SMT Application Notes for BGA

BGA 表面黏着作业说明书

Document Number : RT01-SMT-0002
Version: 02
Release date: 2011-03-21

Table of Contents 索引

Document Revision History 文件版本沿革

Revision	Date	Author	Description
02	2011-03-18	FK Pan Dell Hsu Stanley Lin	SMT Application Notes for BGA

1 Introduction 简介

This document is a SMT user guideline for BGA package type which provides package description, PCB and SMT stencil design recommendations, verification methods of SMT process and optimization.

这是份针对球栅阵列封装贴片工艺的使用指引，内容提供了芯片的描述，印刷电路板跟贴片工艺钢网的设计建议，还有贴片工艺的验证方法与优化。

This report is regarding to the successful evaluation of BGA SMT capability. We recommend that our customers fully characterize their SMT process.

这份文件针对球栅阵列封装贴片工艺提供了一个好的的评估验证，我们建议客户完整的评估自己贴片工艺。

2 Package Design Data 产品设计说明

2.1 0.4mm ball pitch design 0.4mm 球间距设计

2.1.1 MT6573 Package outline drawing MT6573 产品外观尺寸图

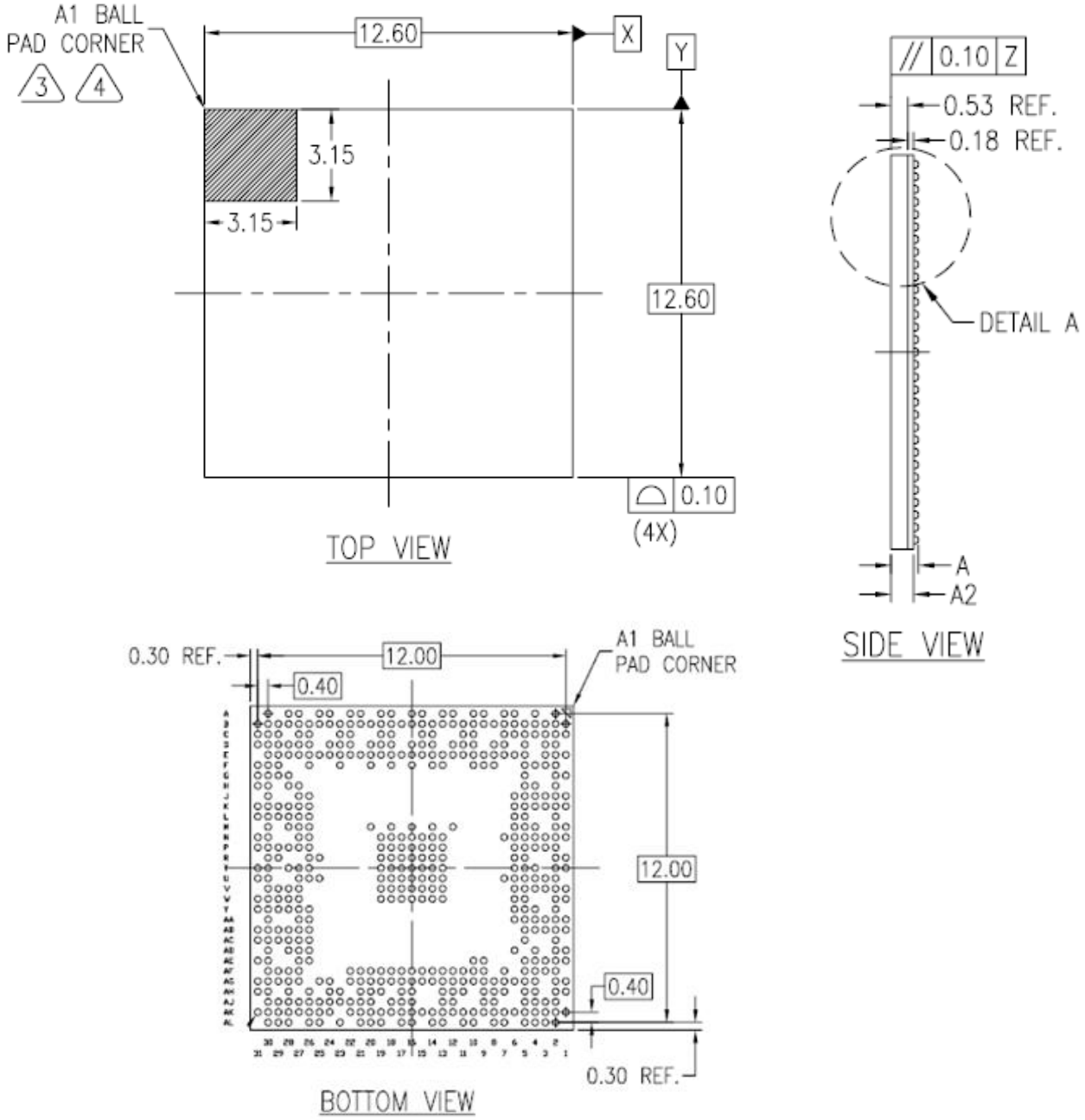

Figure 2-1 MT6573 package outline drawing

3 PCB design recommendation 印刷电路板设计建议

3.1 Common PCB design recommendation 印刷线路板通用设计规范

3.1.1 Placement of blind via under chip recommendation 芯片下方的盲孔布局建议

- Ensure the blind-via at the center of pad or more than 2mil far away from the pad, then it would be able to avoid the SMT failure resulted from deformed pad or the incomplete solder mask clearance.

