exposure. Maintaining proper control of moisture uptake in components is critical to the prevention of "popcorning" of the package body or encapsulation material. BGA components, before shipping, are baked dry and enclosed in a sealed desiccant bag with a desiccant pouch and a humidity indicator card. Most BGA components are classified as a level 3 or level 4 for moisture sensitivity as per the IPC/JEDEC Spec J-STD-020, "Moisture/Reflow Sensitivity Calculation of Plastic Surface Mount Devices."

With most surface mount components, if the units are allowed to absorb moisture beyond their out of bag times for their moisture rating, damage may occur during the reflow process. Chapter 8 of this data book provides an in-depth view of package preconditioning methods and moisture sensitivity requirements. Please refer to Chapter 8 for more information regarding how moisture sensitive components are classified.

Prior to opening the shipping bag and attempting solder reflow, the moisture sensitivity of the packages being used should be understood so proper precautions can be taken to insure that a minimal out of bag time is maintained. This will insure that the highest possible package reliability is achieved for the final product. If previously bagged product cannot be mounted before the elapsed out of bag time for that product, the parts can be rebaked as per Chapter 8. Another option is to store the opened units in a nitrogen cabinet or dry box until needed. Placing units in a dry box effectively 'stops the clock.'

It should be understood that packages continue to gain moisture even after board mounting. Components that need to be reworked must be completely processed throught all thermal exposures before the original out of bag limits are reached. If this is not possible, or the time allotment is not ridgely followed, bake-out of the completed boards must be accompliished before subjecting the components to the heat of the rework process. Products being removed from boards that have been returned from the field for failure analysis, must be baked dry before heat expousure. If this step is skipped, massive damage to the component will result, rendering useless any further efforts at determining the cause of failure.

14.8 Designing Boards For BGAs

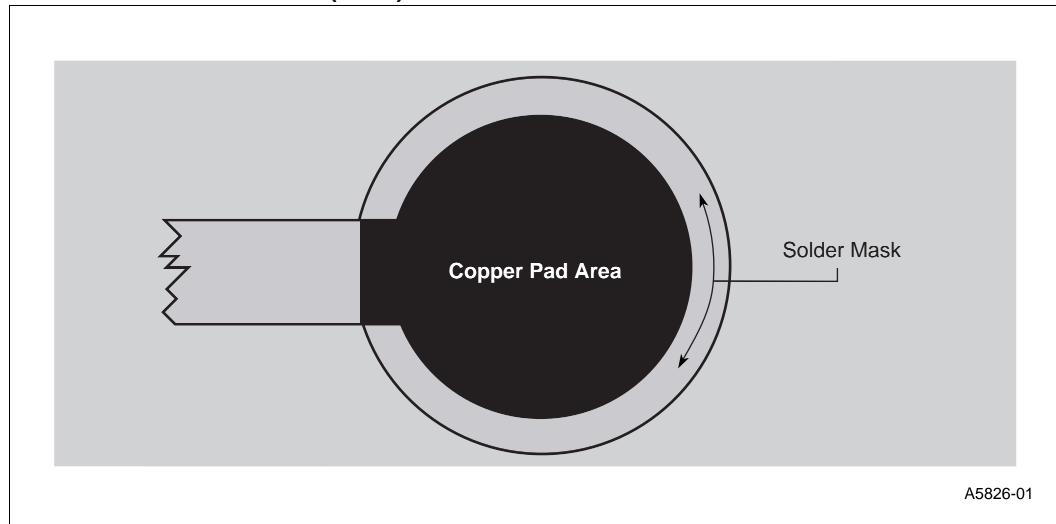
Most BGA packages use Solder Mask Defined pads on the package side of the solder ball. PCB pad size is typically close to or identical to the package pad size. This provides for balanced stress during thermal cycling, which helps to maximize fatigue life.

14.8.1 Solder Mask Defined (SMD) Pad

In the solder mask defined (SMD) pad shown in Figure 14-14, the copper for the pad area is larger than the desired land size. The opening in the solder mask is made smaller than the copper land, thus defining the mounting pad. A couple of points to consider with solder mask defined pad are:

- There is an advantage in that the overlap of the solder mask onto the copper enhances the copper adhesion to the laminate surface. When using resin systems where adhesion is low, this is an important consideration.
- One disadvanatage of SMD pads is that the fatigue life has shown to be lower then NSMD pads through long term reliaiblity testing. Because of this issue, the solder mask angle at the pad edge has been thinned on many new package designs to minimize the mask impingment on the solder ball.

