Part Design and Process Guidelines

For Pulse Heated Reflow Soldering of Flexible Circuits to Printed Circuit Boards

INTRODUCTION

This report will cover the process definition and components used in the pulse heated reflow soldering process. We will look in depth at the component part design criteria necessary to achieve the optimum quality and consistency of a flex to PCB assembly, concentrating on the most popular joint designs and component parts used in the process today.

DEFINITION OF PROCESS TECHNOLOGY

Pulse heated reflow soldering is a process where two pre-fluxed, solder coated parts are heated to a temperature sufficient to cause the solder to melt, flow, and solidify, forming a permanent electro-mechanical bond between the parts and solder. Pulse heated soldering differs from the traditional soldering process in that the reflow of solder is accomplished using a heating element called a thermode which is heated and cooled down for each connection. Pressure is applied during the entire cycle including heating, reflow, and cooling. A pulse-heated control delivers energy to the thermode, which is mounted on the reflow soldering head. A thermocouple, attached to the thermode, provides feedback to the control for repeatable, consistent heat generation. The soldering head brings the two parts into intimate contact. At a precise pressure the head signals the control to begin the heating cycle of the thermode. The thermode conducts heat to the parts, and the subsequent thermal transfer of heat melts the solder between the parts. The molten areas begin to flow resulting in coalescence between the two solder masses. When the reflow cycle is terminated by the controller, the parts continue to be held together during the cooling cycle such that the solder re-solidifies and a joint is formed. A good solder joint is defined as one where the solder adequately joins both surfaces and wetting (flow of solder) has occurred on both part surfaces.

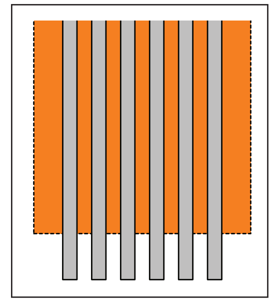
FLEX COMPONENTS

The most common type of flex used in the pulse heated reflow soldering process is manufactured from polyimide (also known under the trade name of Kapton). Two layers of polyimide encapsulate the copper traces (normally 0.5 – 2 oz). The two most common copper conductors are rolled annealed (RA) copper and electro-deposited copper (ED). ED is most cost effective and widely used.

The thickness of copper traces ranges from 0.0007 – 0.004 inches. The polyimide can have operating temperatures ranging from 130 – 200°C and withstand soldering temperatures up to 300°C for a short time. The temperature of the thermode is always higher than that of the parts that are heated by the thermode. A temperature drop of 50-80ºC can occur between the thermode and joint, across the Kapton flex, depending on the thickness. Thickness of the flex ranges from 0.001 – 0.0047 inches.

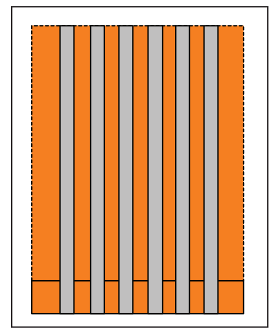
There are three common types of terminationdesigns used on flex circuits for the pulse heated reflow process:

i. The “exposed lead” design has both sides of the polyimide material removed, leaving the traces free of insulation. This allows the thermode to contact the traces directly, and conduct heat to the parts. If the PCB pads and thermode footprint are sized correctly, this design will tolerate some excess solder on the pads, as there will be open areas into which the solder may flow. During the process, solder will also wet to the top of the trace. Caution must be exercised for part handling as the traces can be easily bent or damaged.



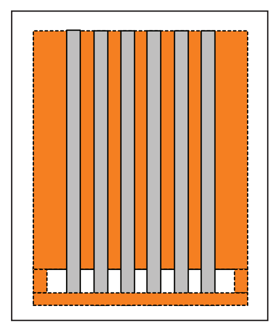
Exposed lead

ii. The “single-sided” flex design has the polyimide removed on one side only. Heat is conducted from the thermode through the solid polyimide surface to the exposed traces underneath. The polyimide conducts heat through the insulation to the exposed traces and pads on the PCB. The polyimide thickness in the joint area is limited to 0.002" enabling conduction. If the polyimide has to be heated much past 260°C, burning of the surface and thermode contamination can result. This design is not tolerant of excess solder on the PCB pads, as there is little room for excess to flow.



