APPLICATION NOTE

PCB Design Guidelines for BGA Rework

Ralph Webber, November 23, 2004

National Research Council Canada
Herzberg Institute of Astrophysics
Dominion Radio Astrophysical Observatory

P.O. Box 248, 717 White Lake Rd
Penticton, B.C., Canada
V2A 6K3
# Table of Contents

1. **INTRODUCTION** ............................................................................................................................................. 2  
2. **OVERVIEW** ........................................................................................................................................................ 3  
3. **DESIGN GUIDELINES FOR REWORK** ........................................................................................................... 4  
   3.1 PCB DESIGN ....................................................................................................................................................... 4  
      3.1.1 Land and Solder Mask ....................................................................................................................................... 5  
      3.1.2 Signal Line Space and Trace Width .................................................................................................................. 7  
      3.1.3 Routing and Vias ............................................................................................................................................... 8  
      3.1.4 Pad Surface Finish .......................................................................................................................................... 13  
      3.1.5 PCB Stack and Thermal Vias ................................................................................................................................. 14  
    3.2 MANUFACTURABILITY CONSIDERATIONS ........................................................................................................ 16  
       3.2.1 Board Design .................................................................................................................................................. 16  
       3.2.2 Process Considerations ................................................................................................................................... 19  
       3.2.3 Reflow Considerations ..................................................................................................................................... 19  
       3.2.4 Wave Solder Considerations ............................................................................................................................... 20  
       3.2.5 Inspection Considerations .................................................................................................................................. 20  
       3.2.6 Component Spacing Considerations .................................................................................................................. 21  
4. **BGA REWORK PREPARATION** ....................................................................................................................... 23  
   4.1 COMPONENT SENSITIVITY ........................................................................................................................................ 23  
   4.2 CIRCUIT BOARD SENSITIVITY .............................................................................................................................. 23  
   4.3 ESD SAFE WORK AREAS ........................................................................................................................................ 23  
   4.4 PROPER HANDLING AND STORAGE OF ESD SENSITIVE DEVICES AND PCBs ..................................................................... 23  
   4.5 PROPER PCB PREHEATING .................................................................................................................................. 24  
5. **BGA REWORK NOZZLE SELECTION PROCESS** ............................................................................................ 25  
   5.1 PRO-SERIES NOZZLES ........................................................................................................................................... 27  
      5.1.1 Pickup Tubes and Nests ...................................................................................................................................... 28  
   5.2 PRO-STEP NOZZLES ............................................................................................................................................... 30  
   5.3 MINI-SERIES NOZZLES .......................................................................................................................................... 30  
   5.4 SUB-PLENUM NOZZLES ....................................................................................................................................... 31  
   5.5 VERTICAL CONNECTOR NOZZLES .......................................................................................................................... 31  
   5.6 STRADDLE MOUNT CONNECTOR NOZZLES ....................................................................................................... 31  
6. **BGA REWORK PROFILE DEVELOPMENT PROCESS** .................................................................................. 32  
7. **BGA REWORK PART REMOVAL PROCESS** .................................................................................................. 34  
8. **BGA REWORK CLEANING PROCESS** .......................................................................................................... 34  
9. **BGA REWORK APPLYING SOLDER PROCESS** ........................................................................................... 35  
   9.1 DISPENSE ............................................................................................................................................................. 35  
   9.2 METAL STENCILS ................................................................................................................................................... 35  
   9.3 STENCILS FLEXIBLE REMOVABLE .......................................................................................................................... 36  
   9.4 SEMI-PERMANENT STENCILS .................................................................................................................................. 36  
10. **BGA REWORK PART PLACEMENT PROCESS** ........................................................................................ 36  
11. **BGA REWORK PART REFLOW PROCESS** .................................................................................................. 37  

Ralph Webber, November 23, 2004
List of Figures

Figure 3-1 NSMD and SMD Pads – Top View ........................................................................................................ 4
Figure 3-2 NSMD and SMD Pads – Cross Sectional View ................................................................................ 4
Figure 3-3 NSMD vs. SMD Lands Pads - Cross-Sectional View ..................................................................... 5
Figure 3-4 Solder Ball Area Stress Caused by Non-Optimized Package Pad/PCB Land Ratio ...................... 6
Figure 3-5 Trace Routing Space .......................................................................................................................... 7
Figure 3-6 Cross-Section of Different Via Types for Signal Transfer Through PCB ........................................ 8
Figure 3-7 Connection Between Vias, Via Capture Pads, Surface Lands and Stringers ................................ 9
Figure 3-8 Space Between Surface Land Pads ..................................................................................................... 9
Figure 3-9 Via Capture Pad Layout for Escape Routing .................................................................................. 10
Figure 3-10 Typical and Premium Via Capture Pad Sizes (in mils) ................................................................ 11
Figure 3-11 PCB Escape Routing for a 0.8mm BGA Pitch Using Laser-Drilled Blind Vias ......................... 13
Figure 3-12 Impact of Number of Thermal Vias vs. Die (Chip) Area .............................................................. 15
Figure 3-13 Impact of Number of 0.33-mm Diameter Thermal Vias Versus Die (Chip) Area ..................... 16
Figure 3-14 General Example of PCB Component Clearances .................................................................... 19
Figure 3-15 Component Spacing Specifications .............................................................................................. 21
Figure 3-16 Layout and Orientation for Multi-Connector Applications ........................................................... 22
Figure 3-17 Layout and Orientation of Different Sized Components .............................................................. 22
Figure 5-1 Recommended Clearances Between Components and Pro-Series Nozzles .................................. 27
Figure 5-2 Pro-Series Nozzle “Positive Air” Feature .................................................................................... 28
Figure 5-3 Custom Pro-Series Nozzle .............................................................................................................. 29
Figure 5-4 Pro-Step Nozzle ............................................................................................................................... 30
Figure 5-5 Mini-Series Nozzle Typical Configuration .................................................................................... 30
Figure 5-6 Sub-Plenum Nozzle .......................................................................................................................... 31
Figure 5-7 Straddle Mount Nozzle Assembly ................................................................................................ 32
Figure 6-1 Representative Profile of Hot Air BGA Rework System .............................................................. 33

Ralph Webber, November 23, 2004
List of Tables

Table 1 Number of Traces Routed Based on Space and Trace Line Width .................................................... 7
Table 2 Via Types for Signal Transfer Through PCB Layers ........................................................................... 8
Table 3 Formula for Via Layouts .................................................................................................................. 11
Table 4 Typical Via Capture Pad Sizes Used by PCB Vendors ....................................................................... 12
List of Abbreviations and Acronyms

BGA – Ball Grid Array. A chip package having solder balls on the underside for mounting. BGA allows for a reduction in die package size, better heat dissipation, and greater module densities.

ESD – Electrostatic Discharge. Electrostatic discharge is the transfer of electrons from one object to another.

PCB – Printed Circuit Board. Board that contains layers of circuitry that is used to connect components to a system.

QFP – Quad Flat Pack. A type of surface mount device with fine-pitch leads projecting from all four sides.

CSP – Chip Scale Package. An IC packaging technology in which solder balls take the place of pins, making the smallest package available. When heated, the solder balls alloy to matching pads on the circuit board.

Pb/Sn – Lead/Tin. Abbreviated term for solder.

TC – Thermocouple. Widely used as a type of temperature sensor.

SRT – Summit. Manufacturer of rework analysis equipment.

CTE – Coefficient of thermal expansion, relative to PCB thickness.

Definitions

Bottom Heater – The assembly that distributes heated air across the bottom of the board. Located inside of the X-Y table.

Liquidus – The temperature at which solder is completely melted under normal ambient conditions.

Pickup Tube – A long hollow metal tube in the heater assembly that contacts the component, applying a vacuum for lifting the component.

Process Profile – A combination of temperature setpoints, ramp rates and dwell times used to control the top heater and the bottom heater during the preheat, reflow, additional reflow, post reflow and solidify phases of a rework process.

X-Y Table – The assembly that supports the PCB and contains the bottom heater.

Ralph Webber, November 23, 2004
1 Introduction

The purpose of this document is to provide basic component design guidelines for PCB’s, subsequent rework and to explain the techniques of reworking BGA components using a manual rework system with autoprofiling capability.


An explanation on autoprofiling is contained in the document AutoProfiling Rework Cycles with Summit 1100 [3] which has been referenced to throughout this document. Nozzle types used in a Summit 110 rework station can be found in [4] and explain their different variations and applications for use in rework applications.

This document outlines the methods of rework preparation and the profile development of BGA components. An explanation of BGA component removal is provided along with the cleaning, re-application of solder, part placement and the BGA part reflow process.

Autoprofiling using the Summit 1100 rework station is reviewed in detail along with the necessary temperature profiling techniques required. A brief section on BGA inspection using X-ray techniques has also been included.
2 Overview

An overview of necessary PCB design considerations required when laying out boards for the intent of rework is covered along with PCB thermal rework profiles and the heating process that is used to create them.

Generating thermal profiles for rework is more difficult than generating profiles for a reflow oven. When an assembly passes through a reflow oven, the entire mass is heated to a relatively uniform temperature. Belt speeds and zone temperatures can be adjusted to achieve the desired thermal profile.

Heating with a rework analysis station eating is localized to the rework site. That makes profiling more difficult, because the thermal characteristics of each component and assembly can vary considerably. If the circuit board has multiple BGA component locations that require profiles, and all the BGA components are the same type and size, only one rework profile need be used.

During rework, the board is heated from both the bottom and the top. Bottom heating brings the entire board to an intermediate temperature. This minimizes stress from local heating and reduces extraction of heat from the rework site. However, because the entire board cannot be heated to reflow temperatures some heat will transfer from the rework site to adjacent areas.

As a component is top heated thermal energy dissipates through the board. The mass of the component and the rate of dissipation determine how much energy is needed to rework the board.

In a convection rework system such as a Summit 1100, most of the heating for BGA devices comes from the transfer of energy from the hot gas to the topside of the component. The heat then flows through the component body to the component-solder interface, through the solder, through the board-solder interface, and into the board. Copper conductors within the board, especially large power and ground planes, provide a substantial thermal path to draw heat away from the rework site.

This procedure automatically creates a thermal profile based on user-defined target parameters. These may include; board temperature, soak duration, BGA package temperature, BGA package ramp rate, solder joint temperature, and time over reflow.

A detailed explanation of BGA rework procedures and design guidelines follow along with rework preparation details, rework nozzle selection and applications, BGA autoprofiling and X-ray analysis.

Ralph Webber, November 23, 2004
3 Design Guidelines for Rework

3.1 PCB Design

The primary board design considerations include metal-pad sizes and associated solder-mask openings. PCB pads/land patterns, which are used for surface mount assembly, can be:

- Non-solder mask defined (NSMD) - The metal pad on the PCB (to which a package BGA solder ball is attached) is smaller than the solder mask opening.
- Solder mask defined (SMD) - The solder mask opening is smaller than the metal pad.

Figure 3-1 and Figure 3-2 illustrate the metal-pad and associated solder-mask openings.