  保证盲孔放置于焊盘正中心或完全在距离焊盘 2mil 以外，这样可以避免因为焊盘变形或不完整的防焊绿油环所造成的贴片工艺不良。

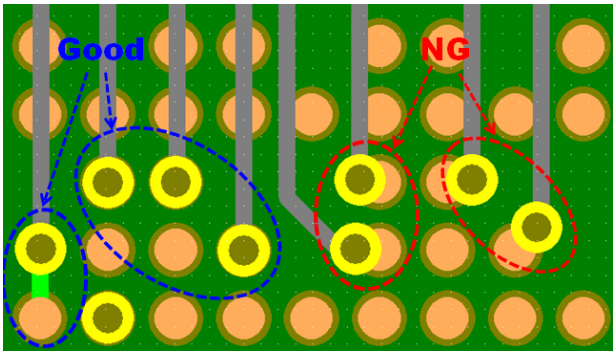

Figure3-1 Recommended placement of blind via

3.1.2 Top layer routing under BGA recommendation BGA 下方表层设计建议

3.1.2.1 Chip fan-out recommendation 芯片下方的出线建议

- Refer to the PCB pad design of each product for the r fan out trace width.

  出线线宽建议请参照各产品焊盘设计建议说明。

- For the signals with large current, recommend to widen the trace outside the solder mask opening.

  对于大电流的讯号脚位，建议在绿油开窗外再放宽线宽。

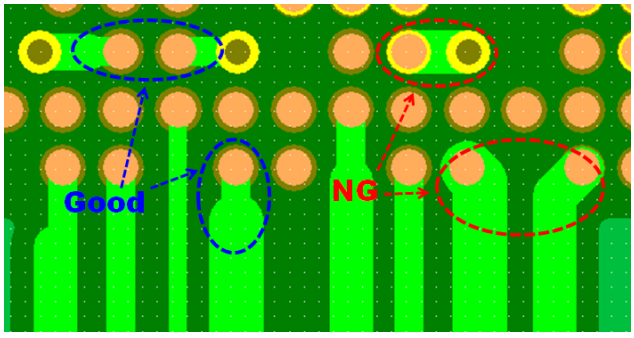

Figure3-2 Chip fan-out recommendation

3.1.2.2 Forbid making copper plane on the surface layer under BGA 禁止在 BGA 下方表层铺铜

- Connect the same-net pins to the inner plane with trace and vias.

  用走线和打孔的方式把相同网络的引脚连到内层的平面。

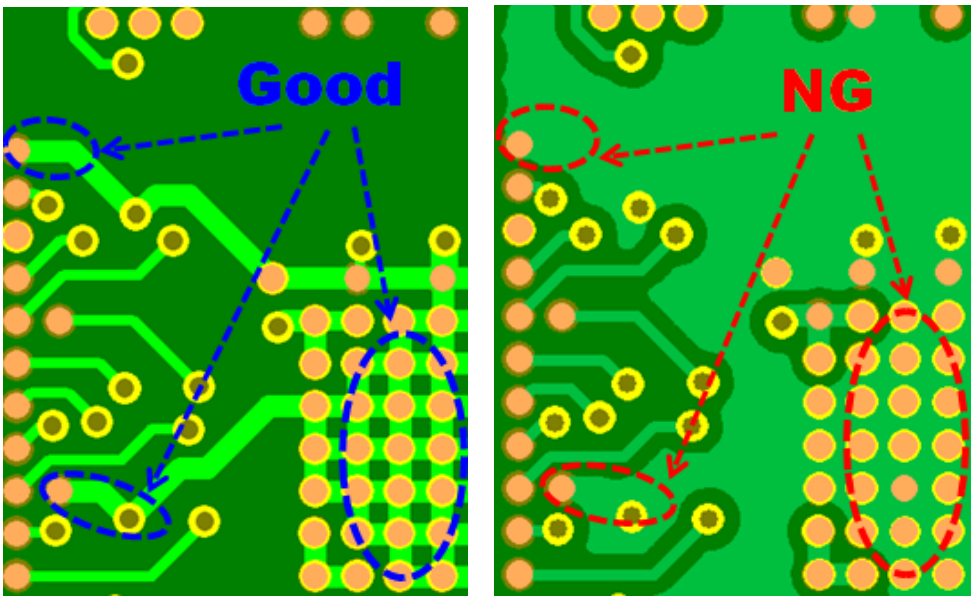

Figure3-3 Forbid making copper plane on the top layer under BGA

3.2 0.4mm ball pitch PCB design recommendation 0.4mm 球间距印刷电路板设计建议

3.2.1 Min. trace width/space recommendation 最小线宽/距建议

	Min. trace width/spacing mm(mil)
Under chip	0.075/0.075mm (3/3mil)
Out of chip	0.1/0.1mm (4/4mil)

