Important notice

Dear Customer,

On 7 February 2017 the former NXP Standard Product business became a new company with the tradename **Nexperia**. Nexperia is an industry leading supplier of Discrete, Logic and PowerMOS semiconductors with its focus on the automotive, industrial, computing, consumer and wearable application markets

In data sheets and application notes which still contain NXP or Philips Semiconductors references, use the references to Nexperia, as shown below.

Instead of http://www.nxp.com, http://www.philips.com/ or http://www.semiconductors.philips.com/, use **http://www.nexperia.com**

Instead of sales.addresses@www.nxp.com or sales.addresses@www.semiconductors.philips.com, use **salesaddresses@nexperia.com** (email)

Replace the copyright notice at the bottom of each page or elsewhere in the document, depending on the version, as shown below:
- © NXP N.V. (year). All rights reserved or © Koninklijke Philips Electronics N.V. (year). All rights reserved
Should be replaced with:
- **© Nexperia B.V. (year). All rights reserved**.

If you have any questions related to the data sheet, please contact our nearest sales office via e-mail or telephone (details via **salesaddresses@nexperia.com**). Thank you for your cooperation and understanding,

Kind regards,

Team Nexperia

APPLICATION INFORMATION

AN01026

(LF)BGA APPLICATION NOTE
ATO INNOVATION, PHILIPS SEMICONDUCTORS
MARCH 2000

Philips
Semiconductors

PHILIPS

BGA Application Note

CONTENTS

BGA Application Note

1 INTRODUCTION

The plastic Ball Grid Array (BGA) and Low profile Fine pitch BGA (LFBGA) packages have developed over recent years from being rarely used devices, to become, for many applications, the first choice for designers requiring medium to high pin-count IC packaging.

When comparing it to other common alternative packages, such as the Quad Flat Pack (QFP), the (LF)BGA device has many clear advantages. Such as:

- The (LF)BGA has no easy-to-bend leads that can cause deviation from coplanarity.
- The (LF)BGA is typically 20% to 25% smaller than an equivalently functional QFP.
- Resolution and smearing problems with respect to the stencil-print process are less because the pitch is larger, and the apertures are circular.
- The self-alignment property of the component results in a large process window for automatic placement.
- The (LF)BGA is compatible with today's assembly techniques, which means that no adjustments are necessary to standard machines or materials.

This application note describes the status of the technology in February 2000. Due to continuous improvements in material processing and board assembly, the specifications of, for example, the BGA substrates can change over time.

1.1 Technology boundaries

The most important items of (LF)BGA technology are shown below.

Printed circuit board

- conductor lines: 100 µm, spacing: 100 µm
- solder resist resolution: 100 µm
- placement accuracy:
  - 75 µm for BGA
  - 62.5 µm for LFBGA
- multilayer

Process characteristics

- Base material: FR4/polymide
- Finish: Ni/Au
- Solder paste (e.g.):
  - RMA for BGA
  - Alpha metals LR735 for LFBGA
- Stencil thickness: 150 µm uniform
  (for an LFBGA with a pitch of 0.5 mm, a stencil of 100 µm will result in a larger process window)
- Process: double-sided reflow
- Printing accuracy: 50 µm
- Operator knowledge: advanced training

Process capability

- Overall failure ≤ 20 ppm (95% confidence level per solder joint)

2 OUTLINE VERSIONS

Philips Semiconductors offers several (LF)BGA outline versions with different pin count, pitch and body dimensions. An overview of our current range is show in our Data Handbook IC26: *Integrated Circuit Packages*.

This application note contains all aspects for processing the (LF)BGA, including footprint design, stencil printing, automatic placement, reflow soldering process and rework.

3 REFLOW SOLDERING PROCESS

The reflow soldering process comprises three steps:

1. Stencil printing
2. Component placement
3. Reflow soldering.

3.1 Stencil printing

For (LF)BGA assembly, stencil printing is the most critical process step. This is because that, after soldering, the joints cannot be inspected as is the case with other surface mounted devices.If one of the balls is not properly soldered, it can only be detected using electrical testing, X-ray or destructive analysis.

Stencil printing can be divided into three sub-processes:

- Filling of apertures
- Levelling
- Stencil release.