The most common PCB material sets on which assembly can be performed are:

- Standard epoxy glass substrate
- FR-4
- BT (bismaleimide triazine)

Ralph Webber, November 23, 2004
The mechanical properties of the PCB, such as its CTE, can be affected by the number of metal layers, laminate materials, trace density, operating environment, site population density, and other considerations.

The more flexible, thinner PCBs consequently show greater reliability during thermal cycling. The industry standard PCB thickness ranges from 0.4 mm to 2.3 mm.

### 3.1.1 Land and Solder Mask

The suggested PCB land pattern specification is IPC-SM-782. (Surface Mount Design and Land Pattern Standard)

The design of the PCB and the BGA itself is important in achieving good manufacturability and optimum reliability. When designing a PCB for fine-pitch BGA packages, the following factors should be considered:

- Surface land pad dimension
- Via capture pad layout and dimension
- Signal line space and trace width
- Number of PCB layers

Figure 3-3 shows the location of the package pad (A) and board lands (B).

![Figure 3-3 NSMD vs. SMD Lands Pads - Cross-Sectional View](image)

Figure 3-4 illustrates why the layout and dimensions of the package pads and the board lands are critical. Matching the diameters of the PCB pad to the package side BGA pad helps form a symmetrical interconnect, and prevents one end of the interconnect from exhibiting a higher stress condition than the other.
In fact, if the design of the PCB pad diameters are even slightly smaller than the package side BGA pad diameter, the joint stress on the PCB side is emphasized rather than on the typically weaker package BGA side.

The top view of Figure 3-4 shows a package pad that is larger than the PCB land. In this case, the solder ball is prone to crack prematurely at the PCB interface. In the middle view of Figure 3-4, the PCB land is larger than the package pad, which leads to cracks at the package surface.

In the bottom view of Figure 3-4, where the ratio is almost 1:1, the stresses are equalized and neither site is more susceptible to cracking than the other. This is the preferred design. Solder lands on the PCB are generally simple round pads. Solder lands are either SMD or non-solder-mask-defined NSMD.
3.1.2 Signal Line Space and Trace Width

Many of today’s circuit board layouts are based on a maximum 100-µm-conductor line width and 200-µm spacing. To route between 0.8-mm-pitch balls, given a clearance of roughly 380 µm between ball lands, only one signal can be routed between ball pads.

The ability to perform escape routing is determined by the width of the trace and the minimum space required between traces. This width is calculated by the following formula: 

\[ g = 39.37 - d \]

The number of traces that can be routed through this space is based on the permitted line trace and space widths. Use the formula to determine the total number of traces that can be routed through \( g \).

<table>
<thead>
<tr>
<th>Number of Traces</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>( G = [2 \times \text{space width}] + \text{trace width} )</td>
</tr>
<tr>
<td>2</td>
<td>( G = [3 \times \text{space width}] + [2 \times \text{trace width}] )</td>
</tr>
<tr>
<td>3</td>
<td>( G = [5 \times \text{space width}] + [3 \times \text{trace width}] )</td>
</tr>
</tbody>
</table>

Table 1 Number of Traces Routed Based on Space and Trace Line Width

By reducing the trace and space size, you can route more traces through \( g \), as shown in Figure 3-5. Increasing the number of traces reduces the required number of PCB layers and decreases the overall cost. On the other hand, as line width decreases, PCB cost may go up and quality may be sacrificed.

![Figure 3-5 Trace Routing Space](image)

Ralph Webber, November 23, 2004
3.1.3 Routing and Vias

3.1.3.1 High Density Routing Techniques

Conventionally, pads are connected by wide copper traces to other devices or to plated-through holes (PTH). As a rule, the mounting pads must be isolated from the PTH.

Placing the PTH interstitially to the land pads often achieves this isolation. As available BGA pitch space contracts, the space available for signal fan-out also decreases. This poses a challenge when designing with BGA packages; however, by using high-density routing, the PCB designer can minimize many of these design and manufacturing challenges.

Vias are actual holes drilled through a multi-layer PCB to provide electrical connections between various PCB layers. In SMT PCBs, vias provide layer-to-layer connections. In some cases microvias are filled with reinforcing material. Table 2 lists the different via types for PCB signal transferring.

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Through via</td>
<td>An interconnection between the top and the bottom layer of the PCB; through vias can also provide interconnections to inner PCB layers.</td>
</tr>
<tr>
<td>Blind via</td>
<td>An interconnection from the top or bottom layer to an inner PCB layer</td>
</tr>
<tr>
<td>Embedded via</td>
<td>An interconnection between any number of inner PCB layers</td>
</tr>
</tbody>
</table>

Table 2 Via Types for Signal Transfer Through PCB Layers

Figure 3-6 illustrates these vias.

Figure 3-6 Cross-Section of Different Via Types for Signal Transfer Through PCB

Ralph Webber, November 23, 2004
Although blind vias can be more expensive than through vias, overall costs can be reduced because signal traces can be routed under a blind via, which requires fewer PCB layers. Through vias do not permit signals to be routed through lower layers, which can increase overall costs by increasing the required number of PCB layers. However, PCBs built using only through-hole vias can be economical due to the reduced complexity in board manufacturing.

### 3.1.3.2 Stringers

Stringers are rectangular or square interconnect segments that electrically connect via capture pads and surface land pads. Figure 3-7 shows the connection between vias, via capture pads, surface land pads, and stringers.

![Figure 3-7 Connection Between Vias, Via Capture Pads, Surface Lands and Stringers](image)

Figure 3-7 Connection Between Vias, Via Capture Pads, Surface Lands and Stringers

Figure 3-8 shows the space available between surface land pads for a 0.40 mm (15.75 mil) BGA pad.

![Figure 3-8 Space Between Surface Land Pads](image)

Figure 3-8 Space Between Surface Land Pads

Ralph Webber, November 23, 2004
3.1.3.3 Via Capture Pad and Dimension

The size and layout of via capture pads affect the amount of space available for escape routing. In general, the layout of via capture pads can be:

- Horizontal with the surface land pads
- Diagonal to the surface land pads

Figure 3-9 shows both inline and diagonal layouts.

Consider the following factors when deciding to place the via capture pads diagonally or inline with the surface land pads:

- Diameter of the via capture pad
- Stringer length
- Clearance between via capture pad and surface land pad

Use the information shown in Figure 3-9 and Table 3 to determine the PCB layout. If your PCB design guidelines do not conform to either equation in Table 3, contact your PCB supplier for assistance.

Ralph Webber, November 23, 2004
According to Table 3, you can place a larger via capture pad diagonally than horizontally with the surface land pads. Via capture pad size also affects how many traces can be routed on a PCB. Figure 3-10 shows sample layouts of typical and premium via capture pads.

Table 3 Formula for Via Layouts

<table>
<thead>
<tr>
<th>Layout</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Horizontally</td>
<td>$a + c + d = 0.6 \text{ mm} \ (23.62 \text{ mils})$</td>
</tr>
<tr>
<td>Diagonally</td>
<td>$a + c + d = 1.0 \text{ mm} \ (39.76 \text{ mils})$</td>
</tr>
</tbody>
</table>

Table 4 shows the typical and premium layout specifications used by most PCB vendors.

Ralph Webber, November 23, 2004
### 3.1.3.4 Via Density

Via density can be a limiting factor when designing high-density boards. Via density is defined as the number of vias in a particular board area. Using smaller vias increases the routability of the board by requiring less board space and increasing via density.

The microvia solves many of the problems associated with via density. Microvias are often created using a laser to penetrate the first few layers of dielectric. The layout designer can then route to the first internal board layer.

Typically, two layers (e.g., each 4 mil thick) can be laser-drilled, creating a 200-micron microvia diameter. In this case, routing to the first two internal layers is possible. In general, the number of PCB layers required to route signals is inversely proportional to the number of traces between vias (i.e., the greater the number of traces, the fewer the number of PCB layers required). You can estimate the number of layers your PCB requires by first determining the following:

- Trace and space size
- Number of traces routed between the via capture pads
- Type of vias used

Choosing the correct via type and using fewer than the maximum number of I/O pins can reduce the required number of layers. Placing the vias in the pad increases clearance. However, a standard via opening of 300 µm causes the solder to wick down into the via, and further causes weak or even open solder joints. In addition, the capture pad is larger than the solder pad.

Laser-drilled microvias can drill a hole of 100 µm in the board, which is reduced to 50 µm after plating. The resulting via-hole diameter is reduced to the point where the solder does not wick down the via.

<table>
<thead>
<tr>
<th>Specification</th>
<th>Typical (mils)</th>
<th>Premium (mils)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Trace/space width</td>
<td>5/5</td>
<td>3/3</td>
</tr>
<tr>
<td>Drilled hole diameter</td>
<td>12</td>
<td>10</td>
</tr>
<tr>
<td>Finished via diameter</td>
<td>8</td>
<td>5</td>
</tr>
<tr>
<td>Via capture pad</td>
<td>25.5</td>
<td>20</td>
</tr>
<tr>
<td>Aspect ratio</td>
<td>7:1</td>
<td>10:1</td>
</tr>
</tbody>
</table>

Table 4 Typical Via Capture Pad Sizes Used by PCB Vendors
A potential drawback to the laser-drilled microvia approach is that after plating, vias that are too small can potentially trap air instead of being properly filled with the PCB dielectric material. Therefore, the risk of reliability issues is increased.

Figure 3-11 illustrates an example of escape routing for a 0.8 mm BGA pitch using laser-drilled microvias. In this example, 0.15mm trace lines and spaces allow escape routing of the first two BGA signal rows through the top PCB layer. Because of the use of blind vias connecting the first two PCB layers, escape routing from the third and fourth BGA signal rows can be done through the second PCB layer. Therefore, signal routing can be accomplished in only two PCB layers.

3.1.4 Pad Surface Finish

Two commonly used PCB pad surface finishes for surface mount devices are:

- Electroless Ni + immersion gold plating (ENIG)
- Cu OSP (organic solderability preservative)

ENIG is a versatile process and enables fabrication of high-density BGA substrates needed for high-performance IC chips. ENIG is used extensively in advanced IC packaging of microprocessors, ASIC, and DSP components. Both finishes require the surface coating to be uniform, conforming, and free of impurities to ensure a consistent, solderable system.

Ralph Webber, November 23, 2004
The ENIG finish consists of plating electroless nickel over the copper pad, followed by a thin layer of immersion gold. The allowable stresses and temperature excursions the PCB is subjected to throughout its lifetime, determine the thickness of the electroless nickel layer. This thickness is typically 5 µm nickel and about 0.05 µm for gold, to prevent brittle solder joints.

By its nature, ENIG plating forms brittle intermetallic compounds of nickel, tin, and other elements in the plating after solder balls are attached to the package. Certain conditions of high strain and high strain rates are known to cause ENIG solder joints to fail. Therefore, you must avoid excessive shock and bending of the PC board during assembly, handling, and testing of FCBGAs with ENIG plating.