Table 3-1 PCB min. trace width/spacing recommendation

3.2.2 Via size recommendation 钻孔尺寸建议

Via Size➔drill/pad mm (mil)
Blind via➔ 0.1/0.3mm (4/12 mil)
Buried via➔ 0.25/0.45mm (10/18 mil)
Thru-via➔ 0.3/0.5mm (12/20 mil)

Table 3-2 PCB via size recommendation

3.2.3 Solder mask design recommendation 防焊绿油设计建议

- Recommend using copper defined for the solder mask printing, and forbid overlapping to pads.

  建议将防焊绿油印刷在焊盘之外，并禁止覆盖到焊盘。

- Design the solder mask opening by the rule of Figure3-4 and Table3-3.

  建议参照图 3-4 与表 3-3 的规范进行防焊绿油开窗设计。

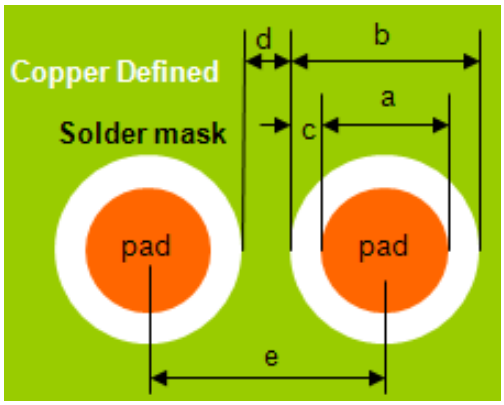

Figure 3-4 PCB solder mask illustration

Solder mask opening rule mm (mil)
Pad size ➔**(a)** =0.25 mm(9.8mil)
Ball pitch➔**(e)** =0.4mm(15.8mil)
Solder mask opening➔**(b)** =a+0.075 mm(3mil)
Mask clearance➔**(c)**=0.038mm(1.5mil)
Solder dam➔**(d)**=e-b=0.075 mm(3mil)

Table3-3 Solder mask opening rule.