Single sided

iii. The “open windowed” flex design has both sides of the polyimide material removed from the joint area, but has support from the remaining polyimide material on the sides and also along the end of the traces. This design gives some strength to the assembly and is resilient to harsher handling. As the traces are exposed, the thermal transfer to the parts is good and excess solder has extra space to flow. Thermode sizing is critical as it must fit into the window and allow space for the molten solder to flow.

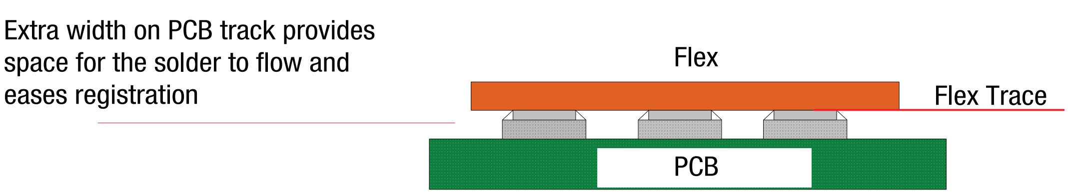


Open windowed

FLEX AND PCB TRACE SIZES

Ideally, the flexible circuit pads should be narrower in width than that of the pads on the printed circuit board. As the solder melts and the parts compress, solder is forced to the side. This design will allow space for the solder to flow on either side of the flex pad and will be more tolerant of solder quantity on the PCB, avoiding solder bridging problems.

A smaller pad width on the flex will help with registration and alignment of the two parts. For fine pitch applications, the width of the PCB trace is designed to be 50% of the pitch. This reduces the risk of short circuits due to misalignment.



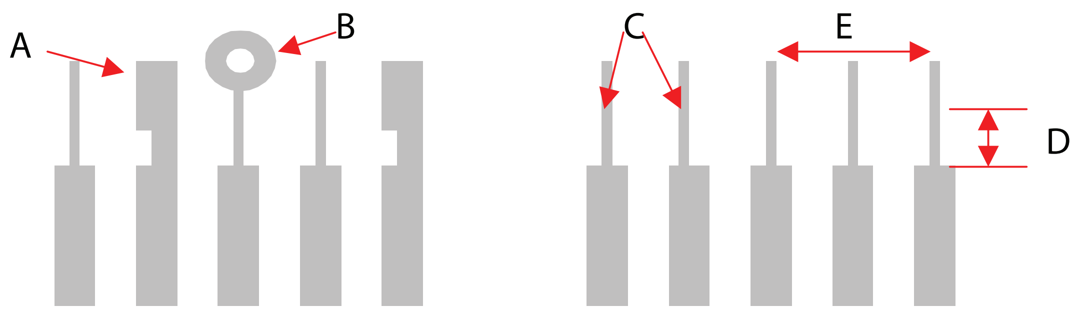
PRINTED CIRCUIT BOARDS (PCB) AND OTHER SUBSTRATES- GENERAL PART DESIGN GUIDELINES

Most PCB materials such as FR2 and FR4 are very resilient to the local application of heat during the process. Materials such as ceramic substrates have to be heated in a more controlled fashion to minimize the chance of cracking. Excessive differences in the heat sinking capability of the two parts can also cause solder cracking during cooling.

Heat sinking differentials along the solder joint length are the most common design problem to overcome. Small differences can have minimal effect, but any large thermal mass change along the joint area will cause inconsistency of temperature and solder joint quality.

HEAT SINKS AND LAND AREAS

Common problems and their possible solutions are detailed below:



Design problem *Design solution*

DESIGN PROBLEMS AND SOLUTIONS NOTES

A. Heat is easily transferred away from the joint area to the large landmass, which is positioned too near to the joint area.

B. Increased trace width and plated through-hole draw heat from the joint area.

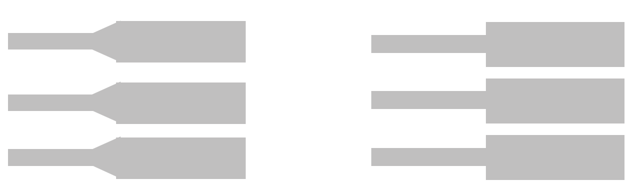
C. The reduced width trace acts as a thermal dam and prevents any heat sinking of the pad.

D. 0.08" is the effective minimum area in which there must be no heat sinks if small trace heat dams are used.

E. Equally sized small traces act as a thermal dam and ensure equal heating across joint area.

Traces leading from pads should be of equal width and be as narrow as possible. This design will act as a thermal dam, and prevent excessive heat drain from the pad area during soldering.