The second recommended solderable finish consists of an organic solderability preservative (OSP) coating over the copper-plated pad. The organic coating assists in preserving the copper metallization for soldering.

The advantages of ENIG plating over Cu OSP are:

- Longer shelf life
- Permanent coverage of copper vias
- Resistance to oxidation during multiple-pass assembly
- Contamination resistance

Other alternative pad finishes, which are available in the market today, are hot air solder leveled (HASL), immersion silver, immersion tin, and electrolytic Ni-Au. Industry efforts are focused on developing and qualifying lead-free metallization. Therefore, the continued acceptance of HASL and other lead-based metallizations may become limited in the Microelectronics/PCB manufacturing industry.

### 3.1.5 PCB Stack and Thermal Vias

Adequately designed thermal vias, along with an adequate number of thermal balls, contributes to the thermal enhancement of both large and small BGA packages.

This section focuses on the design and value of thermal vias.

The thermal balls must attach to a thermal spreading plane or land in the PCB with adequate area to convect and radiate the heat generated by the component. Thermal vias are the primary method of heat transfer from the PCB thermal land to the internal copper planes or to other heat removal sources.

Thermal vias help to give a closer coupling of the device to the buried planes, which results in more efficient heat spreading and more uniform temperature distribution across the PCB. The larger effective cooling area around the device also allows its heat to be more efficiently dissipated off the board surfaces by convection and radiation.

Ralph Webber, November 23, 2004
The overall cooling effect can be significant, especially in smaller packages with less substrate layers, where little heat spreading can occur in the package itself.

Important factors in both the BGA package thermal performance and the package-to-PCB assembly are:

- Number of thermal vias used
- Size of the thermal vias
- Construction of the thermal vias

Figure 3-12 and Figure 3-13 show how varying the number of thermal vias affect PCB thermal resistance. Various sizes of die for two and four-layer PCBs are used.

![Figure 3-12 Impact of Number of Thermal Vias vs. Die (Chip) Area](image)

Note: Apply bare die to the JEDEC board.

**Figure 3-12 Impact of Number of Thermal Vias vs. Die (Chip) Area**
The curves indicate that a point of diminishing returns occurs where additional vias do not significantly improve the thermal transfer through the board. The number of thermal vias will vary with each product assembled to the PCB, depending on the amount of heat that must be moved away from the package and the efficiency of the system heat-removal method.

To arrive at an optimum value for your board construction, you must perform characterization of the heat-removal efficiency versus the thermal via copper surface area. The number of vias required can then be determined for any new design to achieve the desired thermal removal value.

In general, adding more metal through the PCB under the IC improves operational heat transfer, but requires careful attention to uniform heating of the board during assembly.

### 3.2 Manufacturability Considerations

#### 3.2.1 Board Design

A well-designed board that follows the basic SMT considerations greatly improves the cost, cycle time, and quality of the end product. Board design should comprehend the SMT-automated equipment used for assembly, including minimum and maximum dimensional limits, and placement accuracy.
Many board shapes can be accommodated, but the front of the board should have a straight and square edge to help machine sensors detect it. Odd-shaped or small boards can be assembled, but require panelization or special tooling to process inline.

In general, the more irregular the board is (non-rectangular with cutouts) the more expensive the assembly cost will be.

Fiducials, the optical alignment targets that align the module to the automated equipment, should allow vision-assisted equipment to accommodate the shrink and stretch of the raw board during processing.

Fiducials define the coordinate system for all automated equipment, such as printing and pick-and-place. The following guidelines are useful for ensuring ease-of-assembly and high yield:

- Automated equipment requires a minimum of two and preferably three fiducials.
- A wide range of fiducial shapes and sizes can be used. Among the most useful is a circle 1.6 mm in diameter with an annulus of 3.175 mm/3.71 mm. The outer ring is optional, but no other feature may be within 0.76 mm of the fiducial.
- The most useful placement for the fiducials is an L configuration that is orthogonal to optimize the stretch/shrink algorithms. When possible, the lower left fiducial should be the design origin (coordinate 0,0). It is also common to position the package pin 1 (A1) corner in the corner without the fiducial.
- All components should be within 101.6 mm of a fiducial to guarantee accuracy of placement. For large boards or panels, a fourth fiducial should be added.

If the edges of the boards are to be used for conveyer transfer, a cleared zone of at least 3.17 mm should be allowed. Normally, the longest edges of the board are used for this purpose, and the actual width depends on equipment capability. Although no component lands or fiducials can be present in this area, breakaway tabs may be present in this area.

By using the longest edges for support on the conveyor rails, board sag due to self-weight is reduced considerably on large PCB designs with numerous components. On smaller boards, it may not be as critical.

Inter-package spacing is a key aspect of DFM. The question of how close together components can be placed is a critical one. The following component layout considerations are recommendations based on TI experience:

- There should be a minimum of 0.508 mm of space between land areas of adjacent components to reduce the risk of shorting.
- The recommended minimum spacing between SMD discrete component bodies is equal to the height of the tallest component. This allows for a 45 degree soldering angle in case manual work is needed.

Ralph Webber, November 23, 2004
• Polarization symbols should be provided for discrete SMDs (diodes, capacitors, etc.) next to the positive pin.

• Pin-1 indicators or features are needed to determine the keying of SMD components.

• Space between lands (under components) on the backside discrete components should be a minimum of 0.33 mm. No open vias may occupy this space. The direction of backside discretes for wave solder should be perpendicular to the direction through the wave.

• Do not place SMT components on the bottom side that exceed 200 grams per square inch of contact area with the board.

• If space permits, symbolize all reference designators within the land pattern of the respective components.

• It is preferable to have all components oriented in well-ordered column and rows.

• Group similar components together whenever possible.

• Allow room for testing and rework.

With considerations for rework, to allow a rework nozzle to come down to the board to gasket properly you will need approximately .1" clearance to your next adjacent component.

You will also want to take into consideration the thermal heat transfer conducted through the board to the next adjacent component. This is difficult to predict because the amount and location of Cu in the board varies considerably from design to design.

In general, if your next adjacent active component is .250" it will not reflow or partial reflow. Partial reflow of a BGA device could build up additional stresses in the solder joints.

Ralph Webber, November 23, 2004
3.2.2 Process Considerations

The following assemblies are listed in order from the least labor intensive to the most labor intensive.

1). Single Sided Assembly - All SMT and PTH components located on the same side of the board.

2). Double Sided Assembly – All SMT active devices and all PTH devices located on the topside, only SMT passives on the bottom side.

3). Double Sided Reflow – All SMT devices reflowed, and all PTH devices hand soldered.

4). Mixed Double Sided Assembly – SMT active devices and PTH devices located on both the top and bottom side of the assembly. This will require hand soldering and/or wave solder fixturing.

3.2.3 Reflow Considerations

- Avoid large components opposite each other on double sided boards.

- Avoid clustering large SMDs in one area as this will result in uneven heating during reflow.

Ralph Webber, November 23, 2004
• Avoid fine pitch devices or BGA’s on both sides of an assembly.

3.2.4 Wave Solder Considerations

• No discretes or passives larger than an 1810 size package. This includes C size or large tantalum capacitors.

• No active components larger than 16 pin

3.2.5 Inspection Considerations

• No discretes or passives larger than an 1810 size package. This includes C size or large tantalum capacitors.

• No active components larger than 16 pin SOIC’s.

• Adhere to proper wave solder component orientation guidelines.

• Orient all polarity marking the same direction to facilitate identification of reversed components.

• Keep all placement angles at 0/90/180/270 degrees.
3.2.6 Component Spacing Considerations

Minimum component spacing specifications exist to ensure that assemblies will be manufacturable within an automated SMT process. Minimum spacing requirements also ensure that solder joints can be visually inspected, and if necessary, reworked or repaired.

Figure 3-15 Component Spacing Specifications
Considerations must also be made for spacing of different sized components. See Figure 3-17.

![Figure 3-16 Layout and Orientation for Multi-Connector Applications](image)

**Preferred**

- Connectors are placed in same orientation

**Not preferred**

- Connectors are not placed in same orientation

![Figure 3-17 Layout and Orientation of Different Sized Components](image)

- Fragile Device – Micro BGA, CGA, Leaded Fine Pitch, etc.
- Tall adjacent components
- 5mm minimum preferred clearance to fragile devices
- 5mm Preferred clearance between tall components

Ralph Webber, November 23, 2004
4 BGA Rework Preparation

4.1 Component Sensitivity

This method may subject the component to extreme temperatures. Evaluate the component's tolerance to heat prior to using this method. Plastic BGA's are especially sensitive to moisture absorption. Carefully evaluate pre bake requirements.

4.2 Circuit Board Sensitivity

Circuit boards are fabricated from a wide variety of materials. When subjected to the high temperatures they are susceptible to the following types of damage:

1. Layer delamination.
2. Copper delamination, separation of pads, and barrels of inner layers.
3. Burns and solder mask chipping.
4. Warp.

Each circuit board must be treated individually and scrutinized carefully for its reaction to heat. If a series of circuit boards are to be reworked, the first several should be handled with extreme care until a reliable procedure is established.

4.3 ESD Safe Work Areas

BGA’s should be handled in ESD safe work areas in order to prevent damage to sensitive components from electrostatic discharges. These areas must be designed and maintained to prevent ESD damage.

4.4 Proper Handling and Storage of ESD Sensitive Devices and PCBs

The following practices should be adhered to when working with BGA components and PCB assemblies:

1. PCBs should be handled at properly designated work areas only.

2. Designated ESD safe work areas must be checked periodically to ensure their continued safety from ESD. The areas should be monitored for the following:
   - Proper grounding methods.
   - Static dissipation of work surfaces.
   - Static dissipation of floor surfaces.

Ralph Webber, November 23, 2004
Operation of ion blowers and ion air guns.

3. Designated work areas must be kept free of static generating materials such as Styrofoam, vinyl, plastic, fabrics or any other static generating materials.

4. Work areas must be kept clean and neat in order to prevent contamination of the work area.

5. Circuit board assemblies should be handled by the edges. Avoid touching the circuits or components.

6. Components should be handled by the edges when possible. Avoid touching the component leads.

7. When not being worked on, sensitive components and circuit boards must be enclosed in shielded bags or boxes. There are three types of ESD protective enclosure materials including:
   - Antistatic - Provides antistatic cushioning for electronic assemblies.
   - Static Shielding - Prevents static electricity from passing through the package.
   - Static Dissipative - An "over-package" that has enough conductivity to dissipate any static buildup.

8. Whenever handling a circuit board assembly the operator must be properly grounded by one of the following:
   - Wearing a wrist strap connected to earth ground.
   - Wearing heel grounders in contact with a static dissipative floor surface.

9. Stacking of circuit boards and assemblies should be avoided to prevent physical damage. Special racks and trays are provided for handling.