These three steps are shown in Fig.1.

3.1.1 FILLING OF APERTURES

The filling of the apertures is determined by a complex interaction between the material properties of the stencil squeegee, solder paste and the machine settings, such as pressure and speed. To ensure the apertures are properly filled, the solder paste must “roll” on the stencil in front of the squeegee. By rolling the paste, the highest pressure arises at the point where the squeegee is in contact with the stencil. This is the place where the solder paste can flow into the stencil apertures.

3.1.2 LEVELLING

The elevating force on the squeegee determines whether the squeegee correctly levels the paste in the stencil openings. This force is determined by factors such as the amount of paste, the squeegee angle and stiffness, the print speed, and the applied pressure on the squeegee. The last two, which are machine parameters, can be controlled by the operator until a satisfactory result is obtained.

The process window can be increased in two ways: by increasing the pressure on the squeegee and/or by using a stiff squeegee. For example, increasing the pressure above the level that is simply required to clean the stencil ensures sufficient paste is forced through the stencil aperture onto the solder land, which in turn increases the process window. Likewise, using a stiff squeegee increases the process window. However, a careful balance has to be maintained because if the squeegee is too stiff, it would not be able to follow the shallow contours on the board. Refer to Section 5 for more information on the process window.

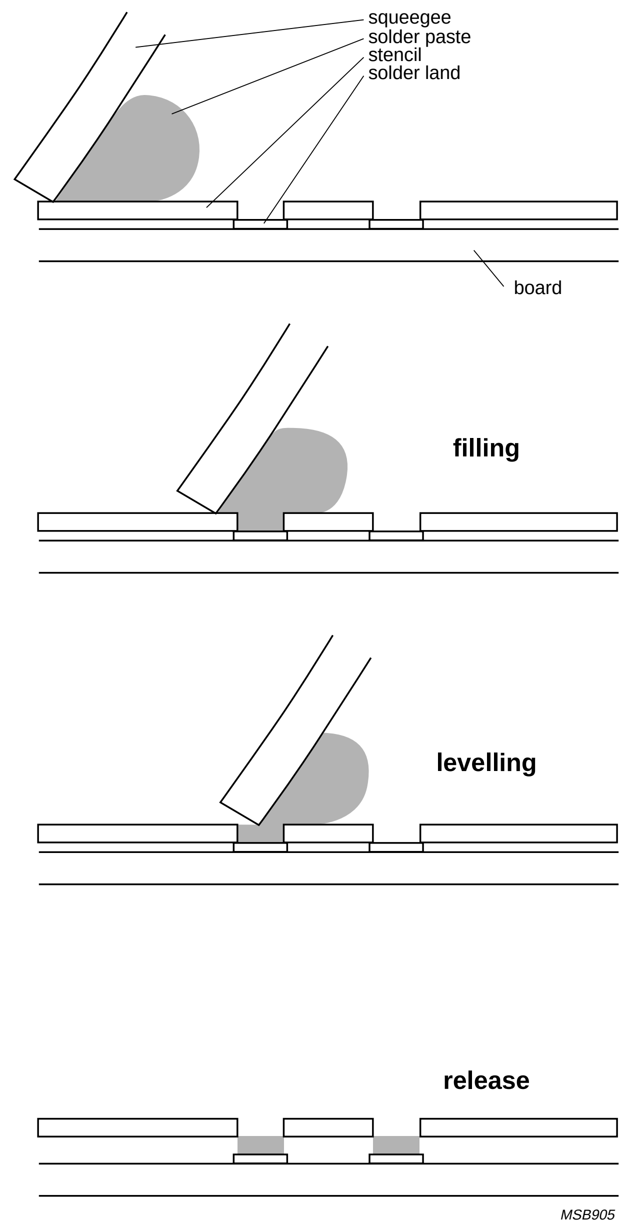


Fig.1 Applying solder paste by stencilling

3.1.3 STENCIL RELEASE

When the apertures are filled with solder paste, the stencil and board are separated. The way of separating determines the smallest printable opening. Though experimentation, we have found that stencil printing in combination with a slow, controlled release speed gives the best results, compared with printing with zero snap-off. The highest resolution is obtained as the separation speed is slow and with small “jerks”. This ensures that any excess paste is removed from the apertures caused by the resulting vibrating motion.