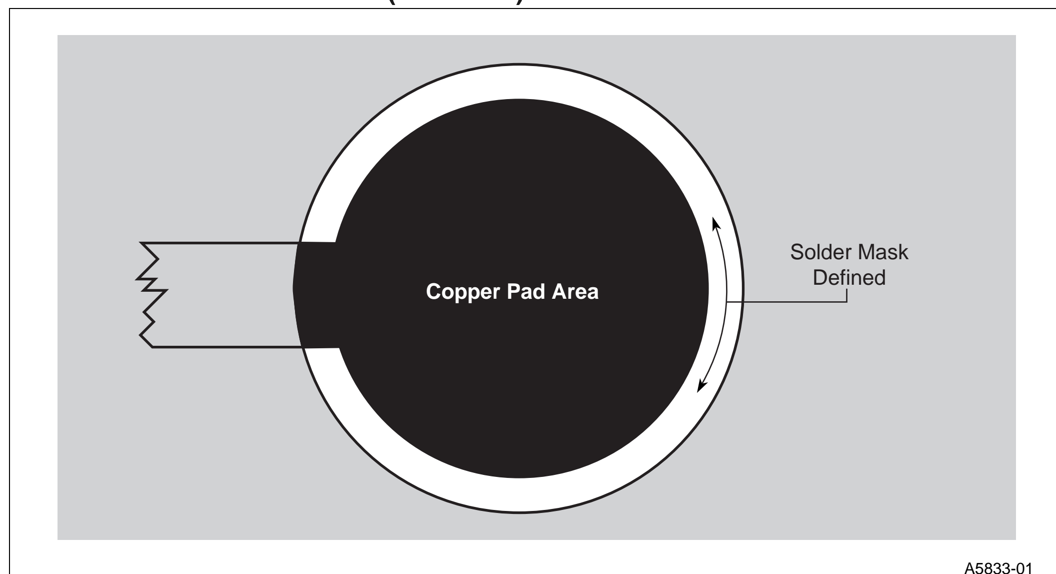
Figure 14-14. Solder Mask Defined (SMD) Pad



14.8.2 Non-Solder Mask (i.e. Metal or Copper) Defined Pad

The non-solder mask (sometimes called metal or copper) defined pad Figure 14-15, has a solder mask opening larger than the copper area. Pad size is controlled by the copper etch quality control. This is generally less accurate than the solder mask photo image control. Non-solder mask pad size varies more than with the SMD pad. However, because the edges of the copper do not need to extend under the solder mask, the pad can be either made larger, or provided more line routing space between pads. Pattern registration is also as accurate as the copper artwork, which is generally much more accurate than the solder mask pattern. Vision registration on copper fiducials (reference points) will give exact location of the site. With SMD pads, the misrepresentation error of the solder mask will also shift the location of the entire site relative to vision fiducials.

Figure 14-15. Non Solder Mask Defined (Non-SMD) Pad



14.8.3 BGA Package Considerations

The following subsections address the BGA package layout and includes guidelines for pad size, vias and routing. It is important to implement a keep-out zone around BGAs for rework purposes. The keep-out zone distance is determined by the type of rework equipment to be used. See Section 14.9.8 for more information.

14.8.3.1 Routing and Pad Size

Many perimeter style BGA package typically contains four or five rows of solder balls; as such, it is possible to route one or two traces between pads to route signals in a four-layer board.

- When routing one trace between pads, the manufacturers preferred line width and spacing technology becomes a limiting factor.
- When routing two traces between pads, 5 mil traces and 5 mil spaces are required for a 24 mil pad size. For a 20 mil pad size, 6 mil traces and 6 mil spacing can be used. Figure 14-15 shows a routing example for 20 mil ball pads. Either pad size is acceptable, so the decision is primarily determined by the manufacturers preferred line width and spacing technology.
- When larger trace widths are desired, another alternative is to route 5/5 or 6/6 (mil space/mil trace) within the BGA pads, and then neck up to the larger trace widths once you have cleared the BGA component

Using the routing scheme shown in Figure 14-16, the first two ball rows are routed on the top signal layer and the inner two rows are routed on the bottom side of the package substrate. In this case vias are required between the BGA pads. Using the routing scheme shown in Figure 14-10, the first three ball rows are routed on the top signal layer and the inner row is routed on the boards bottom side. In this case the vias are not required in between the BGA pads. Vias are discussed in Section 14.8.3.3.