### 4.5 Proper PCB Preheating

One of the keys to insuring a proper reflow cycle is getting to and maintaining the proper preheat temperature. This is accomplished with a PCB preheater, which heats the board to between 75 and 125° C prior to the application of heat to the part area requiring rework. This part of the rework process is a critical component as done properly it minimizes PCB warpage during component removal while at the same time limiting the thermal shock to the PCB. The maximum temperature is determined by the thermal...
expansion of the PCB. In order to maintain the integrity of the PCB laminate the maximum preheat temperature is set approximately 10°C or below the Tg (glass transition temperature) of the PCB material. Higher thermal preheat temperatures minimize the potential thermal distortion and shock to the PCB during the reflow process.

5 BGA Rework Nozzle Selection Process

The following information provides selection criteria for the correct Summit Rework Station hot gas nozzle for your rework application. The alternatives may not be readily apparent and will include both rework system needs and the requirements of your particular application.

Summit rework systems use a pickup tube that is independent of the heating nozzle. This pickup tube is primarily used to place and remove components. However, the independent motion of the pickup tube has several additional advantages. These include:

1. Common size components with differing thickness (heights) do not require a separate nozzle for each component.

2. The ability to control component surface temperature with a feature referred to as Positive Air.

3. Programmable force of 50 – 450 grams to a component during placement.

4. Available sequences to either release or hold a component during the Place-Reflow process.

5. Ability to use specially designed extraction tools for removal of high-density connectors.

6. Zero Force removal of components to avoid undesired pressure on the molten solder balls.

Selecting the best nozzle design for your application includes consideration of the following items.

1. Component size (X and Y)
2. Internal clearance requirements
3. Adjacent component spacing
4. Pickup tube diameter
5. Part nest design options
6. High flow versus low flow processing
7. Nozzle will rest on the PCB, or be elevated above circuit board surface?
The following will review these considerations as well as how the above requirements and system needs influence your selection. There are six main types of nozzles available for a Summit Rework System. These include:

Pro Series Nozzles for components 15 – 68 mm square, or rectangular *

Pro Step Nozzles a premium nozzle for improved temperature control of components 5 – 22 mm square, or rectangular

Mini Series Nozzles for components 5 – 27 mm square, or rectangular

Sub-Plenum Nozzles for leaded components, such as a QFP,

Connector Nozzles for vertical connectors, 50 – 800 mm in length

Right Angle Connector Nozzles for Straddle Mount Connectors up to 127 mm in length

* Larger nozzles can be designed on a custom basis if required

The following diagram Figure 5-1 shows the dimensional relationships between a cross sectional view of a nozzle and components. Separate diagrams are shown for Mini-Series and Pro-Series nozzles. Proper clearance between the component and internal nozzle dimension(s) is required as well as adequate spacing between adjacent components. The clearances shown can be used as a guide when evaluating custom nozzle designs or the availability of adequate spacing between adjacent components.

In extremely tight situations, it may be desirable to raise the nozzle above the plane of the board in order to avoid hitting adjacent components with the “skirt” of the nozzle. Alternatively, the vent holes and skirt may be raised in custom nozzle designs in order to increase clearance availability. A photograph of this type of nozzle is shown in Figure 5-1.

Ralph Webber, November 23, 2004
5.1 Pro-Series Nozzles

Applications use Summit Pro-Series nozzles for component sizes typically ranging from 15 mm to 68 mm. Both larger and smaller nozzles have been made for custom applications.

Pro-Series nozzles are used for reworking BGA’s, CSP’s, QFP’s and a wide variety of other surface mount components. Separately available with these nozzles are component nests. The pickup tube can be used to pick and place components when a vacuum is applied. It is also used to cool the top of large, temperature sensitive, components when
The “Positive Air” feature is used to blow room temperature air on the top, center, of the component during heating to reflow.

![Diagram of Pro-Series Nozzle “Positive Air” Feature](image)

**Figure 5-2 Pro-Series Nozzle “Positive Air” Feature**

### 5.1.1 Pickup Tubes and Nests

Blind hole nests are used when components to be reworked are smaller than the outside diameter of the 3/16” (4.76 mm) pickup tube. The depth of the nest hole is made less than the component thickness. In this manner, the component is raised above the surface of the nest and enables contact between the pickup tube tip and component for pick and placement.

Ralph Webber, November 23, 2004
There are four types of pickup tubes available. In general, it is advisable to use the pickup tube with the largest internal diameter in order to realize the maximum vacuum pickup lifting capacity.

Specifications for the pickup tubes are shown below.

<table>
<thead>
<tr>
<th>Part Number 1</th>
<th>Outside Diameter</th>
<th>Inside Diameter 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>9001-0869-00</td>
<td>¼” (6.35 mm, SS)</td>
<td>0.130” (3.3 mm)</td>
</tr>
<tr>
<td>9001-0870-00</td>
<td>3/16” (4.76 mm, SS)</td>
<td>0.090” (2.29 mm)</td>
</tr>
<tr>
<td>9001-0959-00</td>
<td>3/16” step down to .095” (2.4 mm)</td>
<td>0.052” (1.32 mm)</td>
</tr>
</tbody>
</table>

1 Pickup Tube assembly is provided with retention spring and “chimney collar”
2 Inside diameter is measured by the leading chamfer entry hole diameter

In addition to the many standard Pro Series, Pro Step, and Mini Series nozzles, there are a wide variety of custom, and special application designs. The custom Pro-Series nozzle below was made for an application that required a higher air vent position. This was done so that the hot exhaust gas would not heat adjacent components that were positioned higher than the component to be reworked. Raising the deflector skirt also provided additional clearance with adjacent components.

![Figure 5-3 Custom Pro-Series Nozzle](image)

In addition, Pro-Series nozzles are available in an “extended shield” configuration which increases the length of the nozzle either 7.6 mm or 10 mm from the standard length of 47 mm (1.850”). Use these nozzles when additional topside component clearance is needed inside or immediately adjacent to the nozzle.

Components mounted at 45 degree angles (with respect to the board) can easily be reworked with 45-degree Pro Series nozzles.

Ralph Webber, November 23, 2004
5.2 Pro-Step Nozzles

Pro-Step nozzles are a premium nozzle, designed to optimize airflow across small components (i.e., flip chip and CSP) as well as long narrow components (i.e., connectors). Current designs are available in a range of 5 – 22 mm square. Rectangular shapes are also available, as well as custom sizes larger than the standard 22 mm.

Figure 5-4 Pro-Step Nozzle

5.3 Mini-Series Nozzles

Our Mini-Series nozzles are used for smaller components ranging in size from 5 mm to 27 mm, square and rectangular for applications with minimal adjacent component clearance. The Summit “Low Flow” option is used to reduce air flow velocity and minimize the potential for component displacement during the reflow cycle. Figure 5-5 below shows the typical configuration for Mini-Series Nozzles.

Figure 5-5 Mini-Series Nozzle Typical Configuration

Ralph Webber, November 23, 2004
Raised deflector skirts are also available with Mini-Series nozzles as was previously discussed for Pro-Series nozzles. This provides additional clearance for tightly packed adjacent components.

5.4 Sub-Plenum Nozzles

Sub-Plenum nozzles are used for leaded components, such as Quad Flat Packs (QFP’s), 20 – 40 mm square, or rectangular. This type of nozzle restricts heated airflow to the perimeter of the component and directs the heated air at the leads. It avoids unnecessary heating of the component body during the reflow cycle.

![Sub-Plenum Nozzle](image.png)

Figure 5-6 Sub-Plenum Nozzle

5.5 Vertical Connector Nozzles

Summit Vertical Connector nozzles were initially developed for custom requirements and have evolved into a standard nozzle line. These nozzles are available in a range of lengths up to ~8 inches (200 mm).

5.6 Straddle Mount Connector Nozzles

Straddle Mount Connector nozzles require right angle nozzles that can simultaneously heat the connector leads on the topside and underside of the circuit board. These nozzles are available in sizes capable of reworking connectors that are up to 5” (127 mm) in length. However, this capability is constantly undergoing change and it is suggested that

Ralph Webber, November 23, 2004
you contact the dealer (VJ Electronix) for the most current availability. Many sizes are available with mechanical extractors as shown below. The extractor enables mechanical removal of the connector after reflow.

**Figure 5-7 Straddle Mount Nozzle Assembly**

---

### 6 BGA Rework Profile Development Process

Thermal profiling is required for several operations in the reflow process including device removal and reflow. Each specific BGA site to be reworked must be individually profiled as each site has variations which affect the reflow process including the location of adjacent components and heat-sinking variations of the PCB internal layers. By individually profiling each site the risk of over or under heat exposure to components, lifted pads, damaged solder mask as well as improperly soldered joints is minimized.

The BGA component location to select for the profile should be the one that will generally be the most challenging to rework. The most difficult BGA components to rework will generally be closest to the edge of the circuit board, in the most densely populated area, near or surrounded by a ground plane.

Ralph Webber, November 23, 2004
There are several methods available to correctly identify and build a PCB profile. The method used is a function of the availability of a sample or test profile PCB and component reworked. One method assumes that a profile PCB and part is available, while another assumes that no dedicated profile PCB exists. Whatever the method used it is critical to read the joint temperature. Thermocouples attached next to the part being reworked or in the air are not accurate solder temperature indicators.

Using a test PCB and part is the preferred profiling method as it provides the most accurate joint temperature readings, reduces the chances for damage to nearby components on the boards being reworked and reduces the chances for board damage. In this method the thermocouple is attached to the solder ball either on the side of the BGA ball using conductive epoxy or into the BGA joint by drilling through the backside of the BGA and filling it with thermally conductive epoxy. The thermocouple is then properly routed between the balls or on the underside of the PCB.

Profiling without a dedicated PCB is the other method used. In this method it is critical to measure the temperature under the center of the BGA to be reworked. Insulated thermocouples are pushed underneath the center of the BGA to approximate joint temperatures. It has been shown that thermocouple readings are affected by hot air nozzles of rework stations if they are placed near the outside rows of the solder joints. Thus the center BGA solder joints, which tend to be lower in temperature than the outside rows, need to be measured. In this technique the reflow profile is established during the removal process.

In both profiling methods there are several points, which should be monitored:

- BGA part center joint temperature
- BGA part edge joint temperatures
- Adjacent component nearest body temperature (if applicable)
- PCB temperature

![Figure 6-1 Representative Profile of Hot Air BGA Rework System](image)

Ralph Webber, November 23, 2004
7 BGA Rework Part Removal Process

The objective of this step is to remove the component while minimizing the impact to the PCB. The most common PCB defects that should be avoided include lifted pads and site warping. The basic requirements of this operation are as follows:

Preheat the entire PCB between 75°C and 125°C

Insure that all joints have a temperature greater than 190°C

Make sure that all joint temperatures are less than 220°C

Insure no solder is smeared on the part bottom or edge, which could aid in part recovery

Maintain adjacent component temperatures to less than 180°C in order to minimize the heat impact on these parts

* Note these temperatures assume standard Pb/Sn soldering temperatures.

8 BGA Rework Cleaning Process

After removal of the BGA from the PCB the board must be cleaned. Since the balls are eutectic alloy soldered to both the part and the PCB, some of the ball will likely remain on the PCB after part is removed. The remaining solder may not be uniform in its composition and volume. The volume varies due to how the molten solder separates between the ball and the carrier pad surface. This involves removing all the solder balls and removing all the solder residue. Once complete the board can be cleaned.