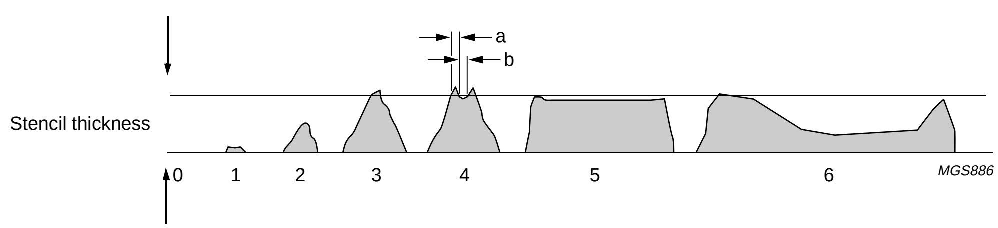
3.1.4 PRINT RESULTS

The circular stencil apertures are relevant. Figure 2 shows the typical appearance of solder paste deposits as the diameter of the stencil openings increase. Also, owing to process vibrations, the deposition may look different even when using only one stencil opening.

3.2 Component placement

The key factor in component placement is the size of the component. For example, small passive components are usually placed with a chip shooter, whereas larger components, such as ICs, are placed with an IC placer. (LF)BGAs are considered large components, irrespective of their actual size.

The major process deliverable of a placing machine are its speed and placement accuracy. The latter of which is determined by the vision alignment system. In addition, placement force is also important it ensure optimum contact. Too high a force and the component may crack damage the solder land; to low a force and the component will have poor contact with the solder paste.



0 No paste.
1 Irregular shape (few balls).
2 Pyramid lower than stencil thickness.
3 Pyramid larger or equal to stencil thickness (acceptable).
4 Beginning flat top with “dog ears” (preferable).
5 Flat topside (b > a) (preferable).
6 Scooped-out or bulldozed.

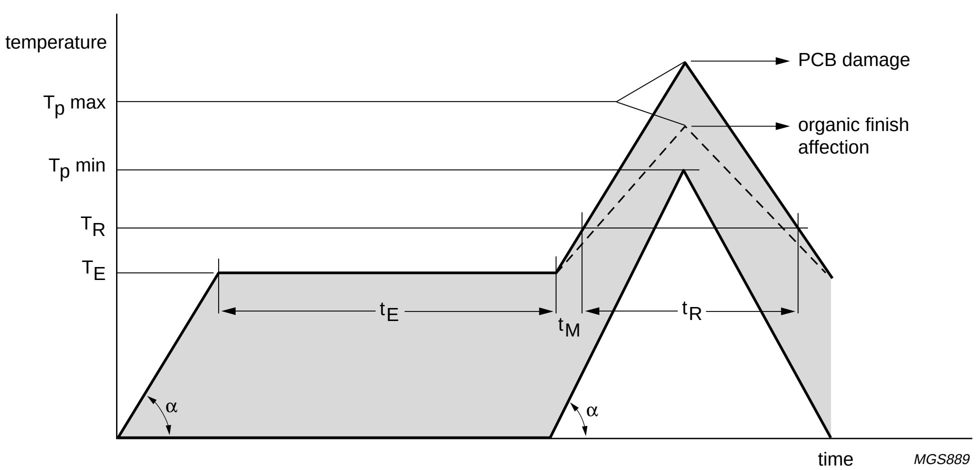
Fig.2 Different deposition appearances with increasing stencil opening diameter.

3.3 Reflow soldering

During soldering, metal pads are joined by molten solder that flows between their adjacent surfaces, which have a higher melting point than the solder itself. The parts to be joined, and the solder paste are heated so that the flux can remove the oxides. After this, the solder is then brought above its melting point. As the solder melts, it flows around the ball contacts of the BGA and forms a meniscus. After solidification, the final joint is formed. To achieve a suitable soldered joint, all parts of the board must be subject to an accurate temperature/time profile. Figure .3 shows a suitable profile framework.

It is important to note that this profile is based on the properties of the printed circuit board and the solder paste. As each component on the board has its own specific profile (see example in Fig.4), the components profile must be superimposed on the board's profile to give a true representation of a particular product's process window (see Fig.5).

Note: All temperatures are measured at the solder joint.



α	$\leq$ 10 °C	t_R	$\leq$ 70s
T_E	$\leq$ 160 °C	t_M	= 2 to 30 s
T_R	$\leq$ 180 °C	$T_p$min	= 205 °C

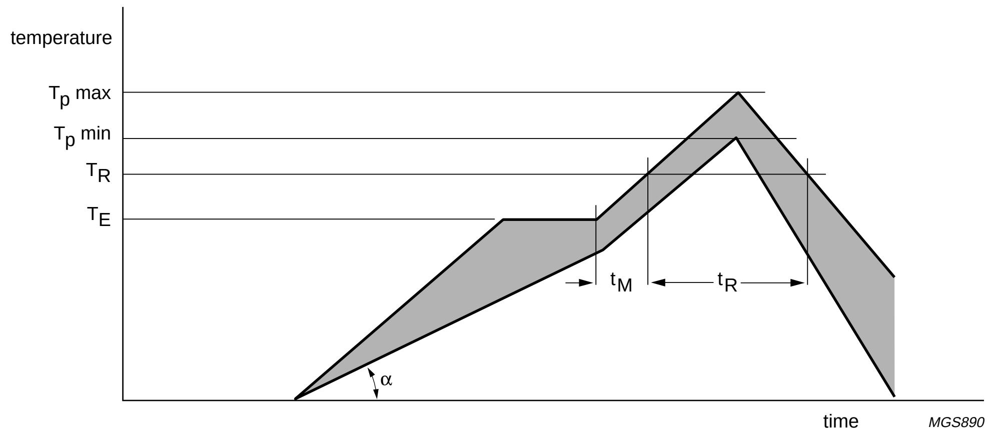
t_E $\leq$ 1 min, if possible (else $\leq$ 5 min)

$T_p$max = 240 °C for soldering the first side of a double-sided board with organic finish

$T_p$max = 280 °C for all other cases.

Fig.3 Reflow soldering process requirements for boards and paste.

BGA Application Note



α ≤ 10 °C/s
$T_p$min = 205 °C
$T_p$max = 235 °C
T_M = 10 to 30 s.

Fig.4 Process window for a specific component.

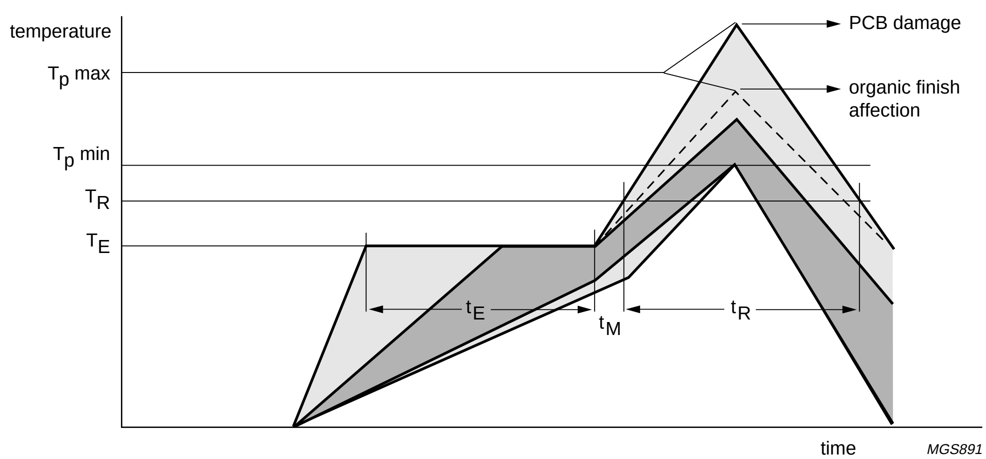


Fig.5 Example of a narrowed process window because of component restrictions.

4 FOOTPRINT DESIGN

An important step in the design of a printed circuit board is the choice of footprints. A well-chosen footprint is the basis for a reliable solder joint.