Figure 14-16. Board Side BGA Routing Example One

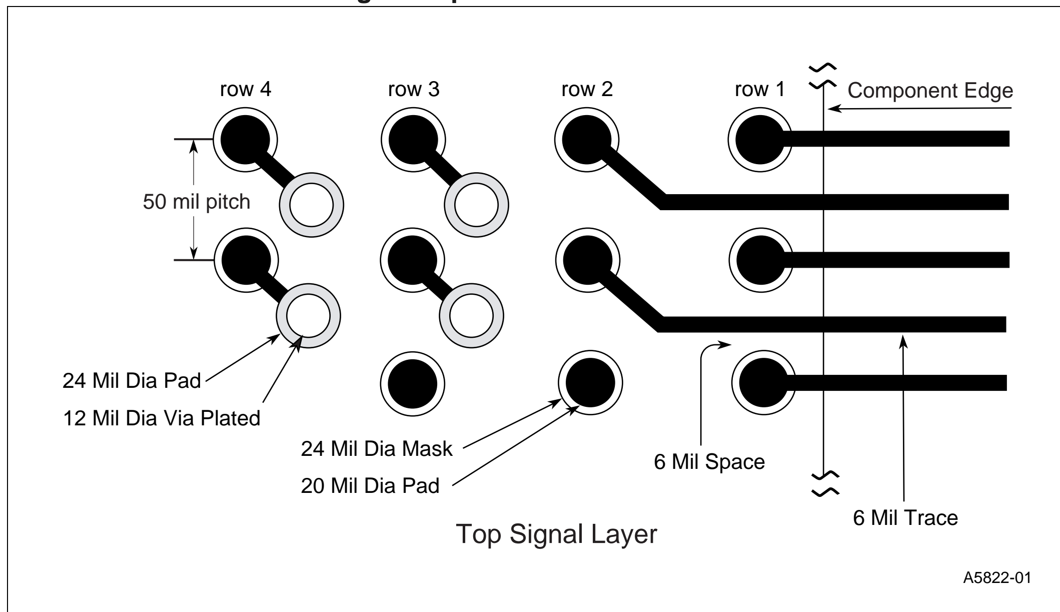
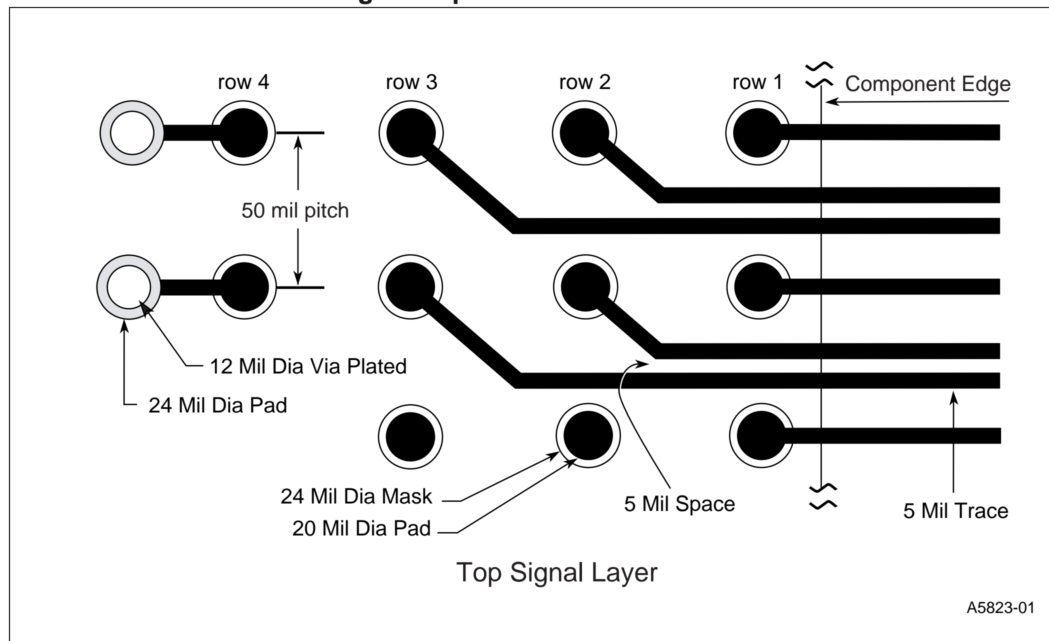