Cleaning of the pads can be done by one of several methods. The methods for solder removal include solder wick or solder vacuum techniques. The solder vacuum process has less chance of damaging the pads while but is somewhat slow. The solder wick [braid] technique, while faster, can lead to lifted pads or damaged solder mask areas if not done properly or with the right size of wick.

After solder removal from the pads of the BGA site is complete the site can be cleaned following similar techniques as those used in the initial assembly of the product. If a water-soluble or no-clean paste flux has been used the site does not require cleaning.
9 BGA Rework Applying Solder Process

Solder must be selectively applied to the PCB area of the BGA in order to attach the solder balls of the BGA to the PCB. Since solder paste provides a "tacking" functionality it is preferred for the device placement. In addition it provides compensation for height differentials while assisting to minimize warpage of the PCB. The most common methods of solder paste for rework are discussed below:

9.1 Dispense

It is possible to manually dispense solder paste with a syringe and needle. This may result in inaccuracies and excessive cycle time. The use of semi-automated dispensing systems, which control the quantity of paste deposited by adjusting air and timer settings provides greater control over the paste quantity, but will not improve accuracy. A fully automated dispensing system usually consists of an X-Y table and a computer controlled dispense head. These systems are programmed with the coordinate data for the locations requiring solder paste and then sequentially dispense the paste at the desired locations. They are very accurate in the paste deposit placement providing very consistent paste deposit quantities. However, one of the drawbacks of using a dispensing system is the requirement to use a solder paste with a lower metal content. Higher metal contents (87%) usually result in frequent clogging of the nozzle. With this lower metal content, the paste deposit is more prone to slump and separation than higher metal contents. Applying paste to the same locations on many boards is fairly easy due to the programming of the system and fixturing of the PCB. The major drawback to the dispense system is throughput. For larger pin count BGA’s, cycle time is limited by the maximum speed of the pump and may take as long as five minutes when set-up time is included. The program and set-up time must be included for every different BGA to be processed.

9.2 Metal Stencils

Metal stencils for depositing solder paste have been in used reliably for many years. The metal stencil printing process begins with the stencil being aligned with the land patterns on the PCB. The stencil must then be held in place in a manner that ensures good contact with the PCB. Next, a squeegee is used to roll a bead of solder paste across and down through the apertures of the stencil.

Finally, the stencil is lifted from the PCB surface resulting in finely defined solder paste deposits. Cleaning the stencil must be done to ensure the stencil will provide acceptable results on the next BGA to be reworked. A variation on the metal stencil process is to print the solder paste directly onto the solder balls instead of the BGA lands on the PCB. This method provides the advantage of printing solder paste where placing a stencil on the board would be impossible due to space considerations.

Ralph Webber, November 23, 2004
9.3 Stencils Flexible Removable

A relatively new material/process for deposition of solder paste is the use of flexible solder paste stencils. These stencils are laser cut from a polymer film with a residue-free adhesive backing that allows for easy removal. The first step to using the flexible stencil is to remove the paper to expose the adhesive backing.

The stencil must then be manually aligned with the land patterns on the PCB and firmly pressed down in place on the PCB surface. A squeegee is then used to roll a bead of solder paste across and down through the apertures of the stencil. The stencil must then be carefully removed from the PCB surface. Cleaning is not necessary as the stencil is disposable.

9.4 Semi-Permanent Stencils

Another method of stenciling solder paste on a PCB is to use a stencil that remains in place on the site location and becomes an integral component of the PCB. The stencil material is a polyimide film with a high temperature adhesive covered with a paper backing.

This material combination has been used on PCBs for many years now in the form of bar code labels and Kapton tape. This material is available in the standard formulation or in an ESD safe variety. Like the removable stencil, the semi-permanent stencil is laser cut and can be provided in a number of different configurations. When using the semi-permanent stencil, the paper backing material is removed exposing the high temperature adhesive.

The stencil is then manually aligned with the land patterns on the PCB and pressed into place. Again a squeegee is used to roll a bead of solder paste across and down through the apertures of the stencil. At this point the paste application process is complete. The stencil is not removed from the PCB and therefore no stencil cleaning is required.

10 BGA Rework Part Placement Process

Once the solder paste has been applied to the pads and cleaned off properly, the device can be placed onto the PCB.

When the device is placed using a modern rework system, split prism optics can be used to assist in the proper placement of the device. Using a prism, optical images of both the solder balls on the device as well as the PCB pads are superimposed in a microscope field of view on a monitor. The two images can then be aligned by X, Y and rotational adjustments.

Placement accuracy criteria are the same as the initial placement at the time of initial assembly. The device should be placed with a force per the manufacturers' specifications.
such that the solder paste contacts 50% of the PCB pad and not be in contact with the vias. For test boards the placement accuracy can be confirmed using double-sided tape adhered to the PCB BGA site. The BGA can then be placed onto this site, firmly held in place and examined at an angle under a microscope for accuracy.

The BGA can also be hand-placed for larger pitch sizes (1.00 and 1.27mm). Care must be taken in handling the parts at the edges while making sure the operator is properly grounded.

11 BGA Rework Part Reflow Process

Once parts have been placed, the applied solder must be reflowed to attach the BGA to the PCB pads. Preheating the entire PCB to between 70°C and 125°C is critical to minimizing PCB warpage during the heat cycle. A developed thermal profile should have the following basic characteristics:

- Peak joint temperature to minimize Pb dissolution
- Minimum joining temperature that is high enough to ensure wettability
- Follow solder manufacturers specified reflow range
- Ensure dwell time above reflow as per the solder manufacturer's specifications

After successfully attaching the BGA to the PCB, and allowing the board to be cooled down, the perimeter row of solder joints should be visually examined. The solder joint between the ball and the PCB pad should be similar in size and shape to the solder joint between the ball and part. If a water-soluble paste is used, the reworked area needs to be cleaned.

The BGA rework process should not introduce any PCB warping and bowing. This is a serious problem seen with thin PCBs. If the PCB is bowed or warped they might not be able to properly fit into the final assembly or opens may be formed.

In addition, care must be taken to ensure that adjacent components are not damaged during the reflow process. This care must be extended to components with low melting points such as connectors or standoffs or adjacent moisture-sensitive devices. Care should also be taken to not overheat any internal thermal grease if the BGA is capped.

Ralph Webber, November 23, 2004
12 AutoProfiling for the VJE Summit 1100 Rework Station

AutoProfiling is a tool that provides an effective way for generating a rework heating cycle in a minimum amount of time. The technique can speed up the creation of an optimum thermal cycle for solder reflow during component attachment or removal from circuit boards. It is accomplished by learning a top heater power cycle that is controlled by actual solder-joint temperatures. The technique relies upon the software to record optimal power settings to heaters for later recall and reuse.

Board thermal conditioning is performed first with the bottom heater to minimize board distortion during rework. Optimum board temperature (100 – 120°C) and uniformity, prior to the application of top heat, are two key factors influencing the quality and repeatability of the reflow process.

AutoProfiling controls top heater power by matching IC component thermocouple (TC) temperature (a TC placed near the IC component solder balls) to a desired, adjustable, component temperature profile (time-temperature curve). The desired component temperature profile represents a preheat, reflow and post-reflow thermal cycle baseline that is recommended by the software and can be accepted or modified by the operator.

The operator has the ability to modify the recommended temperature targets, ramp rates and hold times at temperature. The top heater temperature is adjusted by the software’s proportional heater power control to achieve the desired component temperature profile. This top heater temperature profile is “learned” and stored as heater temperature set points on subsequent runs.

The following is a step-by-step AutoProfiling method, followed by guidance on customizing the capability for your applications. The software capability is available exclusively on SRT Summit 1100 and 2100 rework systems, software versions 6.0.0 and 3.0.0, respectively, or later releases.

AutoProfiling is presented in ten separate steps. Each step is reviewed with respect to detailed software commands and features that enable you to use it as a step-by-step tutorial. The ten AutoProfiling steps include:

1. Board-Conditioning Temperature Profiling
2. AutoProfiling Board Preparation
3. Select Board Location
4. SierraMate Software Setup
5. Datalogging
6. Recommended Profile Selection

Ralph Webber, November 23, 2004
7. Centering Component in Nozzle

8. AutoProfiling

9. Real-Time Viewing and Interpreting AutoProfiles

10. Viewing and Using Saved AutoProfiles

The step-by-step tutorial uses the SRT demo board and BGA 225 component that is supplied with a Summit 1100 or 2100 rework system. Successful AutoProfiling and rework requires precision alignment of the component to the board and achieving an appropriate balance between four key rework objectives. These objectives include:

1. avoiding thermal stress-induced board distortion,
2. limiting the peak temperature at the top of the component,
   1. optimizing the temperature and time interval above solder liquidus, and
   2. minimizing rework cycle time.

12.1 Board-Conditioning Temperature Profiling

When reworking any board it is desirable to begin top heating only after bottom heating has brought the board to a minimum temperature of 100°C. This board “conditioning phase” is represented in the default thermal profiles of the Place_Reflow and _Reflow sequences. In order to determine whether board-conditioning bottom heater temperature is adequate, you must profile the board temperature and datalog the results. This datalog can then be evaluated in order to determine the adequacy of the board-conditioning sequence to raise board temperature to 100°C before top heating begins.

In order to develop an AutoProfile process, you now need to set up the _Reflow sequence to permit datalogging of the board-conditioning phase.
Move the “Start Datalog” command to a position immediately following the “Air on” command. This will require the system to start plotting temperatures as soon as bottom heating for board conditioning begins. Using the pull-down menu, save the file before exiting.

Select the SRT demo board and BGA 225 1A component. With the trackball cursor click on the SRT logo to activate pull down menus. Access the Process pull down menu and select Times and Temps.

Note that the displayed profile does not include the board and bottom heater conditioning steps.

The conditioning steps include preheating the bottom heater to 5°C less than your selected bottom heater set point (for 225°C the preheat temperature becomes 220°C). This slightly lower bottom heater temperature allows for thermal momentum and must be achieved before the timer begins for maintaining bottom heater temperature at 225°C for 60 seconds.

At the conclusion of this time interval the board temperature should be at 100°C, minimum. The first time that a new rework system is used, there will most likely be no learned AutoProfiles saved in the software (this would appear as a purple profile line). In

Ralph Webber, November 23, 2004
In this case, you need to select “Use Learned Auto Profile” and click on the **Default** button. This will apply default profile parameters defined for the board and component.

Press the **Process Steps…** button in order to access the board conditioning parameters as displayed below.

![Figure 12-2 Board Conditioning Parameters Display](image)

The dialog box that appears indicates that the board will be heated for 60 seconds with the bottom heater temperature set at 225°C. There is no top heat applied at this point, so the top heater temperature is set to 0°C. The 225°C temperature is as measured by the bottom heater thermocouple, not an indication of the board temperature after 60 seconds of heating.