3.2.4 Footprint and stencil reference design 组件封装和钢网参考设计

3.2.4.1 MT6573

- **PCB land pattern drawing 印刷电路板脚位图**

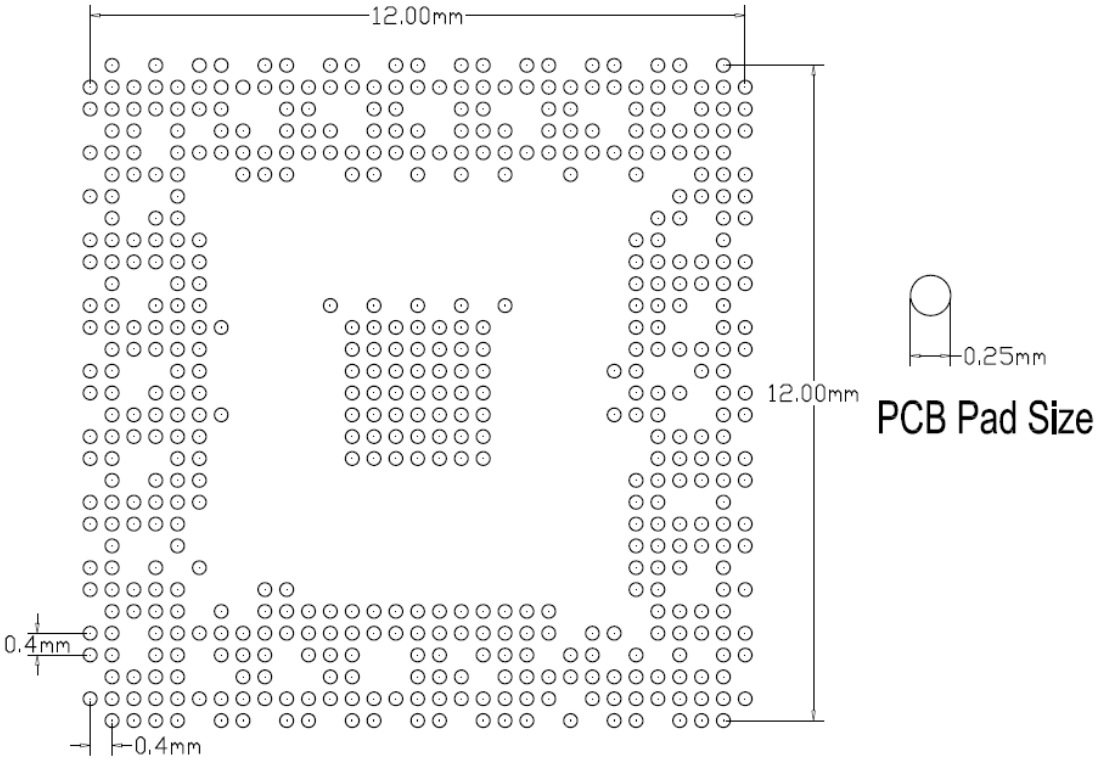

Figure 3-5 MT6573 PCB Footprint recommendation

- Recommend all pad size should be designed the same as package terminal size, the actual PCB accuracy should be controlled within +/-0.025mm (1mil). Fan out pad with 0.2mm (8mil) trace.
  建议所有焊盘尺寸包含中央焊盘按照封装尺寸 1:1 制作，实际印刷电路板成品精度需控制在+/-0.025mm (1mil), 并建议使用 0.2mm (8mil)线宽出线。

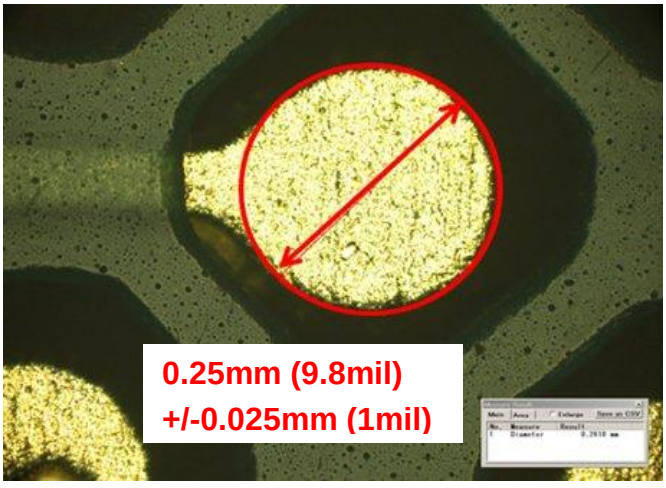

Figure 3-6 MT6573 pad size requirement

■ Stencil design recommendation 钢网设计建议

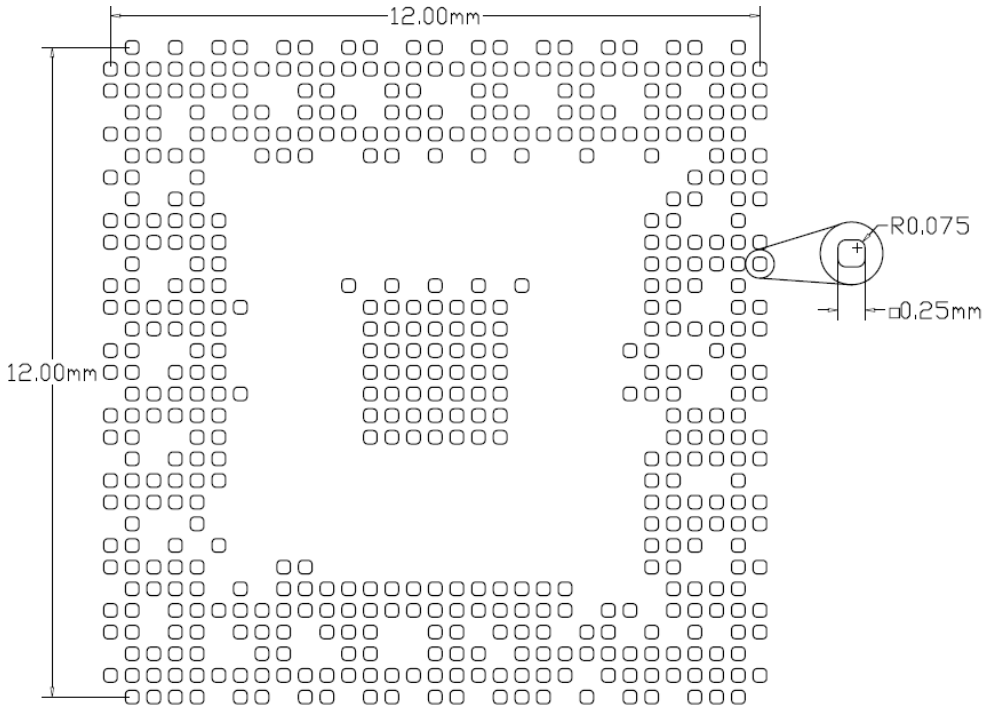

Figure 3-7 MT6573 stencil design

4 SMT Verification 贴片工艺确认

It is very important to fully characterize the SMT process for optimal results. It is recommended to characterize the materials, equipment, and PCB land pattern according to each customer's process, stencil design, and reflow profile prior to the mass production.

