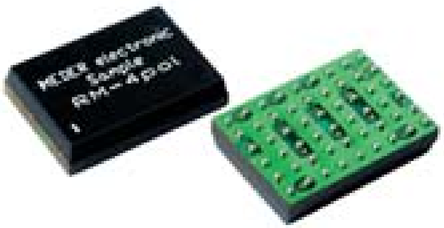
MEDER electronic

Products for tomorrow...

BGA Series

CRF Series
High Frequency
Reed Relays

CRR Series
SPST Reed Relays



RM Series
4pol Relay Module

RM Series
6pol Relay Module

Table of content

I. Soldering Approaches for MEDER's CR Series

- **Introduction**

Ever since the introduction of surface mount technology, components have gotten smaller and smaller, making it more challenging for the interconnect from the component to the PCB. With smaller and smaller components having higher and higher pin counts, the problem has become even more complex. While conventional approaches are still used, Ball Grid Arrays (BGAs) have become a popular way of dealing with this issue, but care must be taken, as we will present in detail in the following application note. We will focus on both approaches, discussing their advantages and challenges as it relates to soldering our CR relay series onto a substrate.

In this application note, we will focus on both approaches, discussing their advantages and challenges as it relates to soldering our CR relay series onto a substrate.

- **Soldering Challenges**

PCBs have varying degrees of Thermal Coefficient of Expansions (TCEs). These TCEs often are much greater than the components to be connected. Also, the TCEs will differ for a PCB in its different planes or axes; and in many cases the plane on which the component is to be mounted has the highest TCE. Solder, being very malleable, will tend to absorb some of the movement of the component vs. the PCB during temperature excursions. With continuous temperature cycling, solder joint fatigue can develop, creating a rift or crack in the solder joint. Many variables play into this, some of which are: size of components, TCEs of the PCB and component, component plating and its thickness, PCB plating and its thickness, thickness and shape of the solder between the component and the PCB, expected number of temperature excursions including min and max temperatures, etc.

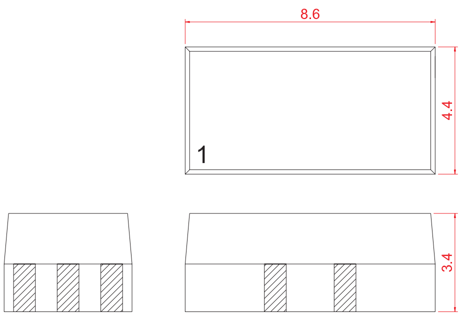
With smaller relays comes the issue of properly cleaning under the relay. If the relay is being used in a low leakage current environment (nano-amp to pico-amp range), solder fluxes may bridge between solder lands, thereby creating potential current leakage paths. BGAs represent the easiest approach to prevent this problem by propping the relay up and off of the PCB. This will allow for ease of cleaning between the relay and the PCB.

Reworking or removing of a relay can also be a trying task. However, many solder equipment companies now have equipment capable of removing and adding a component on already crowded and previously soldered PCBs.

- ### The CR Relay and its Characteristics

The MEDER's patent pending CR relay series was designed to incorporate key design features improving path length, capacitance, distributed characteristic impedance, high conductivity, and extremely low leakage currents to all points. (See Figure 1.)

Its ceramic base with very short, high conductivity, gold signal paths was chosen for several reasons: it is inexpensive, it is agreat material for plating patterns onto the substrate, hard robust packaging, the thermal coefficient of expansion matches the thermoset overmolding, and it has extremely good thermal conductivity. The thermal conductivity serves to dissipate any heat very efficiently, reducing any potential thermal offset voltages from being generated when thermal gradients are present. It also can aid in the removal of the component from a PCB by heating only one area. Here the substrate conducts the heat to all areas of the substrate allowing for ease of removal. The ceramic also eliminates the need of a costly, capacitive lead frame. The circuit connections are made on the bottom of the ceramic with a ball grid solder array eliminating the quality sensitive coplanarity and lead skewing issues when mounting to a PCB.



Non-BGA option

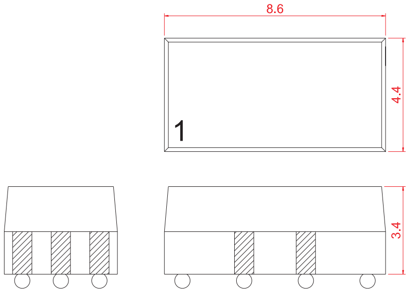


Figure 1. The CR Relay Series with and without BGAs.

TCE Matching of the ceramic and thermoset epoxy is critical for an integral design. The TCEs being closely matched will prevent any stress buildup on the fragile Reed Switch and fine copper wire making up the energizing coil.