Alternate designs:



For multi-layer boards, restrict the traces under the bonding area to the smallest width (signal) traces and spread equally under the pads on the PCB. Any shielding on the PCB should have an equal effect along the joint area.

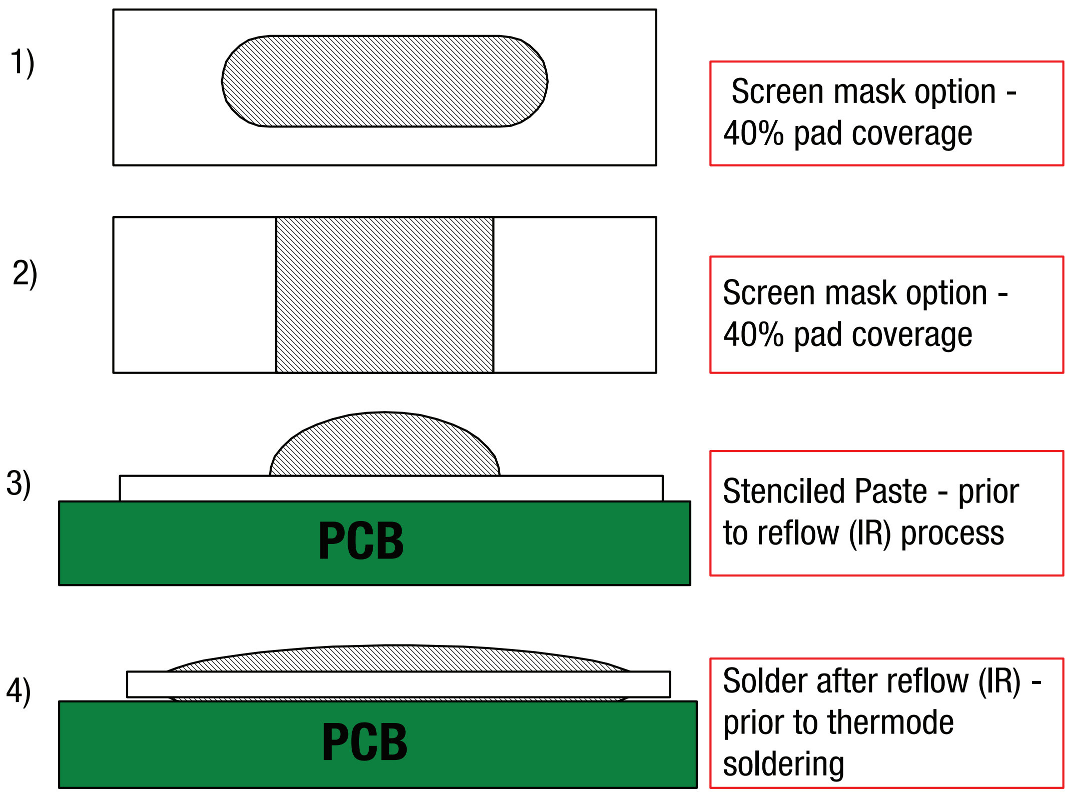
AMOUNT OF SOLDER REQUIRED ON PRINTED CIRCUIT BOARD PAD

The repeatability of solder deposit is critical in order to achieve good process control. In many cases a certain amount of experimentation is required to achieve the ideal solder volume. A good starting point is using a 0.006" screen print stencil, masked to give 40% pad coverage.

The amount of solder required on the pad of the printed circuit board is dependent on a number of factors. The pad size and pitch determine the maximum and minimum solder quantity that can be applied, using the screen stenciling process. Stenciled solder should be fused prior to the reflow process. Small pad and pitch dimensions require less solder thus preventing bridged joints.

The flex design will also influence the volume of solder. Windowed flex and the exposed trace flex, will stand a slightly greater solder volume in comparison to the single sided flex.

1)2)3)4)Stenciled Paste - priorto reflow (IR) processScreen mask option -40% pad coverage Screen mask option -40% pad coverageSolder after reflow (IR) -prior to thermodesolderingPCBPCB



1) A smaller screen aperture can provide 40% solder coverage of pad.

2) Shows an alternate screen design.

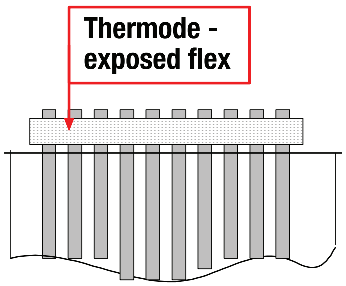
3) Resulting solder deposit prior to the reflow (IR) process.

