

CSPs and other fine-pitch area array packages are seldom the only components on a printed board assembly, so it is important to determine a balance in the selection of stencil thickness. Thin stencils, appropriate for optimal CSP printing, will likely not provide sufficient solder paste for other, larger-pitch BGAs. This can lead to defects such as HoP or reduce thermal fatigue life of critical ASICs.

When making decisions about stencil thickness and paste volume to be deposited, it is safer to deposit more paste than less, with a goal to achieve six times more bridges than opens (ratio and not more defects). The reasons are that bridges can rarely escape inspection and they are sure to be caught by tests (ICT or FT). Opens and intermittent opens, however, can easily escape tests because they may make contact due to pressure applied during testing, but they will result in field failures. This guideline to target more bridges than opens while designing stencil and land patterns is the most cost-effective method to prevent field failures.

It is important to design a stencil aperture that will provide good paste release. To ensure good paste release, an aspect ratio of 1.5 is recommended. Aspect ratio is the ratio between stencil aperture width and stencil thickness. Another commonly used measure is called area ratio. An area ratio > 0.66 is recommended. Designing stencils conforming to these strongly recommended aspect and area ratios will allow more complete paste release and lessen the tendency for paste particles to cling to the aperture walls. Stencil designs violating ratio guidelines risk accumulation of nonprinted solder particles in stencil apertures. In severe cases of this occurrence, in which apertures clog, the reduced paste volume deposited on pads may result in deficiently formed or completely non-formed solder joints after full assembly.

The formula to calculate area ratio is the area of the stencil aperture opening divided by the area of the aperture walls (see Figure 7-1).

Area land contact/area stencil wall equals aperture/area of aperture walls:

$$L \times W / 2(L + W) \times T > 0.66$$

where:

L = aperture length

W = aperture width

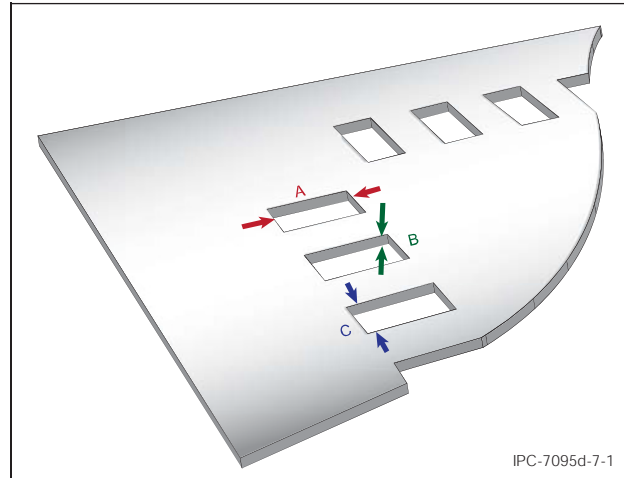
T = stencil thickness

**Note:** Aspect ratio is more common than area aspect ratio in aperture design since aspect ratio is one dimensional simplification of the area aspect ratio when land length is much greater than width. For example, a square aperture with a dimension of 0.35 mm with a stencil thickness of 0.125 mm gives the resulting aspect ratio:

$$\frac{0.35 \times 0.35}{2(0.35 + 0.35) \times 0.125} = \frac{0.1225}{0.175} = 0.7$$

Using an overprint or an aperture larger than the land may be necessary to achieve this ratio, especially for finer-pitch BGAs. Stencil apertures are generally one-to-one or slightly smaller than the land to achieve gasketing. It may be necessary to clean the stencil after each print if the stencil apertures need to be larger than the lands to achieve the desired area ratio for better paste release. Larger apertures may cause paste to leak under the stencil, which can result in paste smudging on subsequent printed boards. It is also important to note stencil apertures are significantly larger than the lands when designing stencils for CBGAs, CCGAs and paste-in-hole processes.

It also is recognized there are scenarios in which it may not be possible to design the stencil in compliance with the area ratio recommendation. Area ratios down to 0.5 can produce adequate prints using modern solder paste formulations and stencil technologies in combination with carefully implemented printing processes. Square or rectangular apertures can also be designed with rounded corners to promote paste transfer in such scenarios. Product vendors should be consulted to identify best materials to fulfill printing and assembly requirements. Table 7-4 explains the common stencil technologies and options currently available. See also IPC-7525.



**Figure 7-1 Stencil Aperture Labels for Aspect Ratio, Area Ratio Calculations (Solder Stencil Feature Dimensions)**

A – Aperture length (L)  
B – Stencil thickness (T)  
C – Aperture width (W)

**Table 7-4 Pros and Cons of Common Stencil Technologies and Options**

Stencil Manufacturing Technology	Pros	Cons
Chemically etched	Low cost if there are many thousands of apertures on the stencil	Can produce rough aperture walls
	Accommodates user-specified foil thickness	Typically limited to large aspect ratio and large aperture designs
	Oldest technology	Shrinking supplier base
Laser cut	Lowest cost stencil for low aperture count designs	Aperture cut quality depends on laser machine and foil material properties
	Cost effective for 10,000-aperture-count designs	Cost increase to manufacture as aperture count exceeds 10,000
	Most common stencil technology	Typically, available in only fixed foil thicknesses
Electroformed	Cost effective for > 10,000-aperture-count designs	Highest cost stencil for medium to low aperture count designs
	Smooth aperture walls	Typically requires longest lead time
	Accommodates user-specified foil thickness	Supplier availability limited
	3D stencil design compatible (see 7.1.1.4)	Stencil thickness uniformity
Stencil Option	Pros	Cons
Electropolish (chemically etched, laser-cut stencils)	Smooths rough aperture walls	Method can produce inconsistent results
	Permanent	May alter aperture shape and size
Nanocoating (all stencil technologies)	Reduces under-stencil wipe and clean frequency	Cannot smooth rough aperture walls
	May improve solder paste volume and height variation	Varied usage results, many proprietary materials and application methods available
	Minimal effect on aperture shape and size	Coatings may deteriorate with stencil use
Fine-grain foil (laser-cut stencils)	Smoother aperture walls and cleaner raw cut finish	Difficult to know if actual cutting result reflects optimum

**7.1.1.3 Fine-Pitch Printing Technology** Increasing functionality and decreasing package size is resulting in continuously shrinking BGA pitch. A 0.35-mm-pitch BGA is the smallest that can be relatively easily used for SMT with a standard laser-cut stencil and type 4 solder paste. As package pitch is reduced to  $\leq 0.3$  mm, there will be new challenges to the assembly process. Printing processes become more critical as the gap between printed features and apertures decreases. Stencils to accommodate such small-scale printing may include aperture area ratios designed  $< 0.66$ , which may demand use of higher-quality stencils, solder paste material and printing equipment to achieve acceptable results.