为了最佳的贴片工艺结果，完整了解贴片工艺特性这是非常重要的。我们建议在量产之前针对客户自己的贴片工艺制程， 钢网设计与回焊温度去了解材料， 机台设备，与印刷电路板焊垫的特点。

In order to achieve a more stable process throughout the BGA SMT assembly, the following items need to taken into consideration before mass production

- Material storage and usage
- Solder paste selection
- Screen printing process control
- Component Pickup and placement
- SMT reflow profile
- SMT verification

为了实现一个更加稳固的球栅阵列封装贴片工艺组装，下面的特性分析是需要被列入考虑的：

- 物料的储存与使用
- 锡膏选择
- 确认印刷工艺
- 零件取件与贴片
- 贴片工艺回焊温度曲线
- 贴片工艺验证

4.1 Material storage and usage 物料的储存与使用

Most of BGA type packages are defined as moisture sensitive devices, therefore, understanding the risks of moisture absorption is important. For the package which is defined as MSL 3, please follow the procedure below;

1. Shelf life in sealed bag: 12 months at <40°C and <90% relative humidity (RH).
2. After this bag is opened, devices that will be subjected to infrared reflow, vapor-phase reflow, or equivalent processing must be:

a. Mounted within 168 hours at factory conditions of ≤30°C/60% RH. Or
b. Always keep the materials in the Dry Cabinet is strongly recommended. Figure 4-1 is the picture of Dry Cabinet.

3. Devices require baking, before mounting, if:
   a. Humidity Indicator Card is >20% when read at 23°C+-5°C, or
   b. The floor life (168 hours) is exceeded.
4. If baking is required, devices may be baked for:
   a. 312 hours at 40°C+5°C/-0°Cand <5% RH for low temperature device containers, or
   b. 9 hours at 125°C+5°C/-0°C for high temperature device containers.

Regarding solder material, it is also very crucial for SMT yield. Keep the paste in the refrigerator at 0~10°C. In addition, it is necessary to define an appropriate procedure and then follow it strictly.

大部分 BGA 封装的材料皆被定义为湿气敏感组件。所以，了解零件吸收过多湿气的风险，并依湿气敏感零件之规范来作业是非常重要的。例如材料是定义为等级三， 即必须遵循下列之注意事项：

1. 材料为真空密封包装之储存期限为 12 个月，储存环境条件: 在温度<40°C，相对湿度<90% RH.
2. 包装拆封后，材料必须：
   a. 储存在≤30°C，≤60%RH 的环境中，并在 168 小时内完成 SMT 焊接程序。
   b. 建议总是将材料放置在防潮箱内. 图 4-1 为一防潮箱的照片。
3. 如果湿度指示卡显示>20%或放置时间超过 168 小时便须重新烘烤 。
4. 烘烤的条件为：
   a. 如果装载 IC 的包装材料无法承受高温，需在 40°C+5°C/-0°C，烘烤 312 小时。
   b. 如果装载 IC 的包装材料可承受高温，烘烤条件为 125°C+5°C/-0°C，烘烤 9 小时。

关于锡膏，使用前除了需储存在 0~10°C 的冰箱中， 也必须定义出一个合理的作业流程，并确保人员有依照流程作业。

Figure 4-1 Dry Cabinet

4.2 Solder paste selection 锡膏选择

Before deciding solder paste, MTK would recommend starting off a DOE to verify which solder paste is most suitable for this package and board. Based on MTK's experience gained from previous

studies, the solder paste with type 4 or above powder size is recommended to be used for 0.4mm pitch BGA. In MTK's studies, it demonstrated that Senju M705-S101-S4 and KOKI S3X48-M500 solder paste combining an appropriate reflow profile could provide positive yield results. If needed, consult your solder paste vendors and MTK for further recommendations.
联发科建议在大量生产前做一些工程验证来定义出哪一款锡膏是最适用在您的产品上。根据联发科验证的结果，四号粉或更小尺寸锡粉的锡膏是适合使用在 0.4mm 间距的 BGA 上的。由验证结果来看，使用 SENJU M705-S101-S4 和 KOKI S3X48-M500 的锡膏再结合适当的回焊曲线能够达到理想的焊接效果。如果在选用锡膏上有任何疑问，请随时联系联发科的相关负责人，我们会立即提供适当的建议。

4.3 Screen printing process control 印刷制程控制

Printing process plays a very significant role for fine pitch BGA soldering issue. The items need to be validated are printing accuracy, printing repeatability and paste release quality. It is learned from the SMT studies conducted by MTK that the production yield would be low and unsteady if the printing quality isn't under control. Understanding the key factors and to setup a proper printing process can prevent from some of soldering defects. It is recommended perform 100% inspection on printing result in the leading panels.
印刷工艺对 BGA 焊接工艺都是一个很关键的因素。检查的要点包含印刷的精度与偏移状况，印刷的连续性与下锡的效果。在联发科多次的验证中发现，如果印刷工艺未能有效的控制，生产质量将非常不稳定。预先了解其重要性并设置一理想适合的制程条件便能够预防一些焊接的问题。另, 建议针对一开始投入的几片板子执行 100% 检验,以确保所设置的制程条件与参数是合理的.