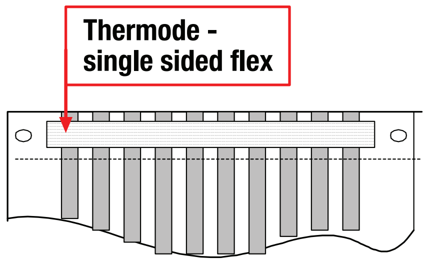
4) Solder is spread evenly across pad after the reflow (IR) process. Note smaller height profile.

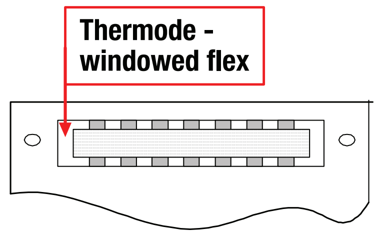
PAD SIZE AND PITCH mm/ inches	**TARGET SOLDER VOLUME** mm³ / inches	**SCREEN THICKNESS** Microns / inches	**OPENING CHEM.NI/AU HOT AIR LEVEL** mm / inches	**Cu PASSIVE / PALLADIUM** mm / inches
Pad size 0.4 x 3.2mm 0.016 x 0.126" Pitch – 0.8 mm 0.031"	0.02 – 0.03mm 0.00079 – 0.0012"	100 micron / 0.004"	0.36 x 1.2 / 0.014 x 0.047	0.36 x 1.6 / 0.014 x 0.063
		150 micron / 0.006"	0.30 x 1 / 0.012 x 0.039	0.30 x 1.2 / 0.012 x 0.047
		175 micron / 0.007"	0.26 x 1/ 0.010 x 0.039	0.26 x 1.2 / 0.010 x 0.047
Pad size 0.8 x 5.5 mm 0.031 x 0.216" Pitch – 1.5 mm 0.059"	0.07 – 0.10 mm 0.0028 – 0.0039"	100 micron / 0.004"	0.74 x 1.9 / 0.029 x 0.075	0.74 x 2.7 / 0.029 x 0.106
		150 micron / 0.006"	0.6 x 1.5 / 0.024 x 0.059	0.6 x 2.2 / 0.024 x 0.086
		175 micron / 0.007"	0.5 x 1.6 / 0.020 x 0.063	0.5 x 2.3 / 0.020 x 0.091
		200 micron / 0.008"	0.4 x 1.7 / 0.016 x 0.067	0.4 x 2.5 / 0.016 x 0.098
Pad size 0.8 x 4.5 mm 0.031 x 0.177" Pitch 1.6 mm 0.063”	0.057 – 0.08 mm 0.002 – 0.0032"	100 micron / 0.004"	0.7 x 1.6 / 0.028 x 0.063	0.7 x 2.3 / 0.028 x 0.091
		150 micron / 0.006"	0.6 x 1.3 / 0.024 x 0.052	0.6 x 1.8 / 0.024 x 0.071
		175 micron / 0.007"	0.5 x 1.3 / 0.020 x 0.052	0.5 x 1.8 / 0.020 x 0.071
		200 micron / 0.008"	0.4 x 1.4 / 0.016 x 0.055	0.4 x 2.0 / 0.016 x 0.055
Pad size 1 x 4.5 mm 0.039 x 0.177" Pitch – 2.0 mm 0.079"	0.07 – 0.1 mm 0.0028 – 0.0039"	100 micron / 0.004"	0.74 x 1.9 / 0.029 x 0.075	0.74 x 2.7 / 0.029 x 0.106
		150 micron / 0.006"	0.6 x 1.5 / 0.024 x 0.059	0.6 x 2.2 / 0.024 x 0.086
		175 micron / 0.007"	0.5 x 1.6 / 0.020 x 0.063	0.5 x 2.3 / 0.020 x 0.091
		200 micron / 0.008"	0.4 x 1.7 / 0.016 x 0.067	0.4 x 2.5 / 0.016 x 0.098
Pad size 1.5 x 5.5 mm 0.059" x 0.216" Pitch – 3.0 0.118"	0.13 – 0.19 mm 0.0051 – 0.0075"	100 micron / 0.004"	1.2 x 2.2 / 0.047 x 0.087	1.2 x 3.1 / 0.047 x 0.122
		150 micron / 0.006"	1.1 x 1.6 / 0.043 x 0.063	1.1 x 2.3 / 0.043 x 0.091
		175 micron / 0.007"	1.0 x 1.5 / 0.039 x 0.059	1.0 x 2.1 / 0.039 x 0.083
		200 micron / 0.008"	0.9 x 1.4 / 0.035 x 0.055	0.9 x 2.0 / 0.035 x 0.055