There are a few methods to address fine-pitch printing. The most common is to use advanced stencil materials like fine-grain foil and stencils with a nanocoating finish (see Table 7-5). Some nanocoating types are designed to coat the stencil aperture walls, producing a smooth wall finish which can help to reduce solder paste adhesion. It will help to release the paste from the stencil walls and increase the stencil transfer efficiency.

There are many coating formulations which can provide various printing qualities. Also, nanocoating application methods can yield different results (i.e., nanocoating by spraying and bake vs. nanocoating by manual wipe-on). Some studies show improved transfer efficiency (as low as 0.5 area ratio); however, it is recommended to evaluate the nanocoating material and process of the stencil.

Using solder paste with finer-powder size type 5 or type 6 is another option for paste printing for fine-pitch BGAs when using a standard laser-cut stencil with challenging-to-print aperture designs. This will improve the chance to successfully print lower area ratios, potentially as low as 0.5, but paste cost will be higher and paste availability might be limited.

Maintaining compliant area ratio stencil designs can also be achieved by reducing the stencil thickness. For example, a 175- $\mu\text{m}$  [0.007-in] aperture on a 75- $\mu\text{m}$  [0.003-in] stencil has a 0.58 area ratio, which requires an advanced stencil material (e.g., nanocoating) to print successfully. The area ratio will increase to nearly 0.7 if the stencil thickness is reduced to 63.5  $\mu\text{m}$  [0.0025 in]. When reducing stencil thickness, it is necessary to consider print volume requirements for the rest of the components on the assembly because leaner solder joints may negatively impact reliability. If a keep-out zone is possible, a step stencil could be used.

BGA paste/flux dipping is another option of assembly of fine-pitch BGAs. In this option, the paste/flux dipping process removes the challenge of fine-pitch paste printing. Figure 7-2 shows BGA balls covered with paste after being dipped into a paste reservoir.

The paste in the pick-and-place reservoir is not a typical type of printing paste. It is a special dipping paste, usually type 5 with reduced metal loading. Dipping can be a good option for small BGA packages with low warpage. Paste/flux dipping parameter optimization such as paste/flux coverage and dwell time is necessary.



**Figure 7-2 BGA Balls After Paste Dipping**  
(Figure source: Indium Corporation)

Table 7-5 shows a summary of fine-pitch printing options.

**Table 7-5 Fine-Pitch BGA (FBGA) Printing Options**

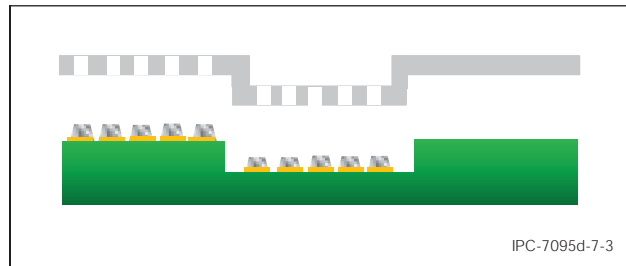
Fine-Pitch Printing Option	Pros	Cons
Advanced stencil	Accommodates challenging low-area-ratio aperture designs	Many nanocoating formulations
Decreased paste type	Accommodates challenging low-area-ratio aperture designs	Higher solder paste cost
Decreased stencil thickness	Raises aperture area ratio values	Reduces printed paste volume
BGA dipping process	Eliminates stencil printing	Limited to small BGAs with low warpage

**7.1.1.4 Cavity Printing (3D Stencil)** Cavity technology (also known as reservoir print) can be an effective solution to reduce total thickness (z height) of the printed board assembly.

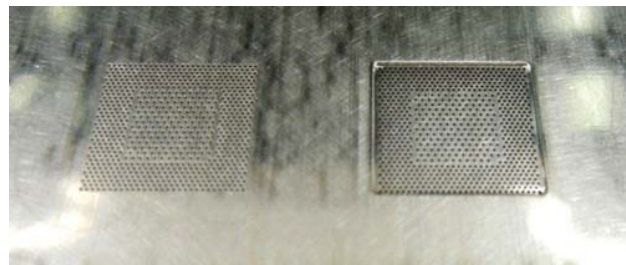
In this approach, the printed board has a cavity in which components need to be mounted on two levels of the board. Figure 7-3 shows a printed board with a cavity. A special stencil with a pocket matching the cavity is used to print the cavity on the two levels of the printed board. Two types of 3D stencils are used for this:

- 1) 3D electroformed
- 2) 3D welded

Figure 7-4 shows a picture of a 3D-electroformed stencil with two BGA cavity pockets.