Figure 4-2 shows an example that the PCB pads were applied with solder paste. The paste volume is sufficient with desirable formation.
如照片 4-2 中的锡膏板,其 锡量与成型都非常理想且无拉尖的现象.

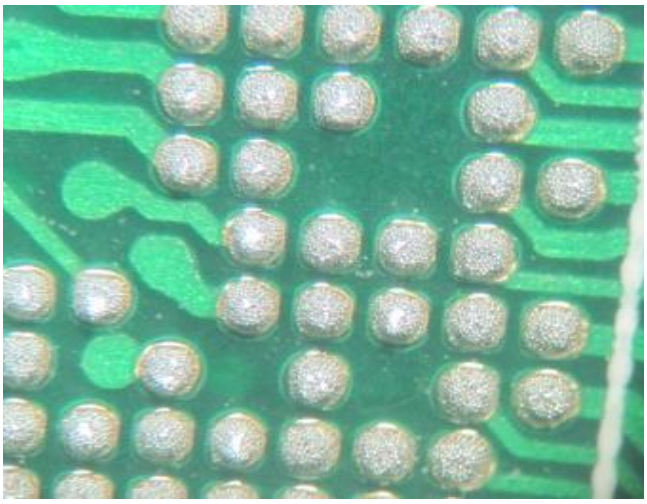

Figure 4-2 The PCB pads are applied with sufficient solder paste

4.3.1 Stencil fabrication 钢网制作工艺

Three major methods of stencil fabrication are used in current SMT industry commonly, there are chemical etch, laser cut with electro-polishing (or nickel plating) and electroforming. Both laser cut and Electroforming stencils are good for 0.4mm ball pitch BGA SMT application since the smooth sidewalls would provide desirable paste release result, nevertheless, it is proven that the yield would be slightly better if electroforming stencil is used. The most popular method right now is laser cutting stencil. Please note that further process, such electropolishing or nickel plating, will be required for obtaining a smoother aperture walls when laser cutting stencil is used.
Table 4-1 is the comparison between these 3 fabrication technologies.
业界有三种钢网制作工艺，分别是化学蚀刻，激光切割加上电解抛光(或孔壁镀镍处理)与电铸。0.4mm 间距的 BGA 可使用激光和电铸钢网因为其光滑的孔壁可提供理想的下锡。由实验证明电铸钢网可得到较佳的印刷质量。目前最普遍的制作工艺仍是激光切割。请注意如果选择激光切割工艺，之后的电解抛光及孔壁镀镍必须执行以确保能有较光滑的孔壁。
表 4-1 是就钢网所需的制造时间，精度，孔壁毛边状况与成本做一比较。

Fabrication technology	Chemical Etch	Laser Cut +Electro-polishing or Nickle plating	Electroform
Fabrication time	Plus	Plus	Minus
Accuracy	Minus	Plus	Plus
Sidewalls quality	Minus	Neutral	Plus
Cost	Plus	Neutral	Minus

Table 4-1 Comparison table for current stencil fabrication methods

4.3.2 Screen printer 丝印机

In general, in the solder paste printing process, the printer is crucial for achieving desired print quality. The best thing to do is discuss your requirements with your machine vendor, solder paste vendor and stencil fabrication vendor, and validate if they are capable of offering the desired application.
在印刷工艺中，印刷机的状况与能力总是能左右印刷的质量。建议与印刷机的设备厂商，锡膏厂商与钢网厂商一同讨论与研究，并确认目前的设备与锡膏是否有能力达到所需要的印刷要求。

4.4 Component pickup, recognition and placement 取件, 零件识别与贴片

4.4.1 Pickup 取件

The key item needs to be taken into consideration is nozzle selection. Normally, using a larger nozzle is recommended since it could improve the placement precision.
吸嘴头的选择对取件来说是一个要件。通常，尽可能使用较大的吸嘴，因为这样能增加吸料与贴片的稳定度。

4.4.2 Recognition 零件识别

According to the MTK's evaluation results, the component recognition program is the important factor to impact placement quality. It is recommended having a pre-study in component library edition prior to mass production. The component library includes component sizes, lighting, speed, etc.
由联发科验证的结果可知，零件的识别参数对贴装质量是非常重要的。建议在正式生产前先预先编辑零件参数并实际识别与贴装，以确保参数为优化。通常零件数据库包含零件尺寸，辨识的光源，工作速度等等。