THERMODE SIZING AND POSITIONING TO PARTS

Thermodes should be sized according to the pad and flex sizes as shown in the figure below. The thermode length must completely cover the traces and overlap by a minimum of one pad pitch on each side. The thermode width should provide enough thermal transfer of heat to achieve the solder joint in the minimum time thereby eliminating the chance of thermal damage to the parts. The width of the thermode should also accommodate enough room for the molten solder to be displaced, eliminating any chance of solder bridging.







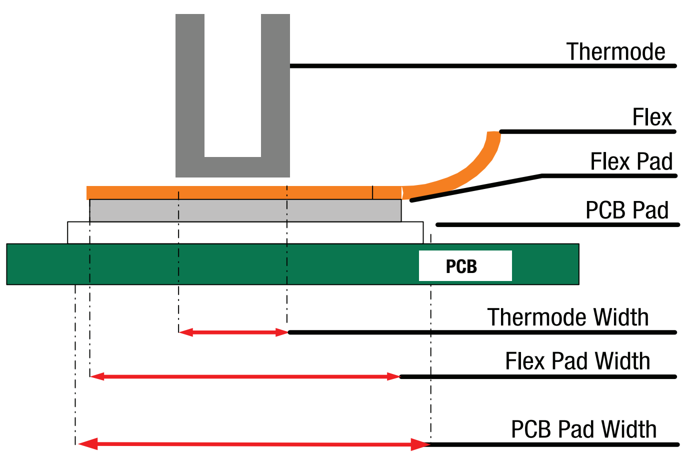
Thermode Width: For the best thermal performance and lifetime of the thermode, the minimum size should be 0.059". The standard size is 0.079", due to better performance and longevity. Where solder amounts are not well controlled or room is limited, it is possible to use a 0.047" wide thermode however thermode life and performance will be reduced.

Flex Pad Width: Note that the flex pad finishes short of the PCB pad. This is to allow easy inspection of the joint.

PCB Pad Width: Extra width allows for excess solder and ease of inspection. PCB pad is approximately three times the width of thermode.

SUGGESTED THERMODE WIDTH / PAD LENGTH AND PITCH

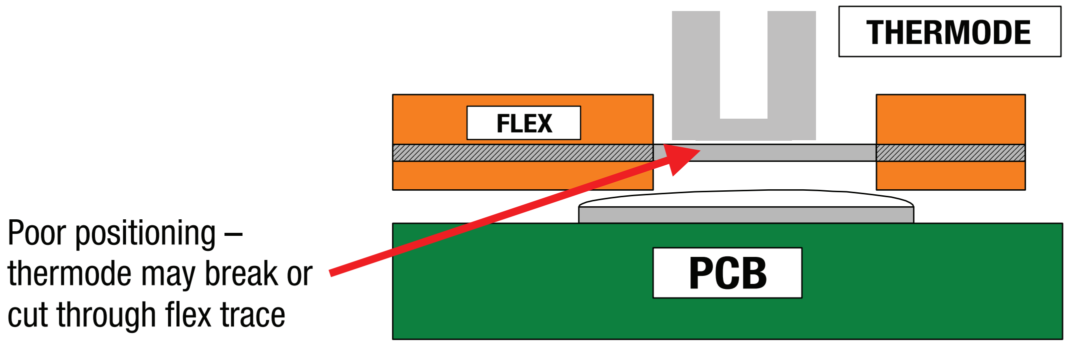
PCB		PCB
Pad pitch mm / inches	Thermode width mm / inches	Pad length mm / inches
0.8 / 0.031	1.5 / 0.059	4.5 / 0.177
1.2 / 0.047	1.5 / 0.059	4.5 / 0.177
1.4 / 0.055	2.0 / 0.079	5.0 / 0.197
1.5 / 0.059	2.5 / 0.098	5.5 / 0.217
1.6 / 0.063	2.5 / 0.098	5.5 / 0.217
1.8 / 0.071	2.5 / 0.098	6.0 / 0.236
2.0 / 0.079	3.0 / 0.118	6.0 / 0.236
3.0 / 0.119	3.0 / 0.118	6.0 / 0.236



The dimensions above are guidelines only. Some experimentation may be required, due to different solder volumes.