Matching the TCE internal to the Reed Relay produces the negative result of not matching to a PCB, where TCEs for many types of PCB's range from 50 to 100 parts per million per unit length. The TCE for ceramic is approximately 10 parts per million per unit length. This mismatch in TCEs is not a major problem if, once the Reed Relay is mounted to the PCB, there is relatively small temperature variation over the life of the product. If, however, there are daily temperature excursions in the equipment, associated with turning the equipment on and off, or if the equipment is used in an outside environment where natural temperature excursions occur, the mismatch will eventually fatigue the solder connection. Here the use of solder balls is the great equalizer. The solder, being very malleable, absorbs the mismatch eliminating this potential problem. This has been qualification tested over several temperature cycles and many products to find it works very well.

II. The BGA Soldering Approach

- **Eutectic BGAs**

BGA usage is probably the best approach that satisfies the many challenges facing the user of the CR relay. BGAs are made up of grids of tiny spherical solder balls that are mass produced and capable of holding very tight spherical dimensions. The process of mounting the CR relay with BGAs to its final resting place on a PCB is a two stage soldering process: First, the BGAs must be soldered to the relay's ceramic substrate at the manufacturer's; and secondly, the relays are soldered to the user's PCB.

Figure 2 shows the ceramic bottom of the CR relay and its solder lands as well as the solder masking. The solder lands are 0.5 mm circles.

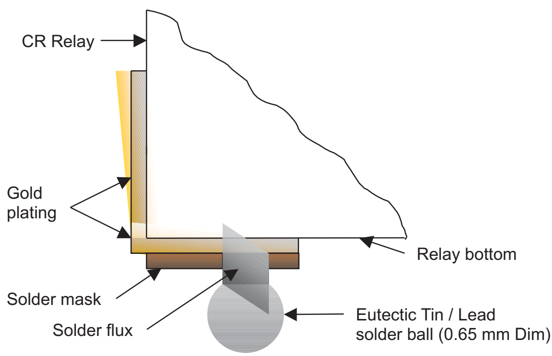


Figure 2. Flux (or eutectic solder paste) is applied to the solder lands on the bottom of the CR relay and solder balls are applied to the solder flux.

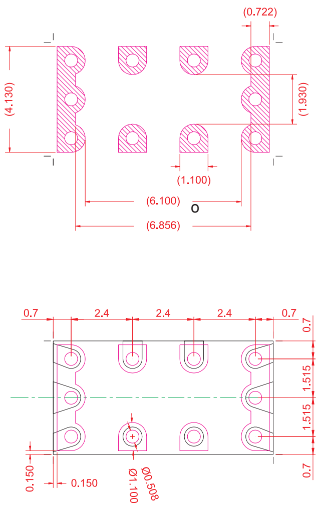


Figure 3. Bottom view of the CR Relay showing the solder lands and the solder masking

The process of soldering the BGAs to the bottom of the relay's substrate begins with the application of flux to the solder lands. (See Figure 3.) The solder balls are then automatically placed onto each flux coated solder land (or eutectic solder paste). The relays are then passed through an IR reflow oven where the solder melts and reflows onto the solder lands, re-solidifying upon leaving the reflow oven. (See Figure 4.) The solder masking plays a critical role that allows the capturing of the BGAs into their proper position. With the melt temperature of eutectic being 183 °C (360 °F), applying a peak temperature of 20 °C over this temperature should be sufficient for good solder reflow. Some manufacturers, when soldering very dense boards, prefer to reflow eutectic solder as high as 40 °C above its melt point. When going much higher than the melt temperature, care must be taken to avoid hot spots generated on the board. These hot spots can cause over temperature conditions on the relay with resulting potential failure conditions. We see this condition with some, who are always interested in minimizing set up time and maximizing throughput. Both actions can result in improper IR reflow oven profiles creating component failures. The CR relays have their own 'built-in thermocouple' with the use of the high temperature solder, which is used internal to the relay connecting the reed switch to the ceramic.

This solder will begin melting in the 270 °C to 280 °C range. Yet we have received relays back from users with solder coming out the ends of the package between the junction of the over mold and the ceramic. This indicates the relays were exposed to much higher termperatures than recommended. Over-temperature conditions can and do occur, so care in this area is critical for proper soldering.
A smarter approach for users with high PCB component concentration is to acknowledge that they may require slightly higher reflow temperatures. Then plan on running the PCBs thru the reflow oven at much slower ramp speeds.

This will give all components a chance to reach this slightly higher temperature setting. Conversely, the alternate approach can be disastrous; that is, running the temperature settings of the IR reflow oven under very high maximum temperatures, but with a corresponding fast ramp speed. This would mean the boards would progress very rapidly through the oven. Here the board throughput would be high, but also, there would be a good chance of hot spots existing during reflow. Even with thermocouples monitoring the IR reflow process, hot spots can still occur and not be detected by the thermocouples depending on where they are placed.