Figure 14-17. Board Side BGA Routing Example Two



14.8.3.2 Bottom Layer Routing

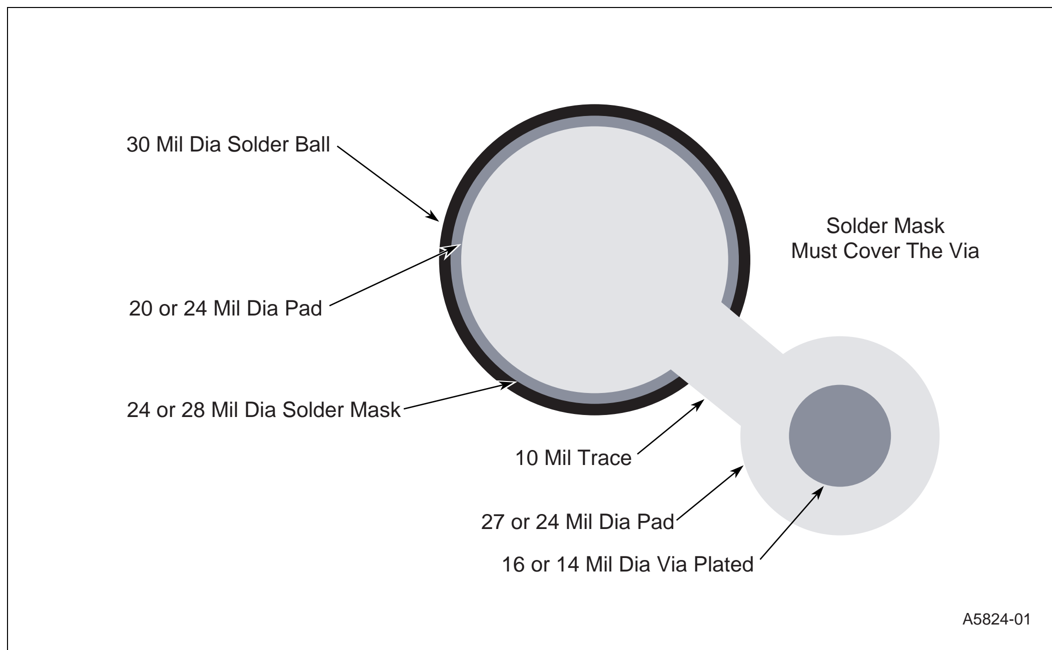
Figure 14-16 shows a routing example for the solder side of a four-layer board. The first two rows (R1, R2) are routed on the component side, while the inner two rows (R3, R4) are routed on the board's solder side. This example shows 6 mil trace widths.

14.8.3.3 Plated Through Hole (PTH) Isolation

Regardless of the technique used for the mounting pads shape or definition, isolation of the plated through hole (PTH) from the mounting pad is important. If the PTH is contained within the mounting pad, solder will wick down the PTH. The amount of solder that wicks depends on many factors, including PTH finish and coating variations. Because of this, the results are somewhat unpredictable. Some solder joints may be unaffected, while others will be starved to the point of creating opens. The worst result is a partially starved joint with severely reduced cross section. This joint can have significantly lower fatigue life and result in early system failure. Because the quality of the solder joint is guaranteed by control rather than inspection, designs/processes that result in random distributions are generally considered unacceptable, and the PTH-in-pad design is not suggested. All vias located between the BGA ball pads must be covered with solder mask. It is suggested that, at minimum, vias on the top side be covered with solder mask. The bottom side can also be covered.

Figure 14-14 shows the connection between the BGA ball pad and a via. This connection is often referred to as the dogbone footprint.

Figure 14-18. BGA Pads and Vias



14.8.3.4 PCB Quality and Co-planarity

The assembly yields of mounting BGA components is influenced by the PCB properties and process control procedures followed by the manufacturer of the PCB. PCB co-planarity requirements are directly related to the package size. Typical PCB manufacturing specifications allow up to 0.01 mm (1%) of warpage. For a 35 mm component, this would equate to nearly 0.35 mm of warpage in the area under the package footprint. It is obvious that no large body (>20 mm) component, peripheral leaded or BGA, would consistently solder to a site with this amount of bow. Most responsible PCB vendors take precautions to be well below the 0.010 mm specification that is recommended in the industry standard specifications.

14.8.3.5 Solderability Testing of BGA Components

Standard solderability test methods for through-hole and other leaded devices aren't suitable for testing BGA components. A 'dip and look' process will strip much of the solder ball off the substrate surface, leaving little to examine for determining if the solder surface was wettable. A wetting balance will only be able to sample specific solder balls on the substrate, which can be an acceptable method for sampling, but would take a considerable length of time to do the whole surface of high lead count devices, and would still only be representative of one unit. The untestablility of production BGA packages has led most suppliers to further develop their ball attach processes to insure minimal solder oxidation occurs. In some instances, this means doing all test and burn-in operations before ball attach. Where this is impossible, it means establishing stringent control over all operations after ball attach that would impact solderability. These operations, which usually include electrical testing, burn-in, inspection, bake and bag, shipping, storage, etc., are characterized to minimize oxidation and solder ball damage. However, with today's fluxes and controls on reflow furnaces, it is very unlikely that board mount problems can be traced to solder ball oxidation. If proper solder type, viscosity, screen print, and reflow practices are followed, very high board mount yields are being obtained even with product that has been stored for a couple of years on warehouse shelves. Most product is used within one year from the manufacturing date, which is also the limit for the desiccant in the moisture barrier bags. If solderability problems persist, a careful evaluation of the PCB solder pads should be done to insure that there is proper wetting occurring on both sides