**Figure 7-3 Cavity Board and 3D Stencil**



**Figure 7-4 3D Stencil with Two Cavity Pockets**

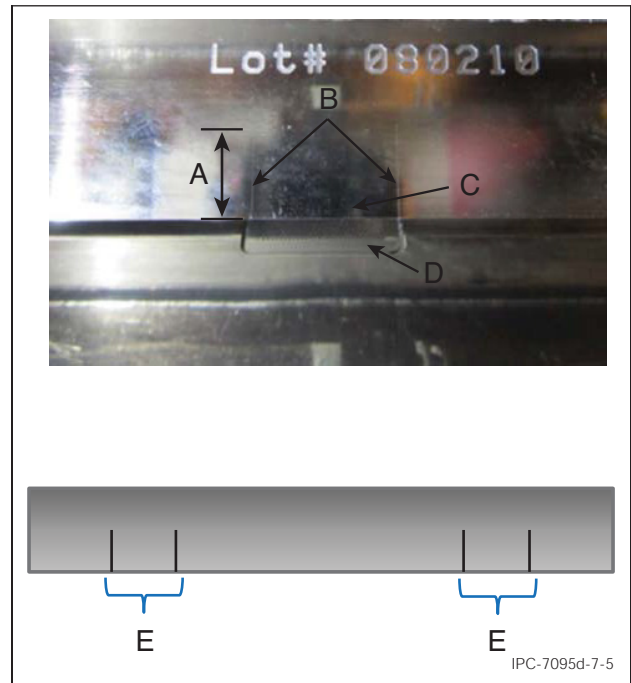
A special squeegee is needed to successfully print inside the cavity. A slit-metal squeegee with a 10-mm to 15-mm slit (See Figure 7-5) can be designed, in which the slit area deflects at a different angle when it goes into the cavity during printing. It is important to adjust the lateral position of the squeegee on the machine to the exact location of the cavity during machine setup.

**7.1.1.4.1 Cavity Print Keep-Out Zone** As a general design guide, for a simple 3D stencil (traditional step-down stencil), the distance from the step edge to the nearest aperture (Z1 in Figure 7-6) should be 0.9 mm [35.4 mil] for every 0.025 mm [0.98 mil] of step-down thickness (see also IPC-7525). When a 3D cavity print stencil is needed for reaching into deeper recesses, the design guide for Z1 is the cavity depth plus 1.27 mm [50 mils], including the stencil fabrication sidewall slope, printed board wall tolerance and alignment of the squeegee.

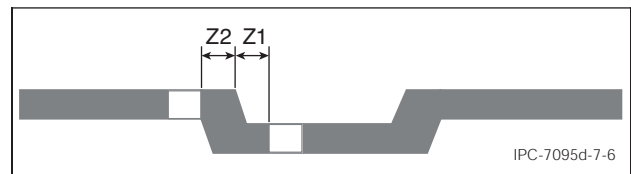
Z2 in Figure 7-6 is the distance between the aperture and the cavity on the flat level. Usually this keep-out zone is less conservative than Z1; at least 150 μm [6 mils] will be needed, plus the foil thickness.

Aside from the keep-out zone required for printing, the designer needs to consider the package outline size and the distance between the outer BGA balls and the package edge. The designer also needs to take into consideration the pick-and-place placement tolerance and rework and underfill space required. This may increase the required cavity keep-out zone.

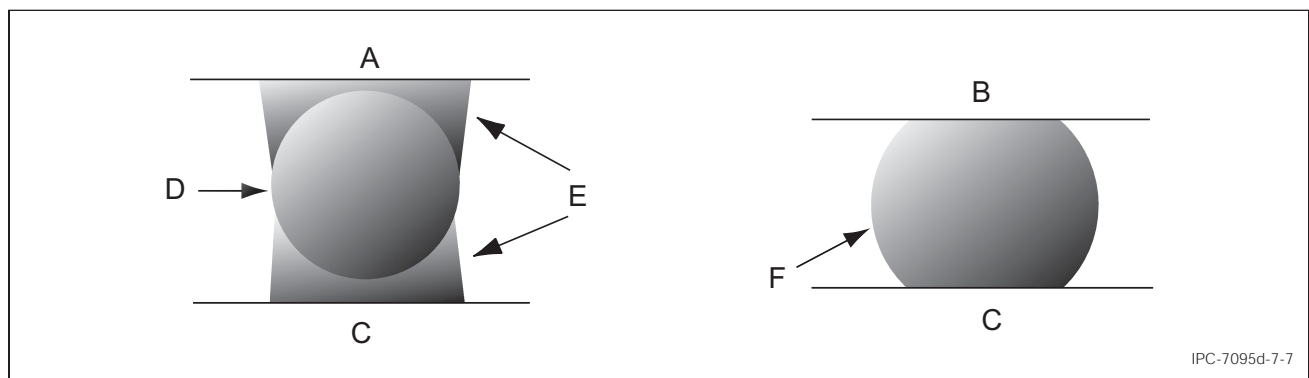
**7.1.1.5 Importance of Paste Volume** Much of the solder volume for plastic BGAs is supplied by the solder ball on the part itself, but paste volume is not that critical. For plastic BGAs (in both SnPb and Pb-free configurations) > 0.8-mm pitch, stencil thickness will be dictated by the other component types used on the printed board assembly. Solder volume and stencil thickness become more critical for ceramic BGAs with noncollapsible Cu balls or high-temperature PbSn10 balls with a melting point of 302 °C. Solder balls used on ceramic BGAs do not collapse during the normal reflow process (see Figure 7-7).



**Figure 7-5 Slit-Metal Squeegee**  
 A – Flap length  
 B – Slits  
 C – Flap  
 D – Pocket  
 E – Cavity area



**Figure 7-6 Cavity Keep-Out Zone**



**Figure 7-7 High-Pb and Eutectic Solder Ball and Joint Comparison**  
 A – CBGA substrate  
 B – PBGA substrate  
 C – Printed board  
 D – PbSn10 ball  
 E – Eutectic SnPb  
 F – SnPb36Ag2 ball

For CBGAs with noncollapsible balls (Cu balls or PbSn10), which require thicker solder volume, a stepped stencil may be used. The step is typically 0.04 mm to 0.08 mm, and it can put two different paste thicknesses on the printed board surface. A metal squeegee can be used successfully when the step is 0.04 mm. If a stepped stencil is used, the step line should be at least 3.75 mm from any print aperture.