4.4.3 Placement 贴装

The placement precision is equipment dependent. However, it also correlates closely with nozzle selection and component library. As for placement pressure, using default setting as the starting point will be recommended, and it may increase the possibility of solder bridging if extra pressure is applied.
X-ray inspection can be a method to validate the placement performance. Figure 4-3 is an x-ray image shows that there's misplacement problem. And good placement is shown in figure 4-4.
一般来说，贴装精度取决于设备的能力与状况.然而，贴装的精度与吸嘴头的选用，零件的数据库也息息相关。关于贴装压力，通常于生产 0.4mm 间距的 BGA 时会建议使用默认值，多余的贴装压力会增加锡球桥接的风险。建议使用 x-ray 来确认贴片的质量。图 4-3 是贴片偏移 x-ray 的照片，而图 4-4 所示是贴片正常的 x-ray 照片。

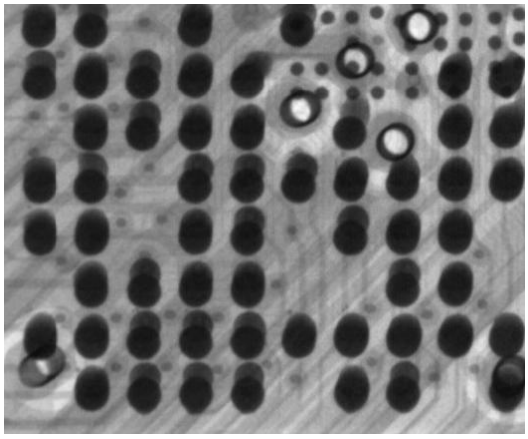

Figure 4-3 Misplacement image

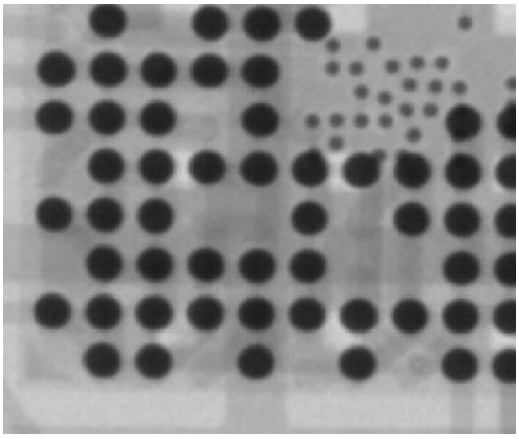

Figure 4-4 Good placement image

4.5 SMT reflow profile 贴片工艺回焊曲线

The SMT profile is just only for reference. Customers may adjust the profile base on the variations of PCB design, solder paste, and reflow oven. The profile show in Table 4-1 is MTK's typical reflow profile.

It is very important that customers follow their own solder paste vendor suggestion for screen printing process parameters and reflow profile conditions

贴片工艺的回焊曲线只是仅供参考。客户可以根据印刷电路板设计，焊膏和回焊炉的差异自行调整。表 4-2 是联发科典型的回焊曲线。

客户按照自己的锡膏供货商建议去设定钢网印刷工艺参数与回焊条件这是非常重要的，

Profile stage	Description	Condition limits
Preheat	Initial ramp	< 3°C/sec max
Soak	Dry out and flux activation	135°C - 175°C 60 - 120 sec
Reflow	Time above solder paste melting	40 - 90 sec
	Peak temperature	245°C max
Cool down	Ramp to ambient	6°C/sec max

Table 4-2 MTK typical lead-free reflow condition

4.6 SMT verification 贴片制程验证

4.6.1 X-ray X 光检测

The photo Figure 4-5 is an acceptable graph performed by solder paste SENJU M705-S101-S4 and shows good placement alignment, and no solder bridge.

图 4-5 为范本，该图使用千住 M705-S101-S4 且表现出好的置件对位以及没有桥接发生。

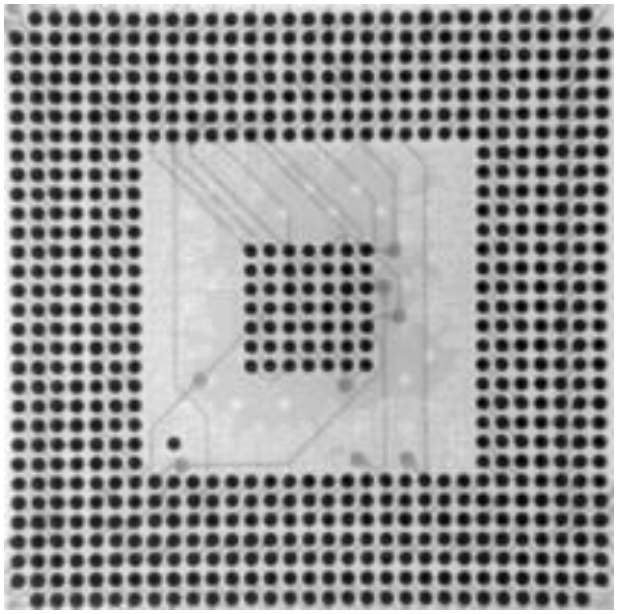

Figure 4-5 X-ray: Good alignment and no solder bridge

Figure 4-6 shows solder bridging at the package corner.