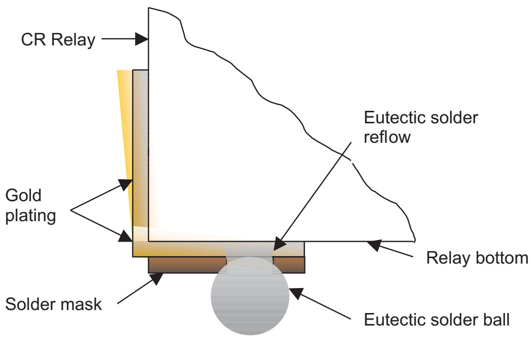


Figure 4. A view of the CR Relay after reflow soldering and the forming of a eutectic solder between the bottom of the relay and the BGAs.

The Cr relay can be provided using either one of two types of solder balls: eutectic or high temperature solder balls. In the case just described, eutectic solder which is composed of 63 % tin & 37 % lead and melts at 183 °C was used. When the CR relay with eutectic solder balls is mounted on the PCB the same general approach is used as above. First, solder paste is applied to the PCB and the relay is then mounted to the PCB. (See Figure 5.)

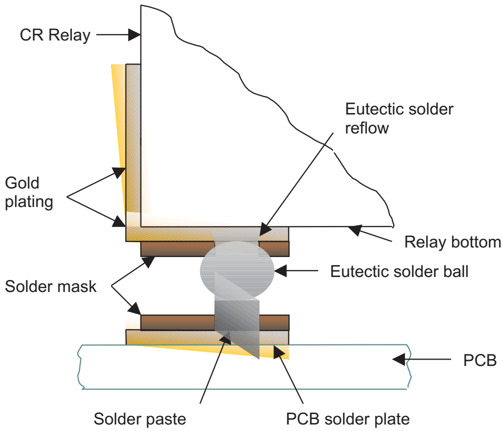


Figure 5. The CR Relay is shown placed on a PCB ready for solder reflow

The PCB then undergoes IR reflow resulting in a soldered relay with solder stand-offs. (See Figure 6.) During this process the ball will melt and form an elliptical ball based on the weight of the relay and the PCB solder land pattern. Under these conditions, the BGAs will collapse from 0.65 mm to approximately 0.5 mm, leaving plenty of room for cleaning fluxes from between the PCB and the relay. Cleaning of any flux residue left after reflow on the PCB or trapped under the relay is dependent on the flux used and its corrosive properties. No clean fluxes have become very popular; however, if low current leakage in the pico-amp range is critical, even no clean fluxes should be cleaned.

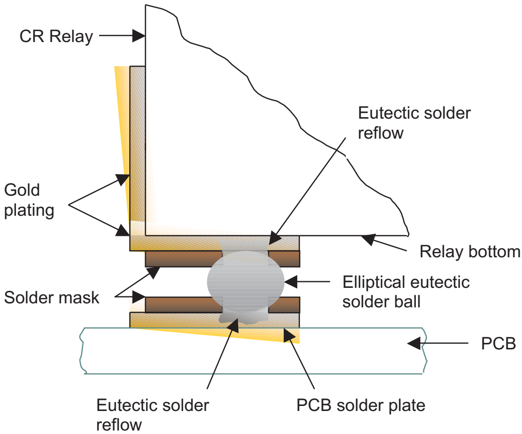


Figure 6. The CR relay has undergone solder reflow connecting the gold pads of the relay with eutectic solder to the solder ball. The solder ball is also connected to the PCB with eutectic solder.

• Coplanarity

Coplanarity is defined as lying or acting in the same plane. On the CR relay there may be 10 BGAs. Three of these balls will define the coplanarity. Therefore, the other seven BGAs will be determined to be in coplanarity or out of coplanarity based on this plane. This can mean that if even one lead is out of an acceptable range, many of the leads (BGAs), not just that particular BGA may not solder properly. (See Figure 7.) In the worst case, lack of coplanarity will cause open solder joints. (See Figure 8.)

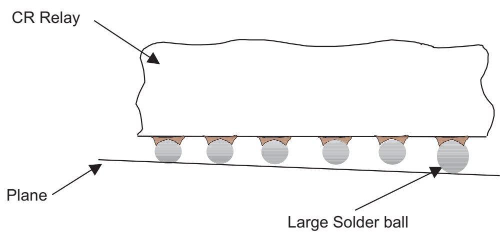


Figure 7. One can see the plane set up by the 3 BGAs that are the largest or most distant from the bottom of the CR Relay. One extra-large BGA can cause poor coplanarity with several other BGAs.

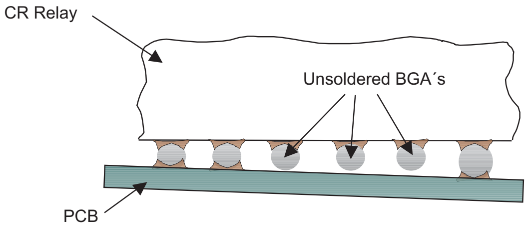


Figure 8. When the BGAs are out of the plane, lack of solderability may occur as shown.

So a user of our CR relays wants all BGAs to lie perfectly in the same plane to avoid manufacturing problems. Since it is impossible to have all the BGAs fall in a perfect plane, a specification of plane non-conformity must be established. The CR relay complies with the industry standard of 100 microns, which represents the maximum allowable distance away from the coplanarity plane. (See Figure 9.) Keep in mind, this is not a +/- number, it is a maximum distance away from the plane.

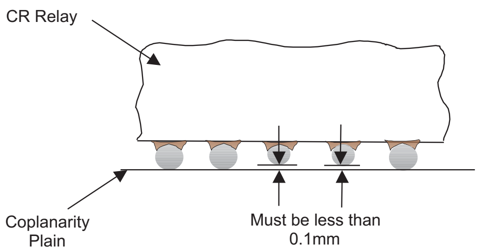


Figure 9. A typical example of the plane set up by the three BGAs most distant from the relay and how the tolerance is set up for the other BGAs. Solder paste on the PCB must be at least 0.1 mm thick to have all BGAs successfully solder.

III. The Reflow Profile

Important to any surface mount soldering approach is the solder reflow oven and its corresponding temperature profile. A typical oven profile will have a preheat zone, a soak zone, the reflow zone and the cooling zone. See Figure 10. To avoid potential relay damage, we specify a maximum temperature change of 2 °C/sec. Generally, in the preheat stage the temperature will rise from ambient to approximately 150 °C in 90 seconds to 120 seconds. Most fluxes will become activated by 120 °C. Leaving the relays in the soak zone for 90 to 150 seconds while gradually raising the temperature to 183 °C will insure a sufficient amount of time for the flux to activate all surfaces. With the surfaces to be soldered, free of any oxidation, the solder will wet to both surfaces under the best of conditions.
This wetting will occur in the reflow stage lasting from 30 seconds to a maximum of 90 seconds at a temperature of 200 °C to 220 °C.

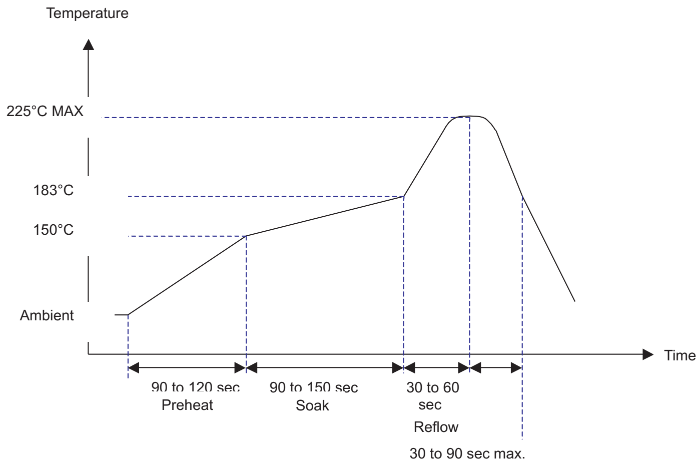


Figure 10. The recommended Temperature versus Time solder reflow profile for the CR Relay.

Using the eutectic BGAs as described above, it is worth pointing out that masking is critical on the relay itself and on the PCB. With multi-layer PCBs the copper etching can be such that interconnects are carried out using vias and sub-layering. In this case, masking may not be necessary on the PCB. It solder lands on the relay and / or

PCB are tin plated and if masking is not present, the BGAs may run over the entire solder pattern, essentially eliminating any solder balls, so great care must be taken in dealing in this area.

To maintain a tight BGA on the connection from the relay to the board, the solder mask must form a tightly toleranced circle on the bottom of the relay. This circle is the solder pad. Also, and equally critical, the PCB should contain a tight mask around the solder point restricting the potential flow of the BGA. The masking on the PCB and the relay only needs to be wide enough to restrict or eliminate the flow of solder along the conductive lands.

- **Using High Temperature BGAs**

The use of high temperature BGAs having 90 % lead will have solder melt temperatures in the 270 °C (520 °F) to 280 °C (535 °F) range. Generally, these temperatures are well above most solder reflow profiles. One of the main advantages of the high temperature BGAs is the lack of any requirement for masking on the relay or on the PCB. Here is why. Tin / lead can be combined in any number of percentages ranging from 1 % tin and 99 % lead to 99 % tin and 1 % lead. Each has it own individual properties. Generally the higher the lead content the higher the melt temperature. However, tin / lead's favorite combination is its eutectic where we have 63 % tin and 37 % lead. This is important to note, because during reflow of two different percentage combinations, when they are brought together or are in contact with each other, the junction areas of both will form a eutectic. This is exactly what happens when high temperature BGAs are used.

The process by which we apply the high temperature BGAs (HTBGAs) to the relay is the following:
We use a silk screen to apply a eutectic solder paste to the CR relay solder pads. We then automatically apply the HTBGAs. (See Figure 11.) The PCB is then placed in an IR reflow system. Figure 12 shows the results of the reflow where the HTBGA forms a eutectic bond with the relay. Again, since we are reflowing eutectic solder, a temperature 20 °C over the eutectic melt point of 183 °C is all that is necessary.

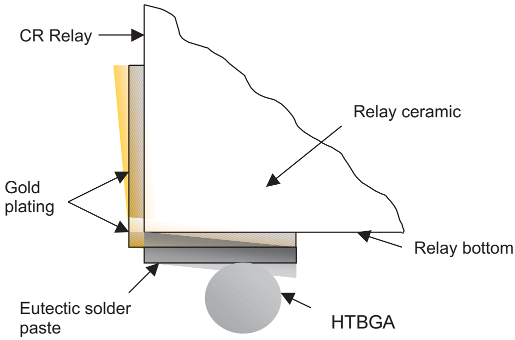


Figure 11. The CR Relay prepared for solder reflow using high temperature BGAs.

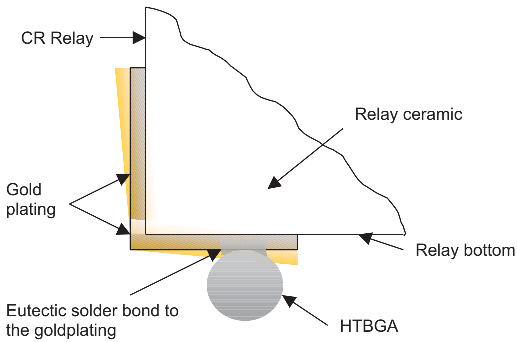


Figure 12. The CR Relay forms a strong eutectic bond between the relay solder pads and the high temperature BGAs.

Connecting the relay with HTBGAs to the PCB is the same as illustrated for the eutectic solder balls except one must use solder paste. Just using flux will not get the job done.

Not surprisingly, applying lateral force to either HTBGAs or eutectic BGAs will result in approximately the same force necessary to break it away from the PCB. This is particularly the case, when masking is used. With a masking producing a 0.5 mm diameter land on the PCB and a 0.65 mm diameter solder ball, shear forces in excess of 3.5 lbs will be the norm. This testing confirms the bonding taking place between the relay, solder and the BGA.

Using a solder mask on the PCB is not necessary when using HTBGAs; however, it can have the added safety of keeping the BGAs from wandering on the patterns. This may or may not be an issue depending on the layout, and whether one is reflowing on both sides of the board. However, HTBGAs without solder mask will have solder reflow beyond the diameter of the masking effectively increasing the shear force. So in applications where shear force can be an issue HTBGAs will be the better approach.

IV. Solder Lands and Masking Definitions with BGAs

As already stated, BGA solder lands must have a border, otherwise the solder ball will run, filling up the complete solder land. If the run area is large enough the solder ball may no longer have any height. So to prevent this, the solder land border must be masked or routed so that no solder run-off can occur. When using solder mask two items must be taken into consideration: 1. the solder mask must be large enough to prevent any bridging; and 2. if the solder mask does not enclose the entire pattern, the resulting solder ball height must be taken into consideration.

With too little solder mask, the chances of break-over of the solder exists and some run-off can occur. This becomes particularly more likely when repairs are involved. (See Figure 13.) Also in Figure 13 on the left,a proper amount of solder mask is applied, giving a good example of an adequate amount of protection against any potential run-off.

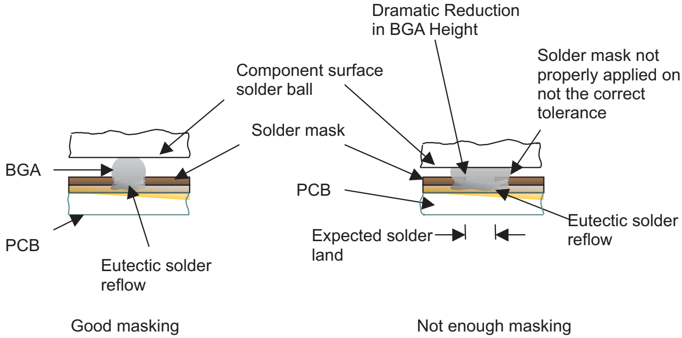


Figure 13. Comparison of proper solder masking on the PCB and when not enough solder mask exists (shown on the right) to properly protect against solder run-off. Here the solder ball height has significantly collapsed.

In the case where the solder mask does not cover the entire land, solder may flow changing the ball height. In Figure 14, on the left the solder mask on the PCB is adequate; the solder mask on the right does not cover the entire expected land area exposing part of the wire run. This will reduce the ball size potentially creating a coplanarity issue. This is shown in Figure 15.

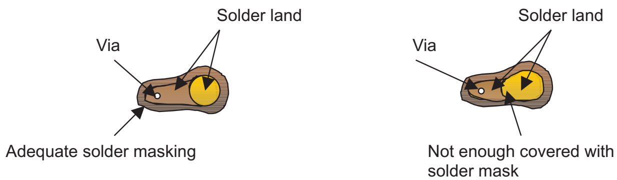


Figure 14. An example of adequate and inadequate solder lands potentially affecting the height of the BGA.

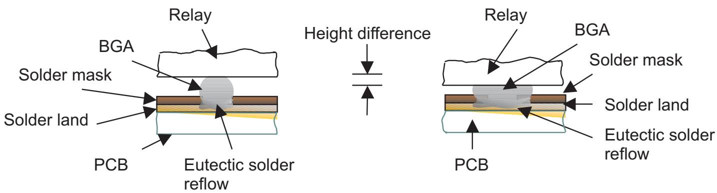


Figure 15. The solder mask on the left is adequate (refer to the left side of Figure 14). On the right the solder mask is not covering all of the solder land (refer to the right of Figure 14) reducing the height of the ball.

If a solder land is formed by cutting the board with a router, the edges must be taken into consideration, since they will effect ball height. This is shown in Figure 16. Here the sides of the metalization of the routed solder land will take on solder affecting solder ball height. If both approaches are employed on the same PCB on the same relay a coplanarity issue may be a problem. Whatever the approach, consistency is very important.

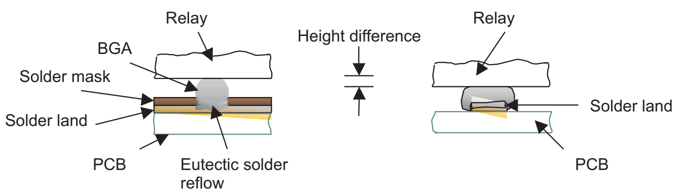


Figure 16. The effects of identical solder lands formed by masking and formed by routing. With solder wetting occurring on the sides of the routed land, a change in solder ball height will be evident.

When vias are used within a solder land they must be capped. Otherwise, the solder ball may drain into the via reducing the solder ball height. (See Figure 17.) If one's layouts do use vias in the solder lands, one must be aware that the solder ball shear force may be reduced, and there could be a slight change in the ball height. Again, on a given PCB, be consistent on a relay to relay basis. Even if properly capped, if a via is used on one of the relay lands, there still can be a small change in the solder height. If this change is within the coplanarity maximum specification, solder joints should be consistent.

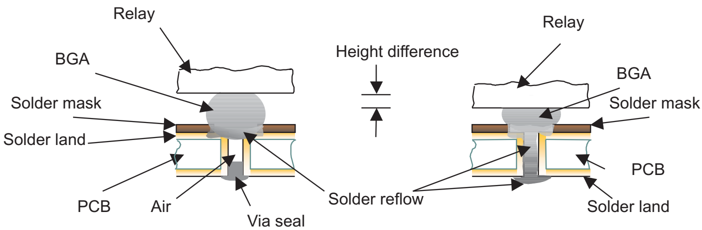


Figure 17. On the right, the via is not capped and solder during reflow has flown into and through the via, puddling on the other side of the PCB. Vias must be capped if they are within the solder land as is the case with the solder land on the left.

V. Soldering the CR Series Without BGAs

In many cases users of our relays find the conventional approach to soldering components on a PCB acceptable. This is perfectly fine as long as two potential problem areas are taken into consideration.

First, without BGAs, the spacing between the PCB and the CR relay may be inadequate for cleaning any fluxes or residue left over after the reflow operation. If acidic organic fluxes are used, any remaining residue may bridge between solder lands creating leakage paths. The insulation resistance can typically drop several orders of magnitude. Several new no-clean solder fluxes have been introduced in recent years that have very good non-conductive properties. Care must be taken in selecting the best no-clean flux for your application. Some of the no-clean fluxes will still offer potential leakage paths especially if your circuit is sensitive to leakages in the nanoamp or picoamp range.

The second consideration when soldering the CR Series without BGAs is that the voltage breakdown between solder patterns may be dramatically reduced. This again, is potentially brought about by the flux residue. If high isolation voltages are required, care must be taken that they are not compromised by the un-cleaned flux trapped beneath the relays.

The soldering operation without BGAs is straightforward. Screening eutectic solder paste on a suitably patterned PCB will accomplish the task. The hole size of the solder screen versus the solder paste thickness should be such that a minimum ratio of 1.5 must be maintained to insure proper paste size on the solder land. This is the same as when soldering with BGAs. Placing the PCB in a reflow oven will complete the process.

A major advantage of soldering without the BGAs is the ability to maintain an overall low profile. In tight spaces where component height is a real concern, soldering without BGAs may be the only alternative.

- **Soldering the CR Series without BGAs while maintaining a space under the relay for cleaning**

Forming your own space under the relay between the PCB can be accomplished in two ways. The easiest way is have MEDER supply the CR series relay with the solder mask on the bottom side of the ceramic as is done with the BGA option. (See Figure 3.)

Next and the most critical part of the process is to have a special solder paste screen made having the thickness carefully calculated. (See Figure 18.)

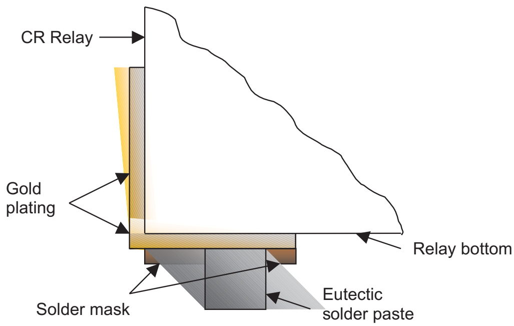


Figure 18. the solder paste volume present between the CR relay and the PCB before solder reflow.

The following must be taken into consideration: 1. the solder land diameter on the relay; 2. the solder land diameter on the PCB; 3. the screen diameter and thickness; 4. the weight of the relay and its compression effect on the formed solder ball. 5. and finally the fact that solder paste volume will be reduced by 20 % once the flux and volatiles are burned off during reflow soldering. If one is looking for a 0.5 mm spacing between the PCB and the relay, the above five items need to be taken into consideration so that the volume of solder present is enough to have the solder balls formed. The weight of the relay will compress the solder sphere formed into an elliptic pattern. (See Figure 19.)

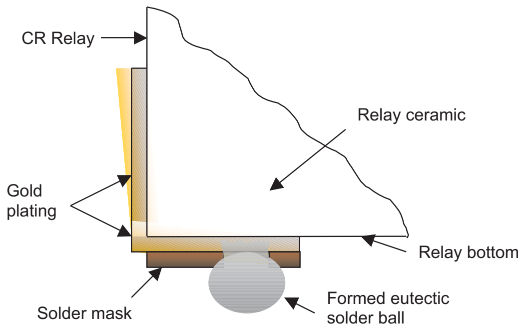


Figure 19. Once reflow occurs the column of solder paste is reduced by 20 % and reflows into a solder ball pattern.

In the above approach it is important to maintain consistent masking on the relay and on the PCB between different solder lands, otherwise, coplanarity issues will develop. Also, the height and diameter of the solder screen must be kept very consistent. If not, the solder balls formed will be inconsistent in size and shape and thereby coplanarity will suffer. If the dimensions outlined are not kept consistent and accurate to each other, maintaining a coplanarity of 0.1 mm max will be difficult.

The second approach to adding space between the relay and the PCB is to do all of the above 5 items minus the solder mask on the relay. The important point to note is that since there is no restriction of solder flow on the relay, care must be taken when fluxing the relay. If flux is carefully applied only on the area you wish to solder, then that will be the extent of the solder flow. If the solder lands on the relay were solder plated this could never happen, but the gold plating will not sustain solder flow in most cases unless there is a flux present. If the flux process is not carefully controlled, the solder balls formed will be inconsistent and coplanarity will suffer. This solder approach is the least recommended because it is the hardest to control and maintaining consistency might be a problem.

VI. Repair and/or Removal and Replacement of the Relay

The repair process is essentially the same for BGA and non-BGA relays. BGAs are a forgiving packaging/soldering approach where misalignments up to 50 % off center will realign themselves. During the reflow process the ball will realign itself. With a carefully controlled process and the correct surface mount equipment, the rework can be kept to a minimum. However, no matter how careful one is, mistakes occur and/or components fail on occasion. Fortunately there are several good rework systems available on the market that makes rework/replacement of tightly spaced components relatively easy.

To complete a repair three steps must be carried out: 1. removal of the component, 2. placing the component and 3. reflowing the new component.

To remove the component, care must be taken not to disturb the surrounding components. Use of a polyimide tape or water soluble mask can properly shield adjacent components.

- **Best Soldering Approach**

Within the global electronics industry today, reflow soldering can generally be divided into two different processes: convection and conduction. When reworking BGA components, convection is a better approach than the traditional contact methods (conductive) such as soldering irons, hot bars, and thermodes.

- **Advantages of Hot Air Pencil Over Hot Contact Tools**

The main advantage of a non-contact / hot air approach to reflow over that of the former contact type is that the air pencil simply never “touches” the component. The transfer of heat is accomplished by the hot air that is precisely directed by the pencil without contacting the part. The opposite is true of hot contact reflow tools where the tool can “stick” to SMD leads, bend SMD leads upwards/downwards causing co-planarity problems on the leaded components or introduce contaminants to the solder connection.

A convective hot air tool allows no opportunity for introducing contamination into the PCB as opposed to the hot contact tools. Also, a hot air pencil can be used to gently pre-heat the surrounding area of the targeted component for removal/replacement allowing for lower temperature settings and shorter reflow dwell times.

- **Temperature preparations for the repair process**

Two of the most critical processes which are instrumental to successful SMT & BGA tasks at the bench-top are unfortunately, the two most commonly neglected:

1) Properly preheating the printed circuit substrate before attempting removal and repair
2) Initiating a quick “cool down” of the solder joints after reflow. This applies to all bench-top work from design & prototyping and low volume production to rework and repair of the PCB assembly.

First of all, preheating or “soaking” the assembly prior to initiating reflow helps to activate the flux and removes oxide and/or surface films from the metal surface to be soldered along with extraneous volatiles from the flux itself. Accordingly, this cleansing from the activation of the flux just prior to reflow will enhance the wetting process.

Preheating also raises the entire assembly to a temperature slightly below that of the melting point of solder or the reflow point. Because this substantially reduces the delta in temperature between assembly temperature and the final reflow temperature application, the risk of thermal shock to the substrate and its surrounding components is greatly minimized. This potentially eliminates embrittling, fracturing, and cracking.

Additionally, if the entire assembly is preheated, both a reduced temperature and a shorter duration of higher temperature application at the final reflow stage are possible. This becomes very evident in cases of PCB’s with heavy ground planes and / or dense component populations where the heat sinking load of the PCB makes rework unduly slow. Without preheating, the only solution is either a further elevated temperature application and / or longer dwell time at the reflow stage.

- **Automatic Lift Off**

Automatic vacuum pick-up equipment also exists that lifts the part from the board once the eutectic temperature has been reached and continues to hold the part during the systems cooling cycle.

- **Nozzle Selection**

An appropriate size nozzle is easily installed and the correct temperature is selected. The workpiece is mounted in the board holder and the nozzle placed over the component allowing a gap of 1/8" (3 mm) above the body of the component.

Replacement of the CR relay in the same manner as it was taken off will result in a successful repair. With the variety of equipment available and following the basics, the CR relay can be easily removed and replaced from the PCB.

- **Summary**

As has been described, there are several approaches to soldering MEDER's CR Relay Series onto a PCB. Care and consistency is important for good results and coplanarity. In the end, the relay user must select the best approach for his application. MEDER's team of international engineers are always available to provide advice for your application. So if you are unsure of your application always feel free to contact us.

DISCLAIMER:

THIS PUBLICATION COULD CONTAIN TECHNICAL INACCURACIES OR TYPOGRAPHICAL ERRORS. CHANGES ARE PERIODICALLY MADE TO THE INFORMATION HEREIN; THESE CHANGES WILL BE INCORPORATED IN FUTURE REVISIONS OF THIS PUBLICATION. MEDER electronic MAY MAKE IMPROVEMENTS AND / OR CHANGES IN THE PRODUCTS, TECHNICAL SPECIFICATIONS AND PRODUCT PROMOTIONS DESCRIBED IN THIS PUBLICATION AT ANY TIME AND WITHOUT NOTICE.

FOR MOST CURRENT SPECIFICATIONS AND PRODUCTS PLEASE CONTACT YOUR NEAREST SALES OFFICE OR VISIT OUR WEB SITE AT WWW.MEDER.COM.

Sales Office Locations

Head Office Europe
MEDER electronic AG
Friedrich-List-Strasse 6
D-78234 Engen-Welschingen
Tel: +49 (0) 77 33 / 94 87 0
Fax: +49 (0) 77 33 / 94 87 32
E-Mail: info@meder.com
Internet: www.meder.com

Head Office Asia
MEDER electronic Asia Ltd.
Unit 708, 7th Floor, Yen Sheng Centre
64 Hoi Yuen Road, Kwun Tong
HONG KONG-Kowloon
Tel: + 852 / 2955 1682
Fax: + 852 / 2955 1684
E-Mail: salesasia@meder.com

Head Office North America
MEDER electronic INC.
766 Falmouth Road
USA-Mashpee, MA 02649
Tel: + 1 / 508 539 00 02
Fax: + 1 / 508 539 40 88
E-Mail: salesusa@meder.com

Sales Office Italy, Spain, Turkey
MEDER electronic S. r. l.
Via Giuseppe Mazzini, 25
I-20056 Trezzo Sull'Adda (MI)
Tel: + 39 (0) 2 / 90 98 01 82
Fax: + 39 (0) 2 / 90 92 96 98
E-Mail: info@meder.it

Sales Office Switzerland, Austria
MEDER electronic AG
Postfach 180
CH-8197 Rafz
Tel: + 41 (0) 1 / 869 20 60
Fax: + 41 (0) 1 / 869 20 61
E-Mail: salesswiss@meder.com

Sales Office France
MEDER electronic SARL
1/3 Pl. Marcel Rebuffat, Villejust
F-91971 Courtaboeuf Cedex
Tel: + 33 (0) 169 31 44 98
Fax: + 33 (0) 169 31 47 83
E-Mail: salesfrance@meder.com

Sales Office East Europe
MEDER electronic CZ s.r.o.
Becovska 1080
CZ-10400 Praha 10
Tel: + 420 (0) 2 / 67 71 17 35
Fax: + 420 (0) 2 / 67 71 23 60
E-Mail: salesczech@meder.com

First Edition

www.meder.com