Because the high-Pb ball does not collapse, having sufficient solder paste is critical. In addition to using a stepped stencil, the stencil aperture may also have to be much larger than the land to achieve the required paste volume for noncollapsible balls. The fillet between the land and ball depend on the solder paste volume. CBGAs and CGAs require a minimum volume of solder paste to make a reliable solder joint. Suppliers of this type of technology can provide the specific solder paste volume for their package. Table 7-6 contains an example of the solder paste volume requirements for some ceramic packages using either SnPb or Pb-free (SAC) solder paste.

The paste volume requirements should be the same for both SnPb and Pb-free paste. In addition, paste volume requirements are the same for Cu balls or high-temperature PbSn10 balls. This is because neither ball will collapse during reflow, and all solder fillet requirement must be met by the paste.

Some CBGAs do not use noncollapsible balls but use Pb-free (SAC) solder balls, which collapse during reflow. This contributes to the formation of solder fillet just as SnPb balls do in PBGAs.

To achieve the correct solder paste volume for a ceramic package with noncollapsible balls (Cu or PbSn10), it is necessary to over-print the land with a larger aperture or to use a thicker stencil. For other components on a printed board with CBGAs, the aperture size may need to be adjusted to compensate for the thicker stencil, or it may be necessary to step the stencil. An example of correct volume for 1.27-mm-pitch BGAs and CGAs are 78,660,000  $\mu\text{m}^3$  [4,800  $\text{mil}^3$ ] and 114,700,000  $\mu\text{m}^3$  [7,000  $\text{mil}^3$ ].

**Table 7-6 Example of Solder Paste Volume Requirements for Ceramic Array Packages**

Module Type	Pitch (mm) [mil]	Assembly Alloy	Paste Volume Min. ( $\mu\text{m}^3$ ) [ $\text{mil}^3$ ]	Paste Volume Nom. ( $\mu\text{m}^3$ ) [ $\text{mil}^3$ ]	Paste Volume Max. ( $\mu\text{m}^3$ ) [ $\text{mil}^3$ ]
CBGA	1.27 [50]	SnPb	78,657,600 [4,800]	90,128,500 [5,500]	163,870,000 [10,000]
CBGA	1.0 [39.37]	SnPb	40,967,500 [2,500]	57,354,500 [3,500]	75,380,200 [4,600]
CCGA	1.27 [50]	SnPb	49,161,000 [3,000]	81,935,000 [5,000]	124,541,200 [7,600]
CCGA	1.0 [39.37]	SnPb	32,774,000 [2,000]	65,548,000 [4,000]	81,935,000 [5,000]
SAC CBGA	1.27 [50]	SAC	24,580,500 [1,500]	36,870,750 [2,250]	N/A
SAC CBGA	1.0 [39.37]	SAC	13,109,600 [800]	28,677,250 [1,750]	N/A
Cu CGA	1.0 [39.37]	SAC	57,354,500 [3,500]	81,935,000 [5,000]	106,515,500 [6,500]

**7.1.2 Component Placement Impact** Utilizing BGA technology requires some new assembly capabilities. Depending on the type of pick-and-place system, a change in package carrier mechanism may be required to transfer packages from matrix tray to the pick position. Fiducials may help vision systems recognize the exact location of the land pattern for the BGA, similar to what is used for fine-pitch peripheral-leaded parts. Large BGA parts on tape-and-reel will require 44-mm and 56-mm feeders, depending on the body size; however, most common feeders for BGAs are JEDEC trays.

**7.1.3 Vision Systems for BGA Placement** Placement accuracy is a very critical part of the BGA process. It is strongly recommended to not move a BGA after machine placement to correct placement problems. This may cause solder bridging in adjacent solder joints, because the connections cannot be seen visually. The placement machine's accuracy is largely dependent on the vision system and the ability of the nozzle to hold the component.

Matching the vision system to the application is also important. The vision system determines the x, y and theta offset of each component prior to placement. In addition to determining the component offset, the vision system can also inspect the component for dimensional integrity and missing solder balls. Charge-coupled device (CCD) camera-based systems employ two lighting methods, referred to as binary and gray scale. Both methods can be sensitive to contrast and lighting changes.

Grayscale systems use front lighting, which illuminates the component from below. Surface features are reflected into the CCD camera for processing. Binary systems use back-lighting, which illuminates the component from above. The outline of the component is projected into the CCD camera for processing. Binary imaging, which is the older of the two methods, locates a feature using the contrast between black and white images. Grayscale systems can usually interpret 256 levels of contrast. Both systems use an algorithm to determine the center of the component. Binary imaging requires less computing capability than grayscale imaging.

Grayscale imaging places BGA components based on ball location, while binary imaging places BGA components based on the component outline. In some cases, the tolerance between the BGA outline and the balls is significant. Grayscale imaging is more desirable for placing BGA components because it eliminates placement error due to variations in the component outline.

It is important to choose a placement machine nozzle with sufficient surface area to hold the part without any shifting during placement. The nozzle must gasket against the part, not allowing vacuum leakage. Tactile sensing, which helps control the z-axis (vertical) stroke of the spindle, is desirable because it prevents a component from being crushed between the vacuum nozzle and the substrate.

**7.1.3.1 Pick-and Place Machine Capability for Nonstandard-Grid BGA Components** This section describes the process for placement of area array components with solder balls onto nonstandard grids, which is also known as balls anywhere design (see 6.2.6). The area array component placement is done with the other components using a pick-and-place machine on the SMT manufacturing line.