In order to determine the temperature of the board, the temperature must be datalogged during the heating cycle using a board mounted thermocouple. This capability first needs to be set up within the sequence editor for the _Reflow sequence._

Use the SRT demo board and attach a thermocouple to the top of the board. Plug the thermocouple into TC port #3 and position and clamp the board to the area of the X-Y table you intend to use for reworking. Attach the thermocouple to the top of the board at the center of the rework site.

Ralph Webber, November 23, 2004
Note: When developing a process for your own production board, you can place the thermocouple close to your site to any available board surface.

Go to the Datalog pull down menu and select Setup. The Datalog Setup box appears as shown below. Verify that Datalog with Live Viewing is selected. Establish a starting File Name. The file name will be limited to eight characters and should end in at least one zero. One zero equals ten files before file overwriting begins, two zeros equals 100 files before overwriting, etc. The file name will include the file extension .LOG.
Select, by checking the appropriate boxes, and name the thermocouples that you will be using. These are the thermocouples whose temperatures will be displayed during the _Reflow sequence.

You are now ready to run the _Reflow sequence and evaluate board-conditioning temperature. In the Process, Times and Temps window, press the Learn Profile button. (Do not press the green hi-lited Run button, as this will merely run any previously learned profile that has been saved.) The process begins and the following screen is displayed. Press on the graph icon in order to watch the temperature profiles develop during the heating cycle.

Note: The most current software will display 16 available data channels. Older software versions will only display 8 data channels.
From a cold start, the plot shows the bottom heater (blue), the board temperature (green) and the top heater temperature (red) all starting at room temperature. As the bottom heater ramps up to 225°C, the board slowly increases in temperature while the top heater remains off and cold. When the board reaches 84°C, top heater power is applied that creates a spike in the top heater temperature. This is because the top heater starting set point, at 100°C, is higher than the starting board temperature. The two temperatures need to become equal before top heater power will reduce to achieve the desired temperature and ramp rate.

Note: This learn sequence was aborted at 240 seconds into the autoprofiling learn datalog. For the purposes of examining board-conditioning response, there is no component on the board to be heated by the top heater. Further datalogging will not produce any additional and meaningful data at this point, so the learning process can be stopped by pressing the red Stop button.

Contrast the prior cold start with a warm start. The board is placed on the work zone at room temperature but both the top and bottom heaters start warmer due to the prior sequence that was recently run.
First, you will notice that when datalogging began, the temperature of the bottom and top heaters were slightly higher than with a cold start. In this case, the board was slightly heated from bottom heater radiation during the site alignment step. As a result of these differences from a cold start, the board reached a temperature of 90°C after the 60 second conditioning step. This was 6°C higher than with the cold start. The top heater still had a small spike. However, it was slightly less pronounced than with a cold start since the board temperature was a bit higher, and closer to the top heater starting set point, when top heater power was applied.

It is still advisable to begin top heating with a board temperature of at least 100°C. In both the warm and cold starts, the board needs a combination of a longer soak period and a higher bottom heater conditioning temperature in order to reach 100°C in a reasonable time frame. Return to the Process pull down menu and select Times and Temps. The additional time can be added to the 60 second holding condition accessed from the Process Steps… button.

Grabbing the blue bottom heater temperature boxes and raising them from 225°C to 250°C will raise the bottom heater temperature for the reflow sequence. The bottom heater board-conditioning temperature must also be increased by accessing Process Steps… and raising the bottom heater temperature to 250°C.

Ralph Webber, November 23, 2004
Relearning the sequence (by activating the Learn Profile button) from a cold start, with a 90 second board-conditioning duration and bottom heater temperature of 250°C, results in the following datalog.

![Graph showing profile temperatures with Learn Profile Activated]

In the above example, the board temperature reached 100°C by the time the top heater heating cycle began. The top heater temperature no longer shows a spike. The board temperature and starting top heater set point are the same (100°C).

When setting up the AutoProfile sequence, we will use the 250°C bottom heater temperature and 90 second soak interval before learning the top heater reflow profile.

### 12.2 AutoProfiling Board Preparation

The installation packet includes a BGA 225 component for attachment to one of the sites on the board. After applying an appropriate flux, attach the BGA 225 where shown by using the Place_Reflow sequence that has been established in the software for this component and board. Use a 6 mm (1/4") pickup tube and 27 mm ProSeries nozzle for BGA placement and reflow. After BGA placement and reflow, insert a thermocouple beneath the BGA and secure with Kapton tape as shown.
It is important to place the thermocouple well under the component and as close to the center as possible. In this manner, the AutoProfiling sequence will best reflect the temperature at the solder balls.

12.3 Select Board Location

Locate and secure the instrumented board on the X-Y table as shown in the following photo. Secure the board in place by using the sliding clamp and connect the component thermocouple to position T/C3 on the rework system. Position shim stock to cover areas of the bottom heater that are not needed for board heating. This will improve the thermal efficiency of the bottom heater.
12.4 **SierraMate Software Setup**

From the main 1-2-3 GO screen, select the SRT DEMO BOARD, the BGA 225 2C component (or other component/site identifier you select to reflow) and _Reflow Site sequence. Site numbers have been added to the screen below to show how board component locations are identified for the SRT demo board. With Summit 2100 rework systems, unique component/site identification is required for automatic placement of the component to a specific site. This is why you will see BGA 225 components listed in the *Select Component* section followed by alphanumeric site identifiers.

With Summit 1100 and 2100 rework systems, the unique name that is described by component/site enables you to have a unique thermal sequence for the same component (e.g. BGA 225) but unique to the thermal characteristics associated with different sites.
12.5 Datalogging

Optional real-time datalogging can be accomplished in the following manner.

With the trackball cursor, click on the SRT logo to activate pull down menus. (On some systems this access may be password restricted) Go to the Datalog pull down menu and select Setup. The Datalog Setup box appears as shown below. Verify that Datalog with Live Viewing is selected. Establish a starting File Name. The file name will be limited to eight characters and should end in at least one zero. The file name will automatically include the file extension .LOG. The zero(s) will automatically increment by one for each subsequent sequence execution. This will enable you to compare profiles later on for selecting the best profile for your intended rework application.

These files will be stored in the indicated default directory, or another directory that you may specify. Select and name the thermocouples that you will be using. These are the thermocouples whose temperatures will be displayed during operation.

When all TC information has been entered, select OK.

Ralph Webber, November 23, 2004
12.6 **Recommended Profile Selection**

Go to the Process pull down menu and select *Times and Temps* in order to view the graphical profile screen. Select “Use Learned Auto Profile”. If an AutoProfile was created previously, it will be displayed as a purple curve on the profile display. If an AutoProfile has not been previously learned, click on the Default button to apply default profile parameters defined for the board and component (typically a very good starting point).

The red line is the temperature curve that the AutoProfile software will attempt to reproduce at the thermocouple placed beneath the BGA component. Once a profile has been learned, it is autosaved and an additional top heater temperature profile will be displayed. The purple profile segment displayed below was learned with AutoProfiling board conditioning, and aborting the sequence at 240 seconds into the learning process.

Ralph Webber, November 23, 2004
From our effort profiling the board temperature, the bottom heater settings for the sequence should have been modified to the display shown above. If this was not done, or has been changed for any reason, there are two places where the sequence must be modified. These are in the above chart and within the dialog box accessed by the Process Steps… button.

Displayed temperature boxes can be grabbed and moved by placing the mouse cursor on the desired box and holding down the left mouse button. While holding the left mouse button down with the cursor placed over a temperature box to be changed, the box can be moved up and down to change temperature. Time lines may be grabbed in a similar manner and moved left or right to change time intervals. Either change will alter the displayed ramp rate for temperature increase, or decrease, over time.

Grab the blue 225 numbers and increase bottom heater temperature to 250°C by dragging the boxes to the higher temperature. (Alternatively, double click the left mouse button while the cursor is placed on the boxed number you wish to edit. Enter the desired number in the displayed dialog box.) Press the Process Steps… button to access the board conditioning preheat sequence.

Increase the temperature from 225°C to 250°C, and increase the time interval from 60 seconds to 90 seconds. Click the button Learn Profile (or Learn AutoProfile with Summit 2100 systems) to use the modified profile and begin the learning process.

Ralph Webber, November 23, 2004
12.7 Center Component in Nozzle

Follow the sequence of screen prompts to center the component within the nozzle. The image displayed below was obtained with the optics magnification dial set to 0.63 while using a 1X coupler. Achieving a good contrast of the superimposed images from the split mirror may require adjustment of the two separate light sources for Component and Board Lighting. (Keep in mind that the split mirror is simultaneously looking up into the nozzle and down at the board.)

![Figure 12-13 Example of Centering a BGA Component](image-url)

12.8 AutoProfile

Click on GO to start AutoProfiling. The software will proceed to walk you through the process of aligning the reflow site and component in the top heater nozzle before performing the AutoProfiling and datalogging.

12.9 Real-Time Viewing and Interpreting AutoProfiles

The following screen appears when AutoProfiling begins. In order to view the AutoProfile heating cycle as it is generated, click on the graph icon at the top of the screen. The temperature will be plotted as the learn sequence is executed. The red curve represents top heater temperature, blue represents bottom heater temperature, and green is...
the temperature profile achieved at the component as measured by the thermocouple placed beneath the component.

![Graph Image]

It is interesting to look at two small blips that are circled on the green profile. These temperatures were measured by the thermocouple placed near the solder balls of the BGA and correspond to approximately 183°C, which is solder liquidus. On the way up in temperature, at the point at which the solder melts, energy is absorbed. Upon cooling (see enlarged view), at the point at which the molten solder solidifies, energy is released. This is what causes the slight decrease and increase in temperature at these points in the thermal profile.

The maximum difference in temperature between the top heater and component (Δt, “delta T”, indicated on the graph) should ideally be kept to no more than 120°C. For some particularly sensitive components, it may be desirable to keep this Δt even smaller. However, as described below, this Δt can be evaluated by using the Graph Utilities software feature. Note that the indicated file name on the top blue bar (SRT027.LOG) is consistent with the discussion on file names earlier.

The next time any profile is used, or relearned with a new set of parameters, the file name will automatically index to SRT028.LOG, unless manually changed. All of these files are automatically stored in the default directory C:\SRTWIN32\DATALOG. Datalog files can be stored under different directories and file names by accessing the Datalog _ Setup

Ralph Webber, November 23, 2004
menu, or at any time by using Windows Explorer to move files to new folders of your choosing. You can also select, query, and print datalog graphs from a dropdown list of stored graphs that can be accessed from the Datalog Graph Utilities Program.

The following Graph Utilities screen was accessed from the Datalog \ Graph Utilities Program pull down menu. The auto saved SRT027.LOG file plot can be interrogated to evaluate critical process parameters.