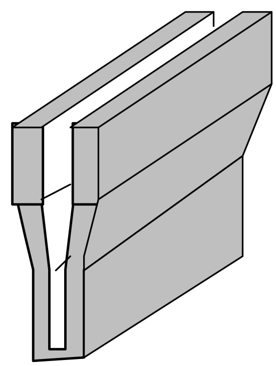
THERMODE POSITIONING

When positioning a thermode on an exposed or windowed flex, it is important that the thermode is not positioned too close to the edge of the main body of the flex. Some flex circuits have thinner and thicker coatings on either side of the traces running through them. If this is the case position the thinner side next to the PCB. This will reduce the chance of the thermode damaging the trace as it pushes it down to the surface of the trace on the PCB.



THERMODE MANUFACTURING AND TEMPERATURE CHARACTERISTICS

Modern wire erosion techniques such as EDM and advanced materials have allowed the manufacture of precisely designed thermodes to suit most applications. Three-dimensional thermodes pass the current around the face and thus have zero voltage potential across the traces. These technological advances in machining processes produce designs with constant temperature across the length, and special alloys achieve flatness and co-planarity under heating. Solder will not wet to the materials used and they are resilient to oxidization.



TOOLING AND PART POSITIONING

Heat resilient high temperature plastic such as peek (trade name Ketron) or tuffnel should be used under the reflow area to prevent heat sinking from the bond area. Tooling nests should be totally flat as the quality of the process depends upon achieving an equal distribution of heat when the pressure from the thermode is applied. The best finishing technique is to grind the surface in preference to milling. If possible, parts should be located on tooling pins adjacent to the reflow area. It is common practice for the tooling holes in the flex to be reinforced with copper trace for strength and accuracy. If no tooling holes are possible, the parts can be positioned and tooled from a square edge. As the flex is not rigid vacuum holes in the part nest may be required to hold it flat. For fine pitch flex, an alignment x-y stage and camera system may be useful. It is important to take into account any tolerances or batch-to-batch variations in size when designing the part fixturing.

PREPARATION

It is more common for both parts to be previously solder plated. If this is not the case, it is still possible to achieve wetting between a single sided flex design and a base pad plated with gold or tin. The base plating of the two parts is often enough solder to achieve a reliable joint with a single sided flex.

Most flex designs, however, will require additional solder normally applied by the screen printing process and previously reflowed (quantities are described earlier). For finer pitch applications the solder is normally hot air leveled prior to the reflow process. Hot air leveling allows even distribution of the solder along the pad and good thermal transfer resulting from a flatter surface. This process also makes alignment under the pressure of the heated bar easier to maintain. Parts must be free of dirt and dust and generally clean and oxide free. Flux is normally used to ensure any oxide barrier is removed to allow proper wetting to occur.

FLUX

Flux has two important features. It conducts the heat to the solder and it promotes the wetting of the surfaces by cleaning and removal of surface oxides. For easy to solder parts, the pulse heated soldering process requires only a minimum of non-activated flux. No clean fluxes are commonly used. The use of a low solids content flux is recommended. The lower solids content the less pollution of the thermode. Any solvents present should be allowed to dry prior to commencing the soldering process.

SAFETY

In comparison to conventional soldering the pulse heat thermode soldering process is very safe, as the heating element is only hot when it is pressed to the parts. In addition, only a very small amount of flux is needed and therefore there are much less fumes generated. Operators should still be prevented from touching the thermode during the cycle and should also be protected from entrapment hazards.

SOLDERING METHOD- PROCESS STEPS

1. The base substrate is located in a fixture and flux applied to the pads.

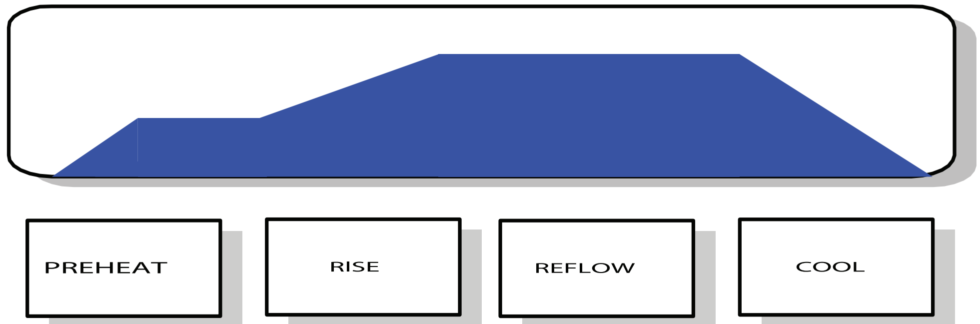
2. The flex is positioned in the parts fixture ensuring alignment of both sets of pads.