图 4-6 表现出回焊后在产品的角落发生锡球桥接。

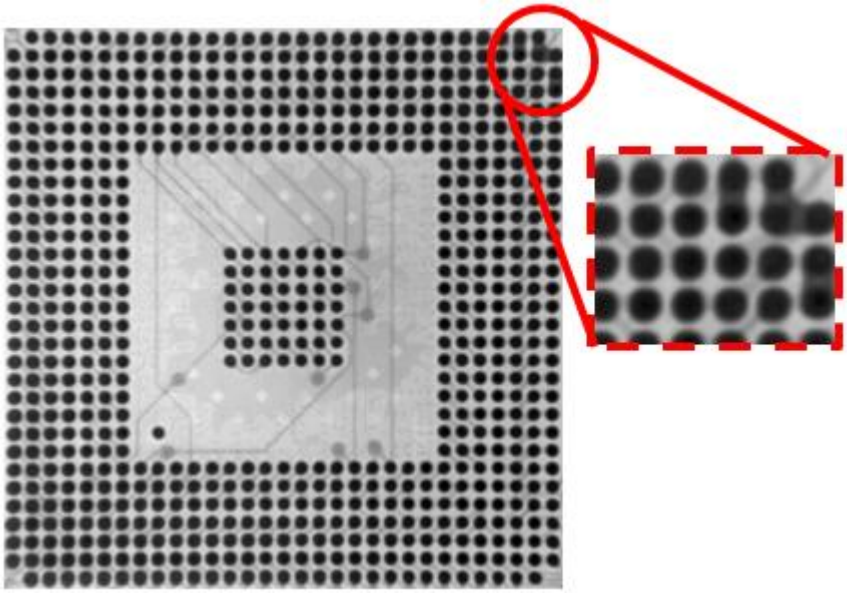

Figure 4-6 X-ray: Solder bridge at corner

4.6.2 Cross section 切片

According to MTK evaluation, using SENJU M705-S101-S4 solder paste, and figure 4-7 shows the cross section of good solder joint.

在联发科评估的过程中，使用千住 M705-S101-S4 锡膏贴片，图 4-7 表现出好的成球。

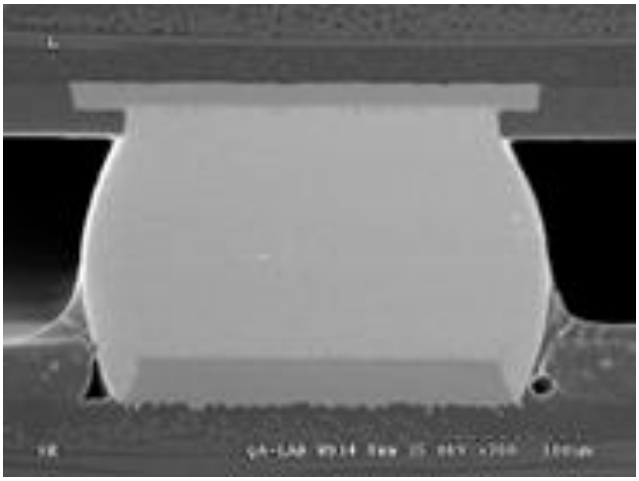

Figure 4-3 Good solder joint on PCB

Figure 4-8 shows common solder non-wet. Various factors may result in this kind of non-wet, such as solder paste activity, PCB design, reflow temperature.

图 4-8 表现出一般常见的不沾锡假焊。很多原因会形成此类型假焊，诸如锡膏的活性，印刷电路板的设计，回焊的温度都有可能导致此类型的假焊。

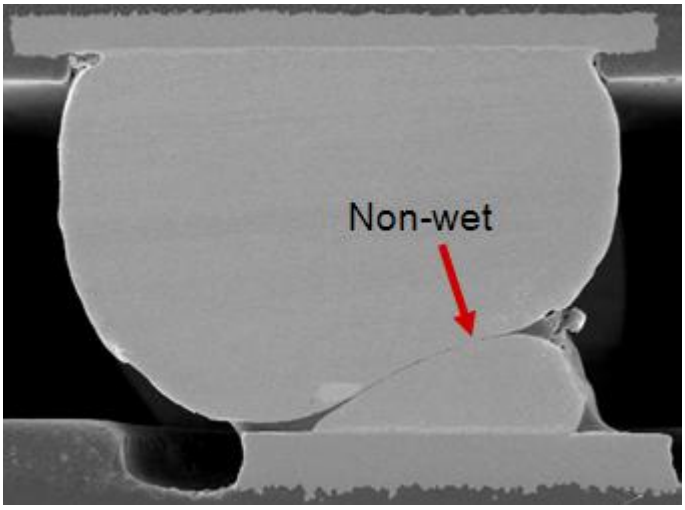

Figure 4-7 Solder non-wet cross section