![Graph Utilities Screen]

**Figure 12-15 Process Parameter Graphing**

The first parameter to be examined was the length of time that the BGA component solder balls were above liquidus. The dotted pink line indicates Liquidus, the temperature at which the solder melts. By placing the cursor on the plot and clicking the left mouse button, Line A appears. Line A can be moved into position by holding down the left mouse button and dragging the line until it intersects the point where the component temperature line (green) crosses the dotted liquidus line during temperature ramping.

Right clicking the mouse button when the cursor is on the graph creates line B. This line can be moved into position by holding down the right mouse button and dragging the line until it intersects the point where the green component temperature plot crosses the dotted pink liquidus line during cool down.

Ralph Webber, November 23, 2004
After Lines A & B are created, the table below the graph will display additional data. The BGA ball data from thermocouple #3 (green line), shows that the distance between lines A and B represents 77 seconds.

This is the time interval during which the BGA solder balls were above liquidus and is in line with recommended reflow durations of 45 – 90 seconds.

Line A also crosses the plot of both the top heater temperature (red line) and the component temperature (green line) where the delta is greatest between the two temperatures. From the table we can see that the two temperatures at these intersections are 235°C and 185°C. The \( \Delta t \) is equal to 235 - 185 = 50°C, well below our maximum target range of 120°C.

As shown below, Line A can be moved to any position over the plot to look at other \( \Delta t \)’s, particularly at points where the top heater temperature reaches a maximum.

![Figure 12-16 Process Parameter Temperature Comparisons](image)

In the above plot, Line A was positioned at the point where top heater temperature peaked. From the table of displayed data, \( \Delta t = 252 - 210 = 42°C \), well within our \( \Delta t \) target. Another important process variable that can be evaluated from these plots is thermal ramp rate, or slope. This is the rate at which temperature is increasing, or decreasing, in degrees Celsius per second. When the ramp rate for temperature increase

Ralph Webber, November 23, 2004
has been set to a value that cannot be transferred to the component (> 2°C/sec), the aggressive thermal slope and maximum temperature of the top heater may overheat the top surface of the component. This can also reduce time above liquidus such that the solder does not have sufficient time to create a good solder joint. It will also expose the top surface of the component to potentially damaging temperatures. Reducing the top heater ramp rate to less than 1.6°C/second, and/or raising bottom heater temperature, is the preferred method for avoiding these problems.

In the plot below we will examine the slope of component temperature increase during heating.

![Graph of component temperature increase](image)

**Figure 12-17 Analysis of Component Temperature Increase**

Lines A & B were placed in an area that appears to show the highest rate of temperature increase for the component (green line). The enlarged view shows the slope and the data table, which was automatically created by the software, indicates that this slope is 1.2°C per second. This is well under the desired maximum slope of 1.6°C per second for the component. This step has given you a feel for the ways in which the datalogging plots can be evaluated.

From this type of evaluation you will be able to modify your AutoProfile rework sequences to achieve better results.

Ralph Webber, November 23, 2004
12.10 Viewing and Using Saved AutoProfiles

When the autosaved graph of the process that was learned is displayed, the screen appears as follows.

![Graph showing saved auto profiles](image)

Figure 12-18 Viewing of Auto Saved Temperature Graphs

On this screen the profile shown in purple indicates the top heater temperature set points that will be used on subsequent runs when the *Use Learned Auto Profile* box is checked and the Run button is clicked.

12.11 AutoProfiling Summary

AutoProfiling can accelerate the creation of an optimum thermal cycle for solder reflow during component attachment or removal from circuit boards. This is accomplished by conditioning the board to an appropriate intermediate temperature, and allowing the software to “learn” a suitable top heater power cycle. The technique is controlled by actual solder-joint temperatures and relies upon the software to record optimal power settings to heaters for later recall and reuse.

Ralph Webber, November 23, 2004
There are four process goals of AutoProfiling that are used to guide the development of optimum rework sequences. These goals include;

1. obtaining a board temperature > 100°C prior to beginning component top heating,
2. using a starting temperature set point for the top heater that is close to the board temp,
3. limiting the ramp rate for the top heater to < 1.6°C per second, and
4. controlling component / top heater temperature delta to < 120°C.

These goals will help you achieve high quality and reliable rework approaches that will;

1. avoid thermal stress-induced board distortion,
2. limit the peak temperature at the top of the component,
3. optimize the time interval above solder liquidus, and
4. minimize rework cycle time.

13 BGA Inspection Process

The inspection of all of the BGA joints is not possible on area array devices. With the use of a combination of both transmissive X-ray and endoscopic inspection, most BGA rework defects can be identified.

Transmissive X-ray systems can be used to identify several potential defects. Solder bridges are the most common defect which can be detected by this technique. Large voids can be seen at greater power and magnification levels. However X-ray systems have limited ability to detect opens.

A supplemental tool to the X-ray system is an endoscopic inspection tool. This tool allows hidden solder joints or joints that are in close proximity to other nearby parts to be inspected and characterized. The ability to see the integrity of a joint condition can be accomplished with this inspection tool. One of the major benefits of using an endoscope is its ability to view underneath the grid array and the surface of the individual solder balls. The quality of joints including its texture, uniformity, smoothness, color and brightness and surface characteristics can be documented with this tool. A poor solder joint with micro cracks on its surface could be seen with the endoscopic inspection tool while not showing up on an X-ray system.

13.1 X-ray Images

An X-ray image is generated by different X-radiation absorption of various parts of the object. Simply spoken, these absorption differences are due to variations in density or thickness of the object. For solder joints, mainly the thickness (or shape) is of interest. The longer the path of the radiation through a part of the object, the darker this part appears in the image.

Any defect that has a remarkable influence on a solder joint's shape can be detected with X-ray technology. Some defects such as bridges can be seen directly, others can be
detected by their signatures that can be used to define test criteria as listed for a top-down view in the table.

The correspondence of the signatures to the various defects is known from experience, but can also be understood from simple geometrical considerations. For example, if a single BGA solder joint appears brighter than its neighbors, it is obviously not as thick and, hence, must be open (not properly soldered). If the joints in the center of a BGA are broader and brighter than those at the edges, most likely the package is more or less bowed (substrate or board warpage), and presses down the central joints so that they become broader and thinner.

![Figure 13-1 Examples of Defects and Appearance in X-ray Inspection](image1)

### 13.2 Wetting Analysis

A more subtle task is the analysis of the wetting quality at both board and component pad which means to check whether the solder is really in contact with the pads. The related test criterion depends on whether the joining pad area is defined by a solder mask overlapping the pad or by the etched copper pad itself. In the latter case, well-wetted pads are embedded into the solder. In the X-ray image, characteristic dark rings at the edges of the pads are visible which are due to the additional solder in this area (Figure 13-2).

![Figure 13-2 Dark Rings Indicating Additional Solder](image2)

Ralph Webber, November 23, 2004
Joining pads appear as dark circular areas if they are well wetted. In practice, the related differences in thickness must exceed 2% of the absolute solder thickness to be visible with an image intensifier.

Though this limit can be lowered to about 0.5% by using digital X-ray detectors, the signatures of wetting defects can be very faint in some cases and might be concealed by the barrel-shaped solder joint itself. The delectability for such fine structures can be enhanced remarkably by just reducing the absolute solder thickness to be penetrated, for example, to take an oblique view of the solder joints. In this way, not only the wetting status of the pads becomes plainly visible but the shape of the solder joint is clearly displayed as well.

13.3 Identification of Bridging

Occasionally there is a tendency of larger (and thinner) BGA components to warp during rework. The type of warping can cause bridging and shorts at the outer corners of the device during replacement creating a requirement for more rework at the BGA site.

It’s generally understood that some minor distortion occurs in BGA components and circuit boards when heated, due to the varying thermal coefficient of expansion (TCE) of all the different materials. In the BGA component, this effect appears greatest at the outermost edges, the furthest area from the center of the BGA package. The larger the BGA, the greater the deflection can be at the outer corners.

The warping can be very slight, the center of the BGA will bow upwards and the corners down. This is often enough to cause the corners to move closer to the circuit board surface, putting extra pressure on the corner solder joints as they flow. The result is that the solder balls at the corners pancake to the point that they may touch and blend with one another (bridging). (see Figure 13-3). Warping may also occur in the opposite direction.

Figure 13-3 Solders Balls at the Corners of BGA Components can Pancake

Once bridged or shorted, solder balls will remain in that condition due to the surface tension inherent in the molten solder. This problem is usually not detected until after rework, during X-ray inspection. At that point, one realizes that the rework procedure was unsuccessful and must be done all over again, subjecting the site to more thermal
cycles, etc. and risking pad degradation and other problems.

Also consider that the circuit board itself may warp slightly and this additional warp if combined with that of the BGA can amplify the problem. Of course the first thing to look at is the profile itself, but even when profiles are tweaked just so there can still be the occasional problem at the corners due to inherent limitations of the rework process from original oven reflow. The solution to this problem is to support the outer corners of the BGA during the critical time that the BGA solder balls are molten.

Measure the clearance between the circuit board surface and the underside of the corner or edge of a properly soldered BGA exactly like the one that is to be reworked. Select small pieces of non-solderable material to serve as spacers. The thickness of these spacers is slightly less than the component standoff height in its post-reflow state. The spacer is made to prevent bridging and not to hold the component at an arbitrary height above the board.

As the new BGA is ramped up to reflow temperature under a hot gas reflow machine nozzle, the spacers prevent the corners from flattening the balls underneath. As the BGA cools, and the warp decreases, the outer edge balls solidify, and the solder connections beneath the center solidify as well. These spacers can be easily removed once the BGA and board have cooled to ambient temperature.

13.4 Common Defect Characteristics

13.4.1 Opens
Opens are usually caused by insufficient reflow, missing balls, doming, popcorning or contaminated board surface conditions. This defect is found much more easily if the board is tilted on its X and Y-axis. This provides a view of the side of the joint, thus rendering a better angle into the suspected area of defect.

13.4.2 Shorts
Shorts are easily found using X-Ray, although on some double sided boards you will see a component from the backside giving the appearance of a solder bridge. This situation can be easily cleared up by rotating the board along the X and Y-axis. This will separate your image distinguishing the front side with the backside components.

13.4.3 Insufficient Reflow
Insufficient reflow is usually a little more difficult to spot. Some of the characteristics of a solder joint, which has not reflowed properly, include rough grainy appearances on the edges of the solder joint and an irregular shape of the joint.
13.4.4 Doming

Doming is a defect common when a package has not been stored in nitrogen or other types of non-humidity chambers. The characteristics of the doming effect are the center solder joints of the BGA are slightly smaller than those on the outside edge. When searching for this defect, be certain to bring the entire BGA through the center of the camera. This allows you to eliminate the parallax error caused by the camera lens.

13.4.5 Potato Chipping

Potato chipping occurs when a component's outside edge lifts up from a pad. This will cause the center joints to appear squashed as a result of overheating the component. This defect can be found by rotating the board on its X and Y-axis allowing you to see the hourglass appearance of the outside edge and corner solder joints. In addition, a squashed view of the center joints may also be apparent.