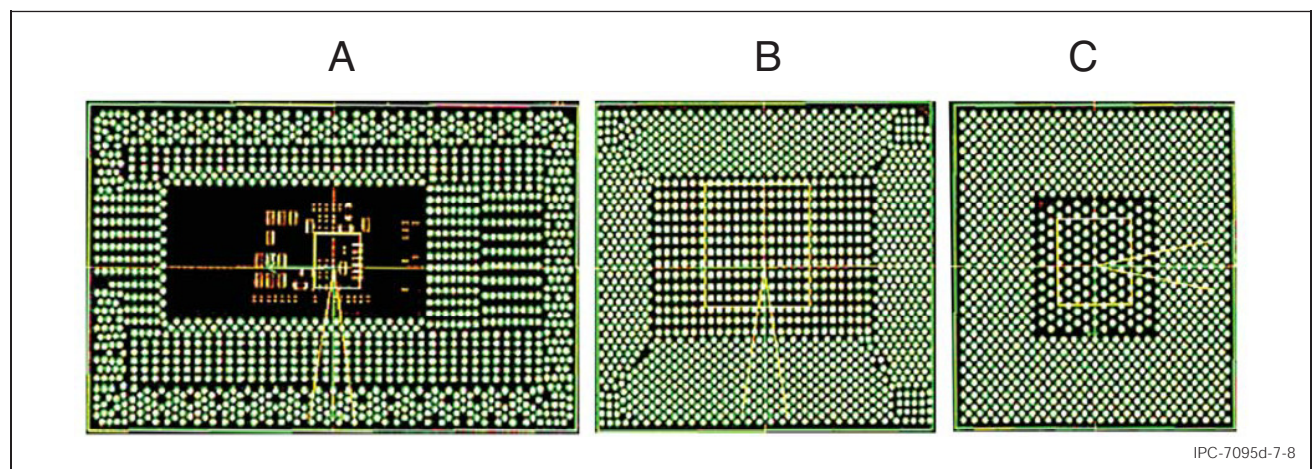
To enable the pick-and-place machine to place off-grid area array components, specifically BGAs, offline teaching and inline production need to be conducted (see 7.1.3.1.1 and 7.1.3.1.2).

**7.1.3.1.1 Offline Teaching** Before a part is fed into the machine for placement, a part description is created. For teaching, the camera used is the same as that for inline production. The part description is taught using a high-resolution camera. This description provides details of:

- Exact length, width and pitch between each ball
- Size of each ball
- Precise location of each ball and topography of the part, including pin 1 marks

An additional method is to create a part description by importing all the actual x-y locations of each ball and provide the above settings for the camera to inspect all balls present at respected locations.

Figure 7-8 shows an image for three different balls anywhere array designs and how the part description is taught on the offline teaching station.



**Figure 7-8 Balls Anywhere Image Captures for Offline Teaching**

A – Component A  
B – Component B  
C – Component C

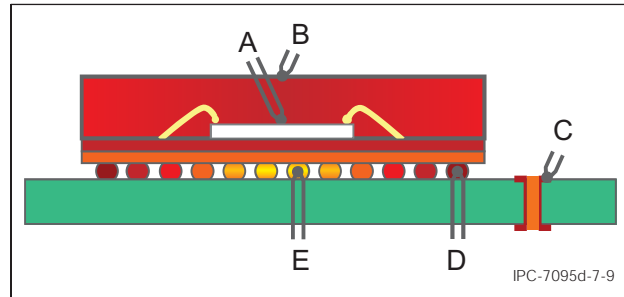
**7.1.3.1.2 Inline Production** During production, the nozzle on the pick-and-place head will pick up the part and use the camera to superimpose the image and details provided in the part description. If the image does not match in any form with the part description (e.g., missing balls, smashed balls or an irregular pitch between two balls), the camera will highlight a vision error and pinpoint its location. This vision error can then be viewed and confirmed with a visual aid. If there is no error, the equipment will bring the part to the x-y location on the board and place it at the provided location.

Usually during the first printed board of every build, the operator will monitor all BGA components, as well as other large components (e.g., shields and sockets) during placement. Before the head places the part on the location, a downward-looking camera will provide an image of where the part is to be placed on the board. This image will be superimposed with

the image of the component, so the operator can look specifically at the balls that are matched and those that are not. If required, the operator can move the part accordingly. This matching occurs in real time.

Once this process is completed, the operator saves the settings and then part will be placed exactly in the same way for all the placements during production.

**7.1.4 Reflow Soldering and Profiling** Reflow soldering is a complex process with many variables. All mass reflow systems incorporate convective, conductive and radiant means of heat transfer; to what degree depends on the design of the reflow system. The solder balls under a BGA component must be primarily heated by conductive heating through the interconnect substrate. Figure 7-9 illustrates the reason for this. The optimal reflow profile (temperature vs. time), including the peak temperature, varies with the specific package and the overall printed board assembly. However, as a guideline, when using SAC305 solder paste for BGA packages with SAC305 solder balls, the minimum peak temperature for moderately complex assemblies may need to be  $\geq \sim 240$  °C. When using SnPb eutectic solder paste for BGA packages with SAC305 solder balls, the minimum peak temperature in most cases can be as low as  $\sim 215$  °C. See Table 7-7 for recommendations for proper reflow temperatures. See also IPC-7530 for complete details on reflow profiling.



**Figure 7-9 Examples of Peak Reflow Temperatures at Various Locations at or Near a BGA**

A	– 240 °C (Pb-free solders) / 220 °C (SnPb solders)
B	– 245 °C (Pb-free solders) / 225 °C (SnPb solders)
C	– 260 °C (Pb-free solders) / 230 °C (SnPb solders)
D	– 235 °C (Pb-free solders) / 217 °C (SnPb solders)
E	– 230 °C (Pb-free solders) / 205 °C (SnPb solders)

**7.1.4.1 Forced-Gas Convection** Forced-gas convection is a noncontact heating method, with some heating accomplished by radiation. Heat is transferred to the printed board assembly by low-velocity heated gas. The rate at which heat is transferred to the object is directly proportional to the difference in temperature between the heated gas and the assembly.

**7.1.4.2 Reflow Atmosphere** Reflow furnace atmosphere will affect solder wetting. Reflowing in a nitrogen environment will result in improved wetting of the solder joint. This will sometimes allow compensation for marginally wettable surfaces on the printed boards as well as oxidized solder balls. To obtain the maximum benefit of the solder paste in a nitrogen environment, it is important to monitor the oxygen content and control it within the limits established for the process.