3. A process start signal is given to the soldering control (footswitch input).

4. The soldering control then actuates the bonding head and thermode module to the parts.

5. At a pre set pressure the heating process is initiated.

THE HEATING PROCESS



PREHEAT

It takes approximately two seconds to heat a modern designed thermode of up to 2” in length to soldering temperature. During this time, the flux activates and starts to promote wetting by removal of the oxide layer. Preheat is only used where there are excessive heat sinks affecting the thermode or where the application has delicate substrates, like ceramic that need to be heated in a more controlled fashion to avoid cracking.

RISE

Rise time to soldering temperature is also programmable and allows precise heating rate control. This again is particularly useful where delicate substrates can be easily damaged by too fast a heating rate. Normal rise time for most thermodes is 1.5 – 2 seconds.

REFLOW

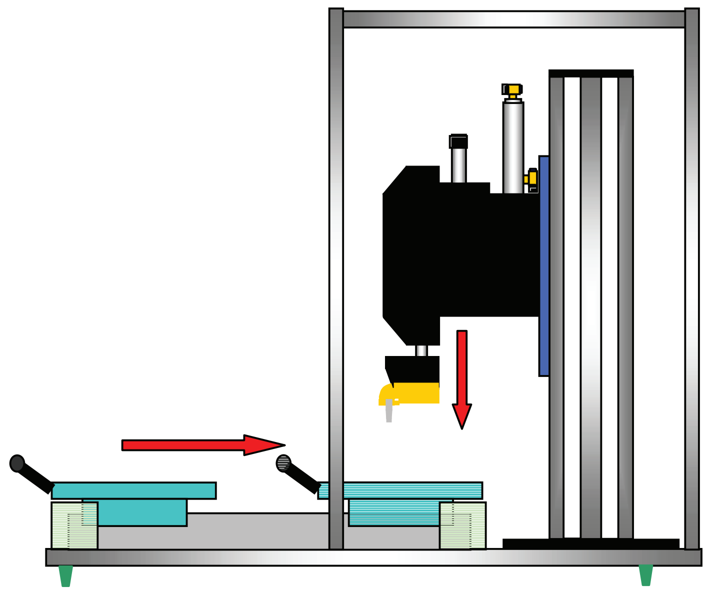
The actual time and temperature can be programmed for this stage of the process. Time is programmable in 0.1-second increments and temperature in 1-degree increments. Typically, the temperature set point for an open solder joint with direct thermode contact to parts will be between 280 - 330°C. Although normal solder will reflow at 180°C the thermode must be set higher due to the thermal transfer losses. A typical single sided flex will require between 330 - 400°C due to the thermal losses in the Kapton material. It is preferred to use the minimum time and temperature to achieve the desired joint, so as to minimize the parts exposure to heat and chance of damage.

COOL

Cool is a programmable temperature at which the control unit will actuate the head to the up position. This temperature is set to just below the solder solidification temperature. Therefore as soon as the solder becomes solid the process is ended and a joint is formed. The cooling process can be shortened by the use of forced air cooling. The power supply can be programmed to switch a relay that controls the flow of air at the end of the reflow period and cool the joint and thermode rapidly. Because most connections have a relatively high heat sink, the temperature in the solder is lower than the measured thermode temperature, even when using cooling air. Therefore the release temperature can be set to 180º C in most cases without the chance of encountering a dry joint.

FORCE CONTROL AND SIMPLE SYSTEM EXAMPLES

Most reflow joints of this nature require fewer than 20lbs. (9 Kg.) pressure. A range of force control modules is available to suit all applications up to 150lbs. (68Kg.). Force must be precisely controlled. Force can be calibrated and set to the correct level to achieve the right thermal transfer of heat to the solder joint. It is desirable to have co-planarity adjustment in the thermode mounting or the head itself, for ease of set up. Modern designs have either air or motorized actuation and built in valves for thermode cooling. Many heads are modular in construction and therefore versatile for integration into tooled or semi automatic fixtures. Linear slides allow loading and unloading of the parts away from the bonding area. Rotary table systems are preferred for high volume production. The operator can be loading one set of parts concurrent with another set of parts being soldered. This doubles the output from one operator, thus reducing the labor cost.