13.4.6 Voids

Voids are usually found in boards, which have not been above reflow long enough. They are typically created by flux gases that are unable to escape. The best way to view voids is to use a lower voltage and amperage, while the board lays flat along the X and Y-axis. An increase in these settings may cause voltage blooming. You can also adjust the X-ray contrast to expose the voids better. The appearance of a void will be lighter in contrast then that of the solder ball.

13.4.7 Popcorning

Popcorning is a violent eruption of moisture from within the package. The eruption may break the die from the substrate; it may pull wire bonds from pads or delaminate the package substrate itself. Plastic packages absorb water from the air. The propensity of the package to absorb water is established through testing. The current industry standard for this testing is J-STD-020B. Packages that have been classified using the standard should be handled according to J-STD-033A.

Proper handling requires users to store parts in special vapor barrier bags with desiccant. If packages are exposed to moisture, the factory air, it may be necessary to bake the parts for some time to drive moisture out of the pack age.

Popcorning of the package may not be visible in X-Ray inspection. X-Rays pass through silicon and epoxy easily. So, if the die is popped off the substrate we may not see that type failure under X-Ray. If the substrate delaminates and blisters, localized swelling may deform some of the solder connections. We have observed this in the form of irregular sized balls around the site of the package failure.

Ralph Webber, November 23, 2004
14 References


[2] South Bay Circuits, Manufacturing Guidelines for Printed Circuit Board Assemblies, Sbcdfm Revision A


15 Appendices

15.1 Summit 2100 AutoProfiling Custom Applications

When preparing your own boards and components for AutoProfiling, there may be a few additional steps to follow if you are using a Summit 2100 rework system. There are two additional pages of information that must be considered – Board and Site.

![Figure 15-1 Screen Setup for Automatic Board and Component Positioning](image)

The pages Board and Site at the top left of the screen must be opened and completed for your application if you are operating a Summit 2100 that automatically positions the board and component beneath the heating nozzle.

The Board screen appears as shown on the next page. Board dimensions can be input manually, while location can be specified by a drag and drop capability of the screen image.

Board and site locations can also be taught using the Teach buttons and jogging the sites into position.

Ralph Webber, November 23, 2004
The board and site geometry will reflect the information provided on the Board page and submenu requests for information. The above page has two teach options, one for the site locations and one for component pitch. If selected, you will be prompted to perform specific tasks that will teach the software what it requires for these data fields.

Some of the requested information is not required for AutoProfiling, but for tasks such as Scavenging.

15.2 Interpreting Profiles for Common Problems

As an aid to understanding common profile problems, the following profile features may be observed and corrected as indicated.

15.2.1 Top Heater Spike

If the starting board temperature is too low when the top heater begins heating the component, a momentary rapid increase (spike) in top heater temperature can occur at the beginning of a cycle.

Ralph Webber, November 23, 2004
Because this spike could potentially overheat the top surface of the component, it should be eliminated by allowing more time for the board to reach temperature, or by lowering the starting temperature for the top heater defined profile. Either way, it is desirable to have the top heating cycle begin at a temperature equal to a stable board temperature.

15.2.2 Controlling Top Heater/Component Temperature Spread

It is advisable to limit the temperature difference between the top heater and component temperature (as measured by TC #3 placed beneath the component) to no more than 120°C (see Figure 15-4).

It is important to remember that the component top surface temperature will be greater than the temperature measure by TC #3. Limiting the top heater and TC #3 temperature difference will minimize the thermal stress on the top surface of the component.

Reducing the defined component ramp rate will decrease the temperature difference. Alternatively, raising bottom heater temperature (blue line) will increase board temperature before beginning top heating, and will reduce the peak top heater temperature.

Ralph Webber, November 23, 2004
15.2.3 Excessive Component Ramp Rate

When the ramp rate for temperature increase has been set to a value that cannot be transferred to the component (> 2°C/sec), the thermal slope and maximum temperature of the top heater will significantly exceed those desired for the component.

This can reduce time above liquidus such that the solder does not have sufficient time to create a good solder joint. It will also expose the top surface of the component to potentially damaging temperatures.

Reducing the top heater ramp rate and/or raising bottom heater temperature so that component thermal transfer capability is not exceeded are strongly recommended.
15.3 Ten Step AutoProfiling Quick List

The Ten Step AutoProfiling approach that was presented in the Technical Bulletin is summarized in this appendix. For each step there is a brief explanation of the requirements. Refer to specific step sections in the Technical Bulletin for more detailed descriptions and instructions.

15.3.1 Board-Conditioning Temperature Profiling

Evaluate learned profiles and modify process steps and your sequence to achieve a minimum board temperature of 100°C and a temperature that is within +/- 5°C of the top heater starting set point.

Obtain a board conditioning temperature profile for the rework preconditioning process step using the system default auto profile. Attach a thermocouple to the center of the rework site that will be profiled. Use the sequence editor to set up datalogging of board heating.

Ralph Webber, November 23, 2004
15.3.2 AutoProfiling Board Preparation

Attach the BGA 225 to the SRT demo board using the baseline Place_Reflow sequence for this component, board and site. Install the 6 mm (1/4”) pickup tube and 27 mm Pro-Series nozzle for this purpose. Following component reflow and cooling, insert the thermocouple beneath the component for autoprofiling.

15.3.3 Select Board Location

Centrally locate the board on the X-Y table and plug the autoprofiling thermocouple into the T/C 3 port.

15.3.4 SierraMate Software Setup

Use the main 1-2-3 GO screen to enter board and site information. Select the Reflow Site sequence for AutoProfiling.

15.3.5 Datalogging

Set up datalogging using T/C 3 to report BGA solder ball temperature.

15.3.6 Recommended Profile Selection

Use the default sequence and select Learn Profile from the screen accessed from the Process, Times and Temps pull down menu.

15.3.7 Center Component in Nozzle

Follow the sequence of screen prompts to center the component in the nozzle.

15.3.8 AutoProfile

Click on GO to start AutoProfiling.

15.3.9 Real-Time Viewing and Interpreting AutoProfiles

Evaluate the resulting AutoProfile in order to determine acceptability.

Interrogate the AutoProfile in the Graph Utilities program to evaluate whether critical process goals have been achieved.

Examine whether the temperature difference between the top heater and component were held to less than 120°C.

Evaluate component time over liquidus to verify that it falls between 45-90 seconds.

Ralph Webber, November 23, 2004
Modify the sequence as required to bring the learned AutoProfile closer to the parameters you desire.

### 15.3.10 Viewing and Using Saved AutoProfiles

View the screen from the Process; Times and Temps pull down menu. The profile shown in purple indicates the top heater temperature set points that will be used on subsequent runs when the Use Learned Auto Profile box is checked and the Run button is clicked.
# 16 Index

- Reflow sequence · 40, 44

## A

*Applying Solder Process* · 36  
AutoProfile · 47, 51, 70  
AutoProfiling · 39, 48, 51, 53, 58, 65, 69  
AutoProfiling Board Preparation · 47

## B

BGA Inspection Process · 59  
*Board and Site* · 65  
Board Design · 17  
Board Location · 70  
Board Preparation · 70  
Board-Conditioning · 40

## C

capture pads · 11  
Center Component in Nozzle · 53, 70  
Circuit Board Sensitivity · 24  
*Cleaning Process* · 35  
Cold Start Profile · 45  
Component Ramp Rate · 68  
Component Sensitivity · 24  
Component Spacing Considerations · 22  
Connector · 27

## D

*Datalog* · 50  
*Datalog Setup menu* · 55  
*Datalog Graph Utilities Program* · 55  
Datalogging · 70  
DataLogging · 50  
*Development Process* · 33  
Dispense · 36

## E

endoscopic inspection · 59  
ESD · 24  
ESD Sensitive Devices · 24

## I

Identification of Bridging · 61, 63  
Inspection Considerations · 21

## L

Learn Profile · 44, 47, 52  
Liquidus · 55

## M

Metal Stencils · 36  
Mini Series Nozzles · 27  
Mini-Series Nozzles · 31

## N

*Nozzle Selection* · 26  
NSMD · 5, 7

## P

Pad Surface Finish · 14  
*Part Reflow Process* · 38  
*Part Removal Process* · 35  
PCB · 5  
PCB Preheating · 25  
PCB Stack · 15  
PCB warpage · 38  
pickup tube · 26  
Pickup Tubes · 29  
Place Reflow · 47  
*Positive Air* · 26, 29  
Pro Series Nozzles · 27  
Pro Step Nozzles · 27

Ralph Webber, November 23, 2004
<table>
<thead>
<tr>
<th>Process Considerations</th>
<th>20</th>
</tr>
</thead>
<tbody>
<tr>
<td>Process Steps</td>
<td>46, 52</td>
</tr>
<tr>
<td>Profile</td>
<td>33</td>
</tr>
<tr>
<td>Profile Selection</td>
<td>51</td>
</tr>
<tr>
<td>Profiling</td>
<td>34</td>
</tr>
<tr>
<td>ProSeries nozzle</td>
<td>47</td>
</tr>
<tr>
<td>Pro-Series nozzles</td>
<td>28</td>
</tr>
<tr>
<td>Pro-Step Nozzles</td>
<td>31</td>
</tr>
<tr>
<td>PTH</td>
<td>9</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>R</th>
</tr>
</thead>
<tbody>
<tr>
<td>Raised deflector skirts</td>
</tr>
<tr>
<td>Reflow Considerations</td>
</tr>
<tr>
<td>Right Angle Connector Nozzles</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>S</th>
</tr>
</thead>
<tbody>
<tr>
<td>Saved AutoProfiles</td>
</tr>
<tr>
<td>Select Board Location</td>
</tr>
<tr>
<td>Select Component</td>
</tr>
<tr>
<td>Semi-Permanent Stencils</td>
</tr>
<tr>
<td>Shim Stock Coverage</td>
</tr>
<tr>
<td>SierraMate Software</td>
</tr>
<tr>
<td>SMD</td>
</tr>
<tr>
<td>Static Shielding</td>
</tr>
<tr>
<td>Stencils Flexible Removable</td>
</tr>
<tr>
<td>Straddle Mount Connector Nozzles</td>
</tr>
<tr>
<td>Sub-Plenum Nozzles</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>T</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature Profiling</td>
</tr>
<tr>
<td>Thermal profiling</td>
</tr>
<tr>
<td>thermal ramp rate</td>
</tr>
<tr>
<td>Thermal Vias</td>
</tr>
<tr>
<td>Thermocouple Placement</td>
</tr>
<tr>
<td>Top Heater Spike</td>
</tr>
<tr>
<td>Transmissive X-ray</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>V</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vertical Connector Nozzles</td>
</tr>
<tr>
<td>Via Density</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>W</th>
</tr>
</thead>
<tbody>
<tr>
<td>Warm Start Profile Temperatures</td>
</tr>
<tr>
<td>warping</td>
</tr>
<tr>
<td>Wave Solder Considerations</td>
</tr>
<tr>
<td>wettability</td>
</tr>
<tr>
<td>Wetting Analysis</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>X</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-ray Images</td>
</tr>
</tbody>
</table>