Solderability of Pb-free solder pastes is not as good as that of SnPb. It is also possible to use up the activators in the solder paste to clean the oxides off the paste, land and balls prior to reflow if paste selection is not made correctly. This would result in a nonuniform-shaped solder joint and possibly nonwetting of the land.

Some solder paste formulations may also be affected by reflow atmosphere in the formation of voids. When using Pb-free processes with OSP surface finishes, it may be desirable to reflow the product using nitrogen, especially for hole fill problems in mixed assemblies. Hole fill problems can be encountered in Pb-free processes, especially when using no-clean flux and first- or second-generation OSPs. Use of water-soluble aggressive fluxes and third-generation OSPs should help minimize hole fill problems.

Reflowing OSP in nitrogen may also preserve the wettability of the lands for downstream processes as well as minimize oxidation of the test points or vias which may have an impact at ICT.

**7.1.4.3 Time/Temperature Profiles** The solder profile, also known as thermal profile, is one key variable in the manufacturing process that significantly impacts product yield. Conveyor speed and panel temperatures are two variables in solder profile development. The solder profile is not only product specific, it is also flux dependent. Different pastes require different profiles for optimum performance, so it is important to consult the paste manufacturer before developing the solder profile. Table 7-7 provides key reflow profiles for SnPb and Pb-free printed board assemblies and for mixed assemblies (backward- and forward-compatibility profiles). Note that profiles for Pb-free and forward-compatibility assemblies are the same.

With SnPb, there is general industry consensus about the composition of solder to be used: eutectic SnPb37 with a melting point of 183 °C. With this composition, there was a big difference between the melting point (183 °C) and peak temperature (220 °C). Even though it has been the recommended practice to maintain the temperature between 210 °C and 220 °C

**Table 7-7 Profile Comparison Between SnPb and SAC Alloys**

Profile Topic	SnPb Alloy Profile	Mixed- / Backward-Compatibility Profile	Pb-Free Alloy (SAC305) / Forward-Compatibility Profile
Alloy solidus temperature	183 °C	183 °C / 220 °C	217 °C to 220 °C
Target alloy peak temp range	210 °C to 220 °C	228 °C to 232 °C	235 °C to 245 °C
Absolute minimum peak reflow temperature **	205 °C	228 °C	230 °C
Component ramp-up rate	2 °C to 4 °C / second *	2 °C to 4 °C / second *	2 °C to 4 °C / second *
Component ramp-down rate	2 °C to 6 °C / second *	2 °C to 6 °C / second *	2 °C to 6 °C / second *
Soak or preheat activation temperature	100 °C to 180 °C *	100 °C to 180 °C *	140 °C to 220 °C *
Soak or preheat activation time	60 to 120 seconds *	60 to 120 seconds *	60 to 150 seconds *
Dwell time above liquidus	60 to 90 seconds	60 to 90 seconds	60 to 90 seconds
Dwell time at peak temperature	20 seconds max	20 seconds min	20 seconds max
Solder paste used	SnPb	SnPb	Pb-free (SAC305)
SMT component types	All SMT type SnPb and Pb-free <i>but not</i> Pb-free BGA Balls	All SMT-type SnPb and Pb-free <i>including</i> SAC Pb-free BGA balls	All components <i>including</i> BGAs are Pb-free <i>including</i> BGAs with SAC305 Pb-free BGA balls
Reason for peak temperatures	Pb-free surface finishes on BGA parts have no problem melting at 205 °C. All SnPb surface finishes have 90% Sn any way. Pb-free finishes have close to 100 % Sn with some other Pb-free elements like Bi.	A compromise temperature is needed so SnPb parts do not get overheated and Pb-free SAC BGAs with melting point of 220 °C can melt, collapse and fully mix with SnPb paste. Lower peak temperatures will cause SAC BGA balls to either not melt or partially melt and increase the incidence of HoP, opens and poor reliability	All components are Pb-free and can take higher heat. Too high a peak temperature may cause BGA ball drop (see 8.4.1.3), opens, dewetting and board warpage. Large BGAs are tested for maximum of 245 °C for MSL level rating.

\* Verify with the supplier

\*\* Coolest temperature on the board

across the entire printed board assembly, one could achieve good reflow soldering results by maintaining a temperature between 190 °C and 225 °C (a variation of almost 35 °C).

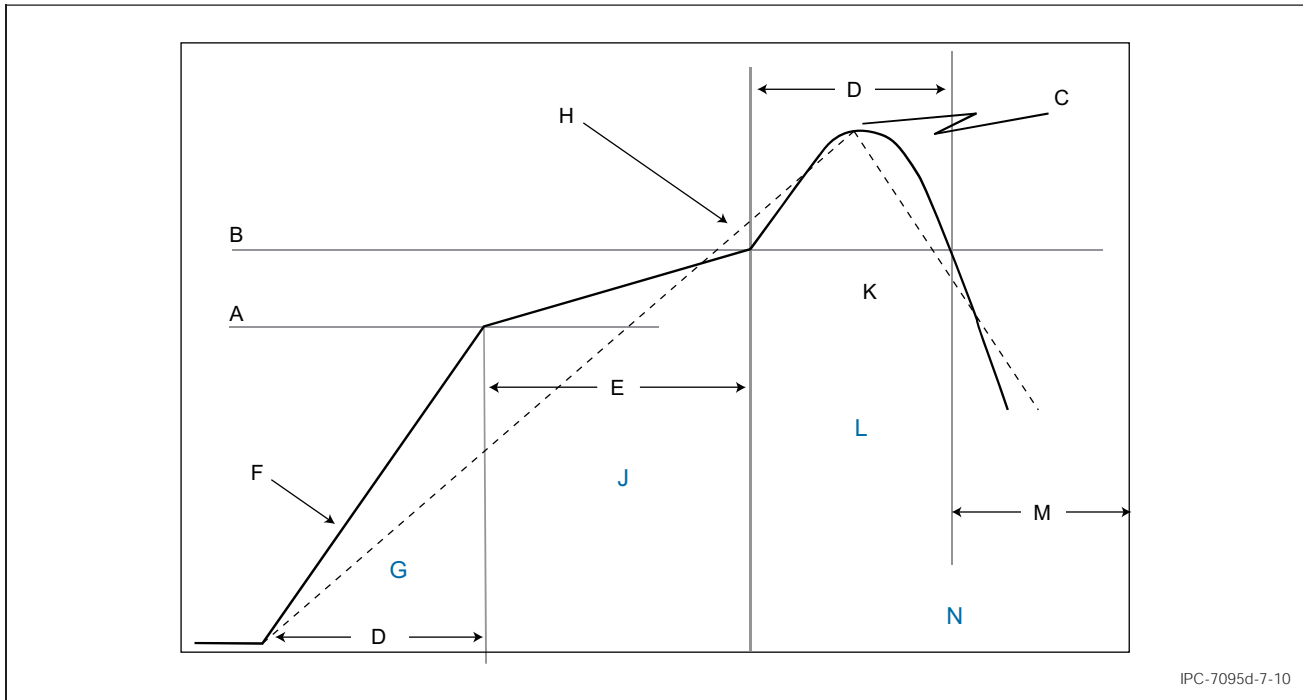
In Pb-free assembly, the commonly used SAC solders contain 3 % to 4 % Ag, 0.5 % to 0.7 % Cu and the rest Sn. These alloys have a melting point ~ 220 °C. A few components (e.g., some Al electrolytic capacitors) put restrictions on maximum temperature and duration above 230 °C to which they can be subjected. Additional constraints will be dictated by low-cost laminates, plastic connectors and moisture-sensitive components.