QUALITY CONTROL AND INSPECTION

Pressure is maintained as the joint is cooled. Therefore there is little chance of a dry joint occurring. The imprint of the thermode should be seen on the solder joint and be even in width and length. There should be visual evidence that reflow has occurred and when the parts are peeled apart the resulting joint will have a granular appearance over the soldered area. There should be no evidence of burning or delamination of the pads to board or flex. Where a single sided flex is used, there maybe marking or discoloration on the top of the polyimide but no burning or separation should be seen. Any flux residues can be cleaned after the reflow process. No clean, low residue fluxes do not require post cleaning.

Temperature and time process data can be collected from the control and displayed in graphical format to illustrate process stability.

PROCESS MAINTENANCE

Maintenance of the cleanliness of part fixtures is required to ensure that the parts continue to sit flush to the base. Periodic maintenance of the thermode is also needed to prevent the build up of baked on flux. Using a flux solvent, or cleaning the thermode with a very fine emery or grit paper mounted to a flat rigid surface will maintain good thermal transfer to the parts. Care must be taken so as not to round the edges of the thermode or spoil the flatness. There is a distinct difference between the pollution of the thermode for soldering processes where the thermode is positioned directly on the leads in contact with the solder and those where the thermode is in contact with a Kapton surface. In the first case the pollution and thermode wear is much higher, and cleaning must be done on a more regular basis. The thermocouple joint connection must be kept clean and in good order to ensure repeatable temperature control. Thermocouple types K and E are not eroded by flux but type J can be attacked and eroded.

CONCLUSION

Pulse heated thermode reflow soldering of flex to PCB is a stable and well controlled process if certain basic design guide rules are followed. These rules differ from the rules that apply to conventional soldering processes. The process window can be made substantially wider by a joint design that promotes easy and equal heat generation. Even more by a design that accommodates the flow of solder and can compensate for variations in the prior processing steps. Good joint design and repeatable fine control over solder quantities are the keys to production success. The growing need for product miniaturization and reduced weight are major drivers increasing the use of flexible circuit technology with the electronic industry. Today's control over the thermode soldering process offers a production oriented, reliable solution to the interconnection demands of this growing market.

AMADA MIYACHI AMERICA, INC.
1820 S. Myrtle Ave. • Monrovia, CA 91016 US
T: (626) 303-5676 • F: (626) 358-8048
info@amadamiyachi.com • www.amadamiyachi.com
ISO 9001 Certified Company • 24/7 Repair Service: 1-866-751-7378

MIYACHI UNITEK | MIYACHI PECO | MIYACHI EAPRO

AMERICAS

Amada Miyachi America Midwest
Wixom, MI 48393
T: (248) 313-3078
midwestsales@amadamiyachi.com

Amada Miyachi America Mexico
El Paso, TX 79925
T: (915) 881-8765
mxsales@amadamiyachi.com

Amada Miyachi do Brasil Ltda.
Sao Paulo, Brasil
T: +55-11-4193-3607
antonio.ruiz@amadamiyachi.com

EUROPE

Amada Miyachi Europe GmbH
Puchheim, Germany
T: +49 (0) 89 83 94 030
infode@amadamiyachi.eu

ASIA

Amada Miyachi Co., Ltd.
Noda, 278-0016 Japan
T: +81-4-7125-6177
sales@miyachi.com

Amada Miyachi Shanghai, Co., Ltd.
Shanghai, China
T: +86-21-6448-6000
zqzhang@msc.miyachi.com

Amada Miyachi Korea Co., Ltd.
Gyeonggi-do, Korea
T: +82-31-8015-6810
dykim@mkc.miyachi.com

Amada Miyachi Taiwan Co., Ltd.
Taipei, Taiwan R.O.C.
T: 886-2-2397-4778
keigaku@miyachi.com

Amada Miyachi (Thailand) Co., Ltd.
Samutprakarn, Thailand
T: +66-2751-9337-8
info@mtl.miyachi.com

Amada Miyachi Vietnam Co., Ltd.
Ho Chi Minh City, Vietnam
T: +84-8-3771-7972

Amada Miyachi India Pvt., Ltd.
Bangalore, Karnataka
T: 080-4092-1749 & 3549
info@miyachiindia.com

follow us on: