The thickness of the stencil is suggested to be stepped-up to 0.15mm~0.18mm. The following figure shows the recommended stencil design. And it is highly recommended to monitor the solder paste height, registration and proper placement during the squeegee process.

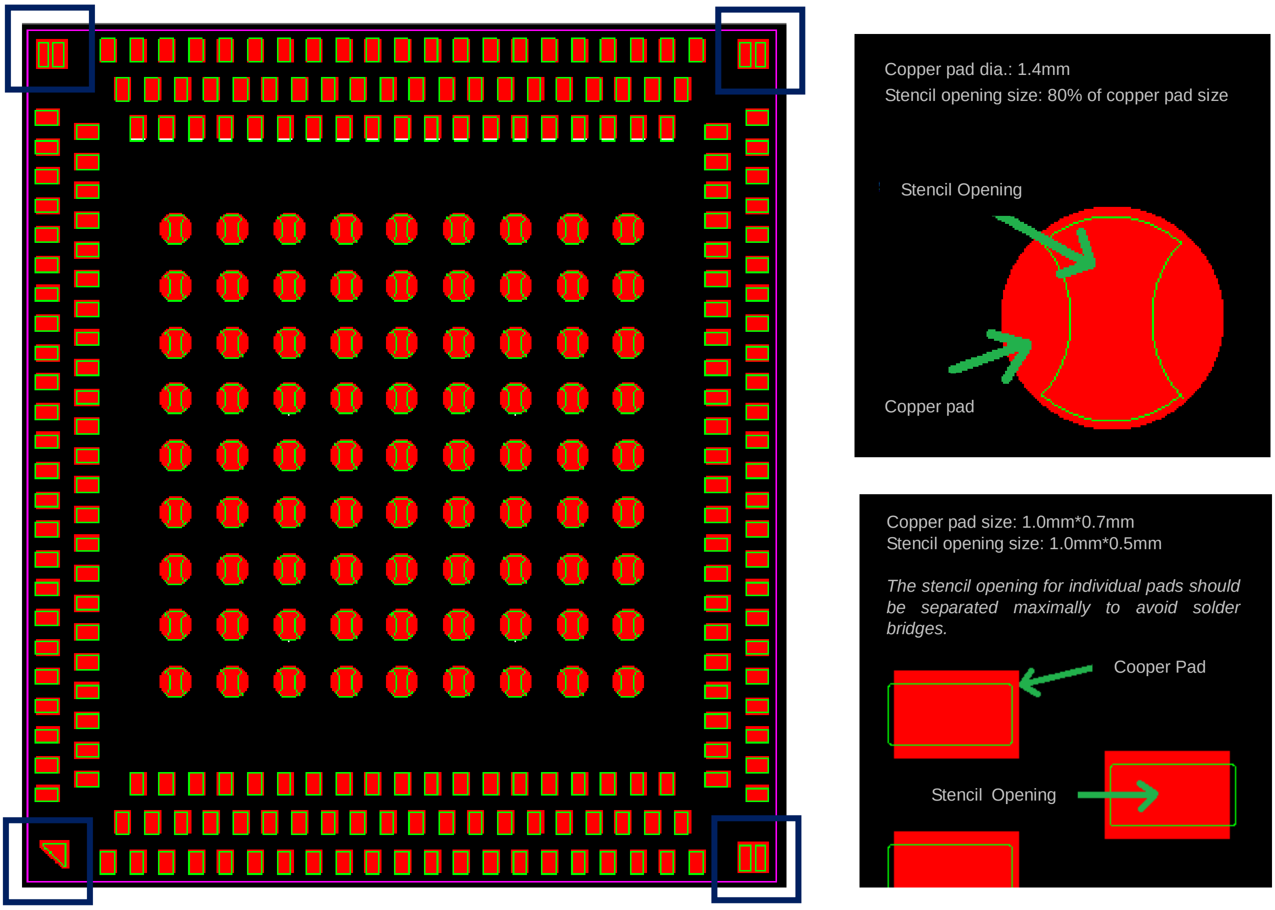

Figure 1: Recommended Stencil Design

For the pads at the four corners of the module (marked with dark blue boxes), the stencil opening sizes should be shrunken inward into about 60% of the size of corresponding pad.

3 Methods to Reduce Solder Joint Voiding

The chapter introduces several methods to reduce solder joint voiding during the SMT reflow process so as to achieve better soldering quality.

3.1. Selection of Specialized Solder Paste

It is recommended to use Indium 10.1HF solder paste, which can reduce the void rate.

3.2. Use of Solder Performs

A solder perform SAC305 (0201) is recommended to be added at any three corners of the module before SMT reflow process. Thus the lifted module leaves much room for outgassing, and the total solder joint voids can be reduced to 10% or less. Meanwhile, these solder performs will be melted under 150°C ~200°C without affecting soldering, but the stencil thickness should be matched to the height of the solder perform, otherwise the module will be a misaligned component.

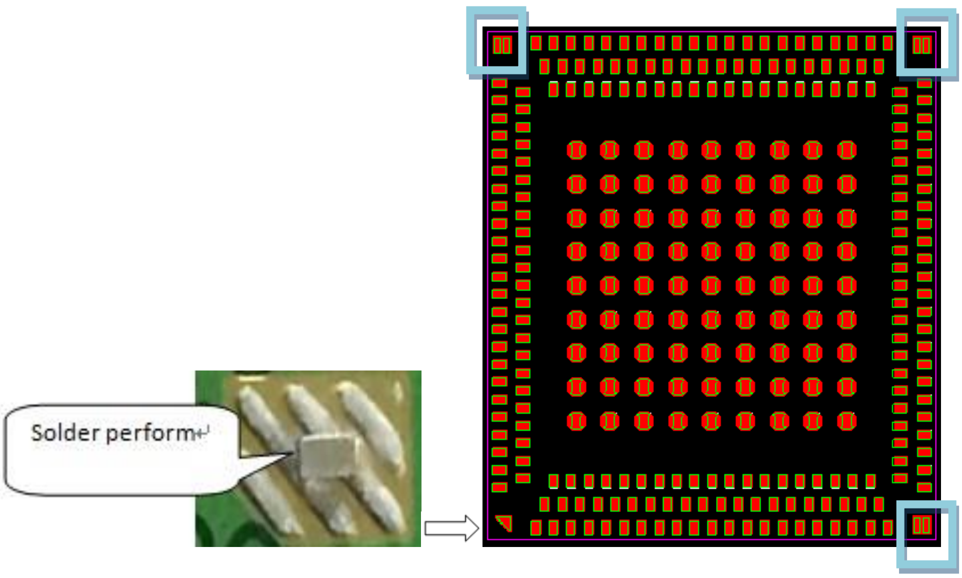

Figure 2: Placement of Solder Perform

3.3. Vacuum Reflow Oven for Voidless Soldering

Figure 3: Vacuum Reflow Oven for Voidless Soldering Results

3.3.1. Principle of Vacuum Reflow Soldering

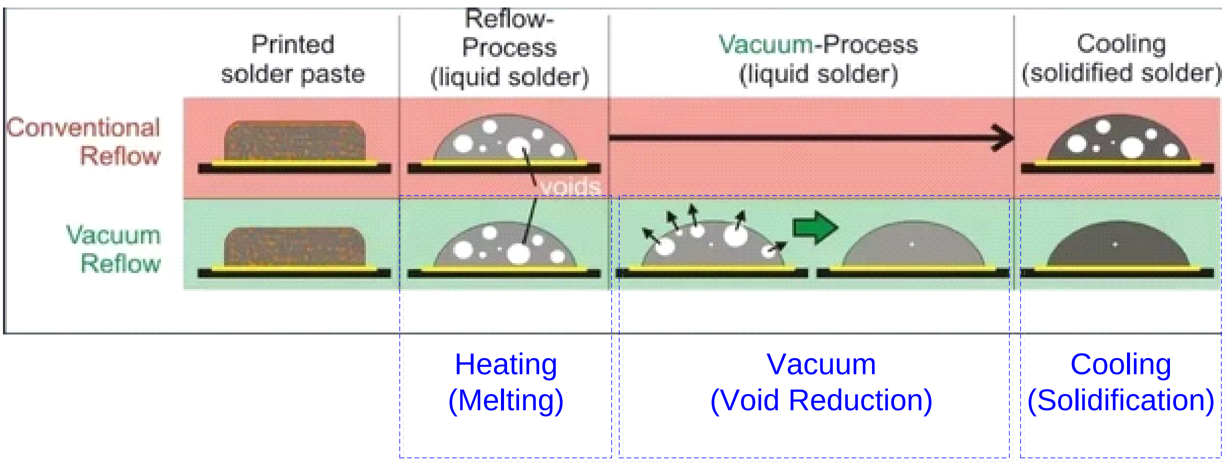

Figure 4: Steps of Vacuum Reflow Soldering

3.4. Follow of Recommended Soldering Profile

Lead-free SMT reflow profile should be used for surface mounting of AG35 module.

The reflow profile is determined by PCB density and type of solder paste being used. The recommendations from paste manufacturers should also be considered to determine the proper reflow profile.

The figure below is a recommended reflow soldering profile for lead free reflow of AG35 module.

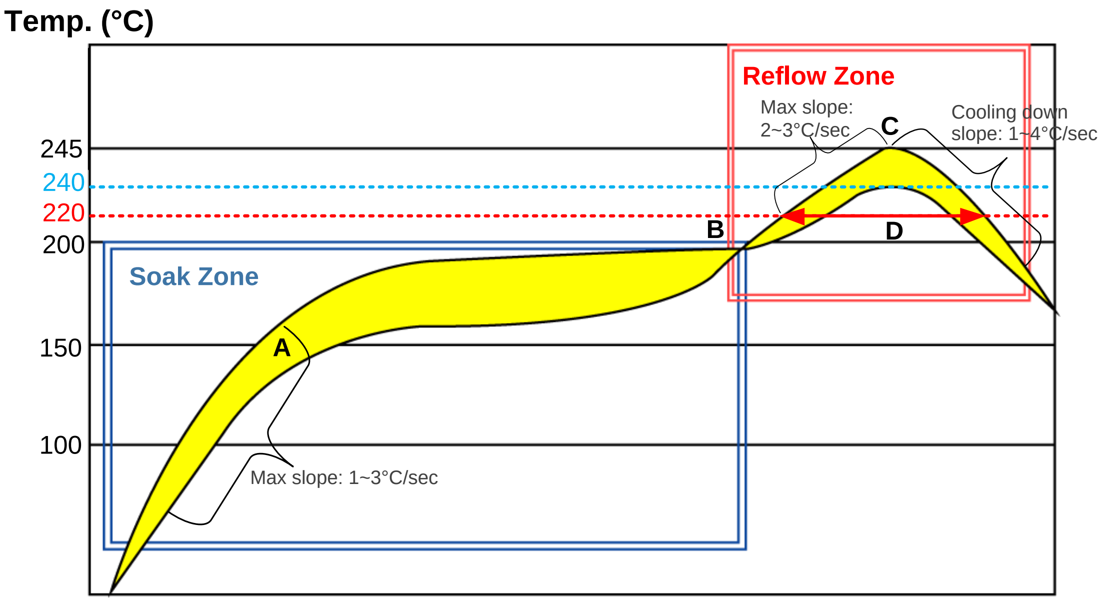

Figure 5: Recommended Reflow Soldering Profile

The detailed recommendations are presented in the table below for consideration.

Table 1: Recommended Reflow Soldering Profile Parameters

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

Reflow Cycle	
Max reflow cycle	1

NOTES

1. It is recommended to perform reflow under nitrogen atmosphere in order to minimize size of solder joint voids occurred when soldering the LGA module.
2. The number of heating zones should be at least 10 in order to achieve good soldering profile.

4 Rework Guidelines

Rework tools and operating parameters are customer/application-specific. Rework tools, heating profiles and the rework process should be tailored to these specific needs for optimum performance.

Prior to any rework, if the component's floor life has been exceeded, it is highly recommended to bake the PCB in order to remove moisture from the assembly. (Please refer to *JEDEC J-STD-033* paragraph 6 - Board rework for more details). The pre-baking process will prevent damage to any component due to moisture vapor pressures caused during reflow.

Prior to the removal, please make sure AG35 module is an integrity (that is, the module and its shielding are not separated) so as to withstand the high temperatures during reflow.

4.1. Remove Components

The removal step includes reflow of the solder joints attaching components to the PCB. Ideally, the reflow profile for component removal should be the same as the one used for component attachment. However, the time above liquidus can be reduced as long as the reflow is completed.

During the removal process, it is recommended that the board should be heated from the bottom side using convective heaters and hot gas, or hot air or IR should be used on the top side of the component. Special nozzles or IR lens should be used to direct the heating in the component area and heating of adjacent components should be minimized. Meanwhile, excessive hot air flow should also be avoided to prevent the components from overheating.

Once the joints have reflowed, the vacuum lift-off should be automatically engaged for pick-up during the transition from reflow to cool down.

NOTE

If heating conditions are not properly controlled during manual hot removal from PCB assembly, the module's package integrity can be damaged by overheating.

4.2. Clean Pads

Once the component has been removed, the site and pads need to be cleaned properly. Once the residual solder has been removed, the ground pads should be cleaned with a solvent. The solvent is usually specific to the type of solder paste used in the original assembly and the recommendations from paste manufacturers should be followed.