To accommodate such constraints, the peak temperature in Pb-free assemblies should be maintained between 230 °C and 245 °C, a variation of only 15 °C—a tight process window. This is a ~ 60 % drop from 35 °C variation within SnPb printed board assemblies as mentioned earlier in this section. The difficulty of achieving a reflow profile to meet the defined process window is further increased if large components with high thermal mass are used on the same printed board with smaller, temperature-sensitive components.

The reasons are simple. Large components with high thermal mass require a larger heat input to meet the process window requirements for peak temperature and time above liquidus (TAL). However, this large heat input may result in the smaller, temperature-sensitive components falling outside the process window requirements. To resolve this issue, very tight process control and narrow temperature bandwidth across the printed board are necessary. Many assembly houses may have a hard time meeting such requirements, especially on complex boards, without concerted time and effort in developing reflow profiles.

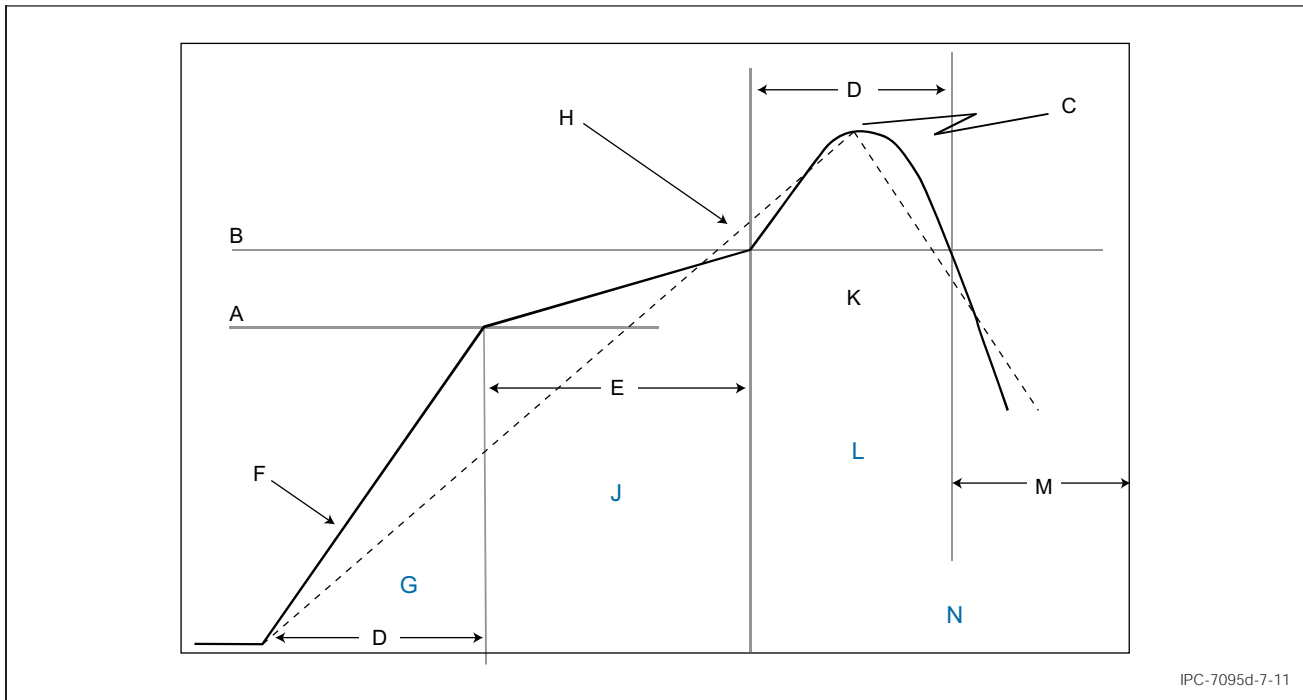
The problem can be further compounded by backward-compatibility issues in which some Pb-free components are used on a primarily SnPb printed board assembly. In such cases, the profile must accommodate both SnPb and Pb-free package requirements.

Figure 7-10 and Figure 7-11 show schematic profiles for printed board assemblies.