4.1 Print board dimensions and pattern positions

The accuracy by which the conductive pattern can be positioned relative to fiducial marks is an important factor in footprint design. A large tolerance on a pattern position results in a large footprint, and limits the number of components that can be assembled on a board. The maximum standard dimension of a printed circuit board is 300 mm × 200 mm. And the maximum deviation in distance between the position of a fiducial mark and the conductive pattern of a processed board relative to CAD data is 0.04 mm, or 0.05%, whichever is the larger. The finest pitch that can, therefore, be processed within such technology is 0.4 mm.

4.2 Component placement

The (LF)BGA package owes much of its popularity to the fact the component is self aligning, i.e. during reflow, an (LF)BGA that has not been properly placed, will float back to its optimal position on the solder lands thanks to surface tension forces (see Fig.6).

The maximum displacement for e.g. a BGA256 is 400 µm and for a LFBGA64 is 125 µm. Furthermore, as the (LF)BGA does not have leads that can be bent, the placement force can have a large tolerance of roughly between 3 N and 10 N for BGAs, and between 1.25 N and 10 N for LFBGAs. This force is dependent on the board support and the construction of the placement force-control unit.

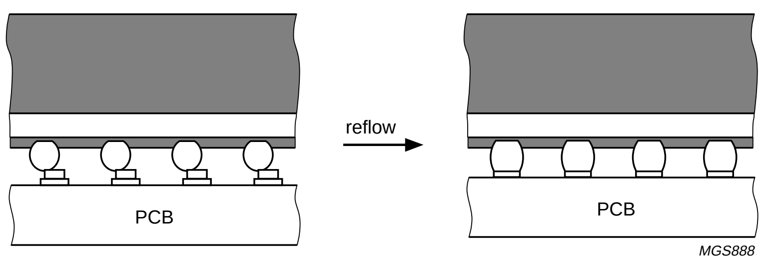


Fig.6 (LF)BGAs are correctly positioned during reflow soldering.

4.3 Reliability

Experimentation has shown that the larger the standoff height of the component, the more reliable the solder joint. One method of increasing this height is to choose a layout that is defined by the solder resist rather than the etched copper (see Fig.7). It can be seen that the solder resist and the opening determine the standoff height. As the resist thickness in a standard process technology is defined as being 12 and 30 µm, the influence is limited. However, it can play a role, particularly for small (LF)BGAs.

It’s known that connection balls directly beneath the die edges experience the maximum stress. What is not known, however, is whether the footprint design can improve the stress distribution over the solder joints.

We have found from simulations that solder joints that have the same solder land on the board and on the component are more reliable. It is, therefore, preferable to have symmetrical solder joints.

We recommend, especially for the small (LF)BGAs, copper-defined solder lands. This is because:

- more space is available for tracks running in between the solder lands
- standard copper-defined fiducials can be used.

4.4 Routing

The best situation would be that in which all the tracks are routed in one copper layer. This implies that all inner row tracks must be routed between the solder lands of the outer row. This number of inner tracks permissible depends on the board specification and the footprint design. The print board specification for standard technology stipulates that the minimum track width and conductor spacing is 0.1 mm, and the accuracy of the solder resist application with respect to the copper pattern is ± 0.075 mm.

If none, or not enough, track can be routed between the solder lands, the tracks have to be routed along vias. The minimum via land is 0.5 mm for standard technology, and the number of vias that can be placed in the array is determined by the footprint design.

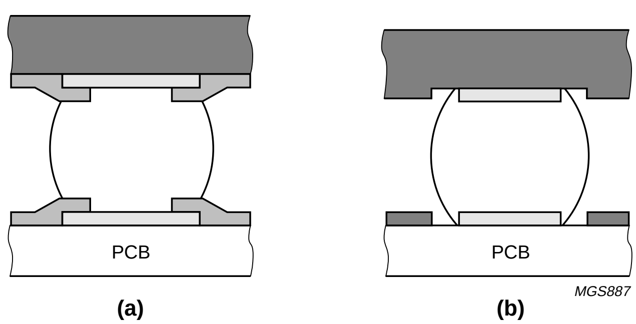


Fig.7 Different solder joint layouts (a) solder resist defined (b) copper etched defined.

5 PROCESS WINDOW

5.1 Stencil printing

For SMD assembly on printed circuit boards, one of the requirements after it has been placed on the solder paste is that all the leads (or terminations in the case of leadless components) must be in contact with the solder. This is because if a lead is not in contact with the paste, then the wetting of the lead by the liquid solder is not always good. In practice, this means that the solder pasted deposits should have the same height, i.e. their height should be equal to the stencil thickness (see Fig.2).

The fundamental difference between an (LF)BGA and normal SMD is that the leads of (LF)BGAs consist entirely of eutectic solder, whereas the leads of the standard SMDs, in most cases, only had a eutectic or tin-plating of a few tenths of microns on the leads (or terminations).

Also, the tolerance on coplanarity for an (LF)BGA package is larger than for a QFP with a comparable amount of input and outputs. The maximum coplanarity tolerance for BGAs is 150 µm, and for QFPs is 80 µm. At a non-coplanarity of 150 µm, some termination balls of the BGA will not touch the solder paste, even if a 200 µm thick stencil is used. This is because the ball will be pressed into the paste to a depth of only 80 µm.

5.2 Reflow soldering

The peak temperature for reflow soldering should remain below 230 °C (typically the peak temperature is between 200 and 205 °C) and the dwell time above 183 °C should not exceed 70 seconds, with a preference for temperatures at the higher ends to permit good wetting and ball shear.

If any moisture is present in the plastic package during soldering, it may turn into steam and expand rapidly. Under certain circumstances, the force exerted by this expansion can cause internal delamination or, in the more severe cases, may result in internal or external crack (known as the popcorn effect).

To avoid this problem, components should not be removed from their drypack longer than is specified on the box label.

The reflow soldering profile is shown in Fig.3. Although the minimum peak temperature is defined as 205 °C, to enlarge the process window and to be less sensitive to oven temperatures, in practice the minimum peak temperature of 215 °C is often defined.

6 REWORK

Although (LF)BGA assembly yields are very high, there may still be a requirement for component rework. By rework, we mean the process of removing the component from the pcb and replacing it with a new component. If an (LF)BGA is removed from a pcb, the solder balls of the component are deformed drastically so the removed (LF)BGA has to be discarded.

6.1 Device removal

As is the case with any component, it is essential when removing an (LF)BGA that the board, tracks, solder lands or surrounding components are not damaged. To remove an (LF)BGA, the board must be uniformly heated to a temperature close to the reflow soldering temperature. A uniform temperature reduces the chance of warping the pcb.

To do this we recommend that the board is heated until it is certain that all the joints are molten. Then carefully pull the component off the board with a vacuum nozzle.

6.2 Site separation

When the component has been removed, the vacant I site must then be cleaned before replacing the (LF)BGA. Removing an IC often leaves varying amounts of solder on the mounting lands. This excessive solder can be removed with either a solder sucker or solder wick. The remaining flux can be removed with a brush and cleaning agent. It is recommended that both sides of the board are cleaned to ensure maximum success.

After the board is properly cleaned and inspected, apply flux on the solder land and on the connection balls of the (LF)BGA. Do not apply solder paste as this has shown to result in problems during re-soldering.

6.3 Device replacement

The last step in the repair process is to solder the new component on the board. Ideally, the (LF)BGA should be aligned under a microscope or magnifying glass. If this is not possible, try to align the (LF)BGA with any board markers.

To reflow the solder, apply a temperature profile that is as close as possible to the profile shown in Fig.4. So as not to damage neighbouring components, it may be necessary to reduce some temperatures and times.

7 (LF)BGA FOOTPRINTS

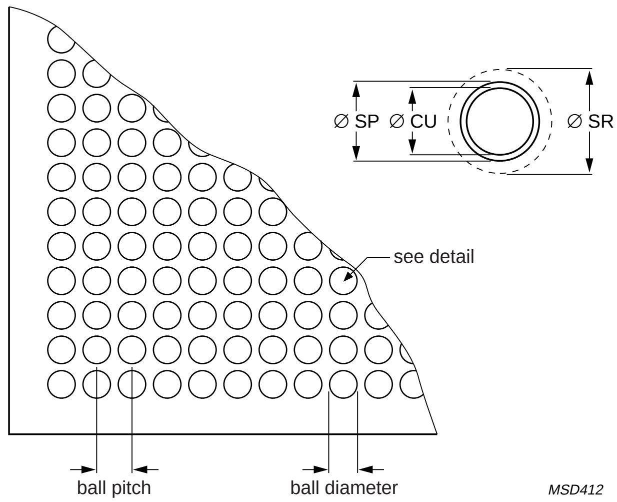


Fig.8 Generic footprint diagram of an (LF)BGA.

7.1 Ball pitch 0.50 mm, ball diameter 0.32 mm

See Fig.8.

Occupied area:

- package outline + 0.2 mm

Dimensions:

- copper ØCU = 0.275 mm
- solder paste ØSP = 0.300 mm
- solder resist ØSR = 0.425 mm

Clearance between solder lands should be 0.15 mm

The packages that fit this footprint are shown in Table 1. For an exact ball layout of the IC packages, refer to the relevant drawing in Data Handbook IC26: *Integrated Circuit Packages.*

Table 1

PACKAGE NAME	PHILIPS OUTLINE CODE	OUTLINE DIMENSIONS
LFBGA20	SOT479-1	3.32 × 4.45 × 1.05
LFBGA32	SOT478-1	4.5 × 4.5 × 1.05
LFBGA48	SOT488-1	5.5 × 5.5 × 1.05
LFBGA56	SOT516-1	6 × 6 × 1.05
LFBGA84	SOT518-1	7 × 7 × 1.05
LFBGA56	SOT542-1	6 × 6 × 0.8
LFBGA64	SOT543-1	6 × 6 × 0.8
LFBGA80	SOT557-1	7 × 7 × 0.8

7.2 Ball pitch 0.80 mm, ball diameter 0.46 mm / 0.40 mm

See Fig.8.

Occupied area:

- package outline + 0.3 mm

Dimensions:

- copper ØCU = 0.425 mm
- solder paste ØSP = 0.425 mm
- solder resist ØSR = 0.575 mm

Clearance between solder lands should be 0.15 mm

The packages that fit this footprint are shown in Table 2. For an exact ball layout of the IC packages, refer to the relevant drawing in Data Handbook IC26: *Integrated Circuit Packages*.

Table 2

PACKAGE NAME	PHILIPS OUTLINE CODE	OUTLINE DIMENSIONS
LFBGA96	SOT536-1	13.5 × 5.5 × 1.05
LFBGA114	SOT537-1	16 × 5.5 × 1.05
LFBGA144	SOT512-1	10 × 10 × 1.05

7.3 Ball pitch 1.00 mm, ball diameter 0.50 mm

See Fig.8.

Occupied area:

- package outline + 0.3 mm

Dimensions:

- copper ØCU = 0.45 mm
- solder paste ØSP = 0.45 mm
- solder resist ØSR = 0.60 mm

Clearance between solder lands should be 0.15 mm

The package that fit this footprint are shown in Table 3. For an exact ball layout of the IC packages, refer to the relevant drawing in Data Handbook IC26: *Integrated Circuit Packages*.

Table 3

PACKAGE NAME	PHILIPS OUTLINE CODE	OUTLINE DIMENSIONS
BGA256	SOT466-1	27 × 27 × 1.75

7.4 Ball pitch 1.27 mm, ball diameter 0.75 mm

See Fig.8.

Occupied area:

- package outline + 0.3 mm

Dimensions:

- copper ØCU = 0.60 mm
- solder paste ØSP = 0.60 mm
- solder resist ØSR = 0.75 mm

Clearance between solder lands should be 0.15 mm

The packages that fit this footprint are shown in Table 4. For an exact ball layout of the IC packages, refer to the relevant drawing in Data Handbook IC26: *Integrated Circuit Packages*.

Table 4

PACKAGE NAME	PHILIPS OUTLINE CODE	OUTLINE DIMENSIONS
BGA256	SOT466-1	27 × 27 × 1.75
BGA256	SOT471-1	27 × 27 × 1.55
BGA292	SOT489-1	27 × 27 × 1.75
BGA304	SOT550-1	31 × 31 × 1.75
BGA316	SOT531-1	27 × 27 × 1.75
BGA388	SOT532-1	35 × 35 × 1.75
BGA492	SOT514-1	35 × 35 × 1.75