**Figure 7-10 Schematic of Reflow Profile for SnPb Assemblies**

- |                      |                                       |  |  |
|----------------------|---------------------------------------|--|--|
| A – 100 °C to 180 °C | E – 90 to 120 seconds                 | J – Soak                                   | M – 30 to 60 seconds (cooling to room temperature) |
| B – 183 °C           | F – Maximum slope of 5 °C/second      | K – Cool down (4 °C/second to 8 °C/second) | N – Cooling  |
| C – 210 °C to 220 °C | G – Preheat                           | L – Reflow                                 |  |
| D – 60 to 90 seconds | H – Ramp to peak profile with no soak |  |  |



**Figure 7-11 Schematic of Reflow Profile for Pb-Free Assemblies**

- |                      |                                       |  |  |
|----------------------|---------------------------------------|--|--|
| A – 140 °C to 220 °C | E – 90 to 120 seconds                 | J – Soak                                   | M – 30 to 60 seconds (cooling to room temperature) |
| B – 217 °C           | F – Maximum slope of 5 °C/second      | K – Cool down (4 °C/second to 8 °C/second) | N – Cooling  |
| C – 235 °C to 245 °C | G – Preheat                           | L – Reflow                                 |  |
| D – 60 to 90 seconds | H – Ramp to peak profile with no soak |  |  |

**7.1.4.4 Thermocouple Attachment** Figure 7-12 shows recommended locations of thermocouples on a printed board assembly. It is important that thermocouples be attached to small and large components at the solder joints. For BGAs, it is also important to attach a thermocouple to the top of the package.

It is very important to use the right thermocouple for developing any profile. Type K thermocouples with wire gauge of 36 American Wire Gauge should be used. Thicker thermocouple wires add too much heat sink. Thermocouple wire length should not exceed three feet for good accuracy. To ensure accuracy, thermocouple junctions must be welded. Do not twist, crimp or solder them.

Care should be exercised when using high-temperature tapes such as polyimide (Kapton) or Al. Tapes tend to come loose during reflow, and the system measures the temperature of the air in the oven and not the temperature of the solder joints.

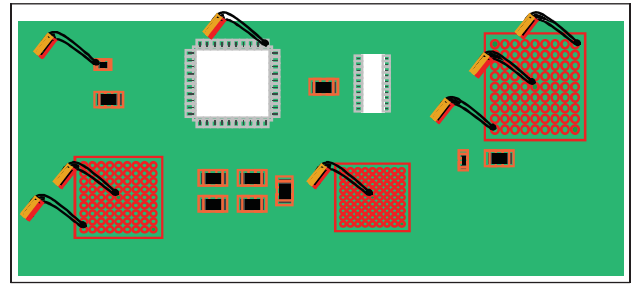
It is important to ensure there is good contact of the tape. Otherwise, a high-temperature solder or thermally conductive adhesive should be used to attach thermocouples to the solder joints. One benefit with using tape is that the thermocouples can be reused repeatedly without damaging them.

In the case of BGAs, drill holes in the center and corner balls of the BGA from the bottom of the printed board and push the thermocouples to the top to correctly measure the temperature of the BGA balls. It is important that the difference in temperature of the center and corner balls of the same BGA are within 2 °C of each other.

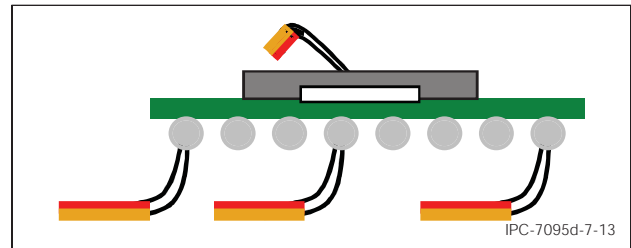
There are also some thermocouples that can be inserted under the BGA, eliminating the hole drilling process; however, in this instance, the thermocouples may only measure the temperature under the device.

Four to six thermocouples should be attached at various component locations to represent the lowest to highest thermal mass areas, including at least two thermocouples for BGAs. Figure 7-13 shows locations of thermocouples on a BGA.

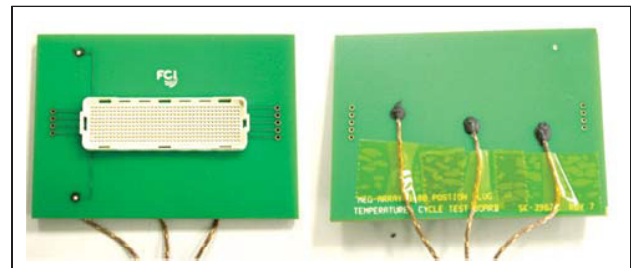
Proper thermal profiling of BGA connectors is critical to ensure proper solder attach. Because the connector substrate is typically made of a resin material, heat does not transfer through the body of the connector like a typical BGA active device. It is vital that, at a minimum, a thermocouple is placed in the center of the connector to verify proper temperatures are reached. Optimally, thermocouples will be placed in the corner and middle of the connector. Figure 7-14 shows a thermocouple connected to a solder ball interface.



**Figure 7-12 Locations of Thermocouples on a Printed Board Assembly with Large and Small Components-**



**Figure 7-13 Recommended Locations of Thermocouples on a BGA**



**Figure 7-14 Proper Thermocouple Location on a BGA Connector**

(Figure source: FCI USA, Inc.)

**7.1.4.5 Thermal Profiling for Backward-Compatibility with SnPb and Pb-Free Alloys** Developing a reflow profile is made difficult with backward-compatibility issues. Backward compatibility is a scenario in which some components are available only with Pb-free surface finishes and are used on a primarily Pb-free assembly. Such a scenario arises since it may not be economical for many component suppliers to supply both SnPb and Pb-free versions of the same component. It is not an issue when using Pb components such as small outline integrated circuits (SOICs), plastic-leaded chip carriers (PLCCs) or fine-pitch components with Pb-free surface finishes. Most SnPb components primarily have 85 % Sn surface finish with ~ 15 % Pb. When component manufacturers eliminated Pb from parts they plated with pure tin; however, solderability suffered. To correct this condition, up to 5 % Bi was added to the plating alloy to improve the wetting or solderability to the part.