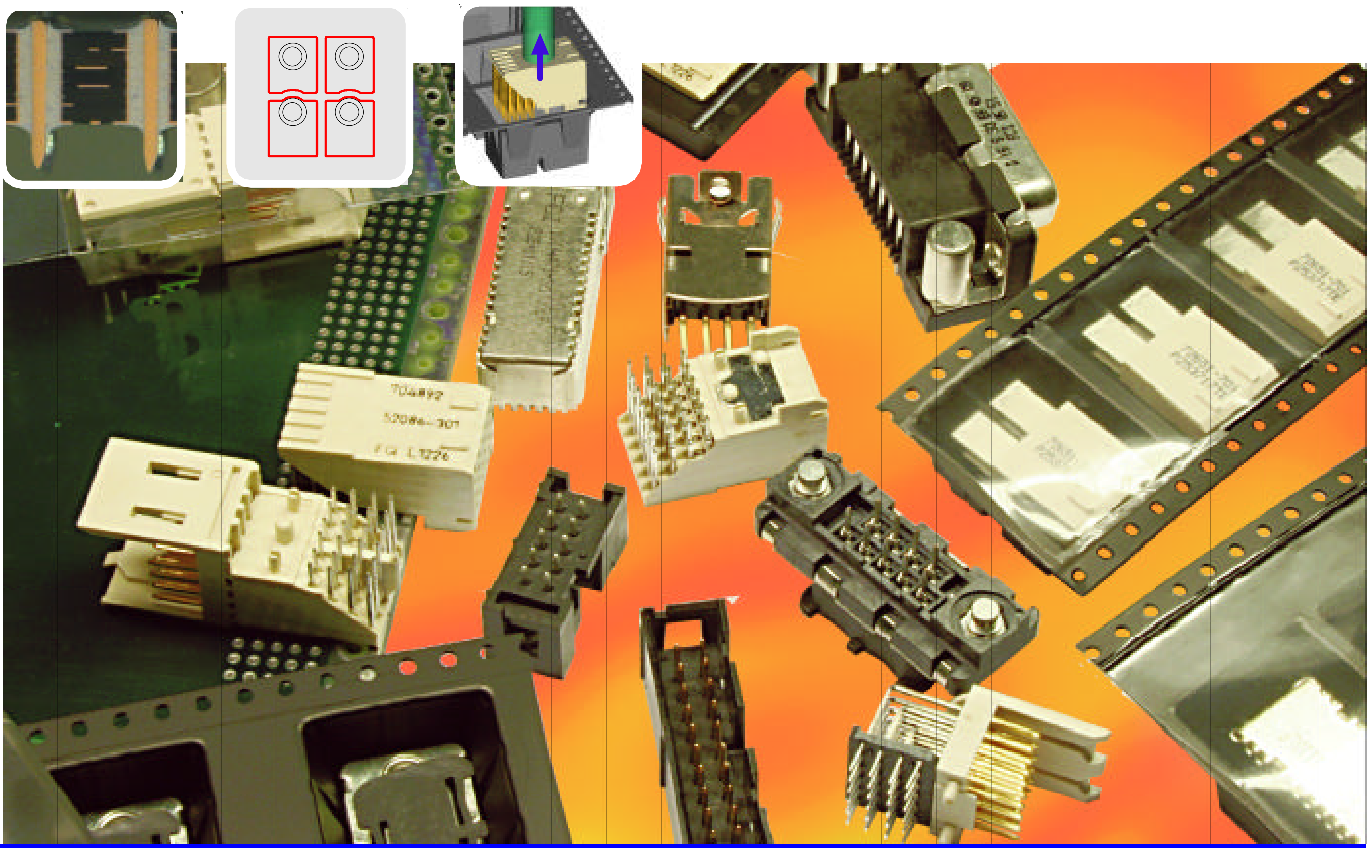

COMMUNICATION, DATA AND CONSUMER DIVISION

Pin-in-Paste Application Guide

CDC-PIPGUIDE-02/07-E

FCI: SETTING
THE STANDARD
FOR CONNECTORS

With operations in 30 countries, FCI ia a leading manufacturer of connectors. Our 13.500 employees are committed to providing customers with high-quality, innovative products for a wide range of consumer and industrial applications

Introduction

This application guide is a brief review in implementing through-hole connectors to SMT-processes. The aim is to give information to all people involved in the process of developing and manufacturing electronic hardware.

Content

Liability

AW: KM

www.fciconnect.com

CDC-PIPGUIDE-02/07-E

Introduction

SMT PROCESS DESCRIPTION

A typical SMT-process is in principle 3 steps, application of solder paste on the bare boards, assembly of components and reflow soldering. To achieve high efficiency the process in reality is more complicated.

The boards are automatically handled into the screen printing equipment for solder paste application. A screen printing stencil is provided with openings (apertures) to control the paste printing. The stencil thickness may vary, but typical values are 0,125 and 0,15 mm. The paste is pressed into the apertures with a squeegee, or with a special paste head providing vertical pressure on the paste while passing over the stencil (e.g. ProCon). The screen printing can be made in one or more strokes to provide wanted aperture filling

After the paste application vision-based automatic inspection equipment can inspect crucial areas for defects in the paste application. The boards are carried on a conveyor through the process steps.

The first component assembly station normally handles the smallest components with a very high mounting speed. They are placed in the paste, and a vision inspection may be placed after the assembly, or after some more assembly steps. Further assembly stations may be used, depending on type of boards and components involved.

The final assembly normally deals with larger and odd shape components. After final assembly and optional vision inspection, the boards pass through the solder reflow oven. The final inspection is either manual or automatic.

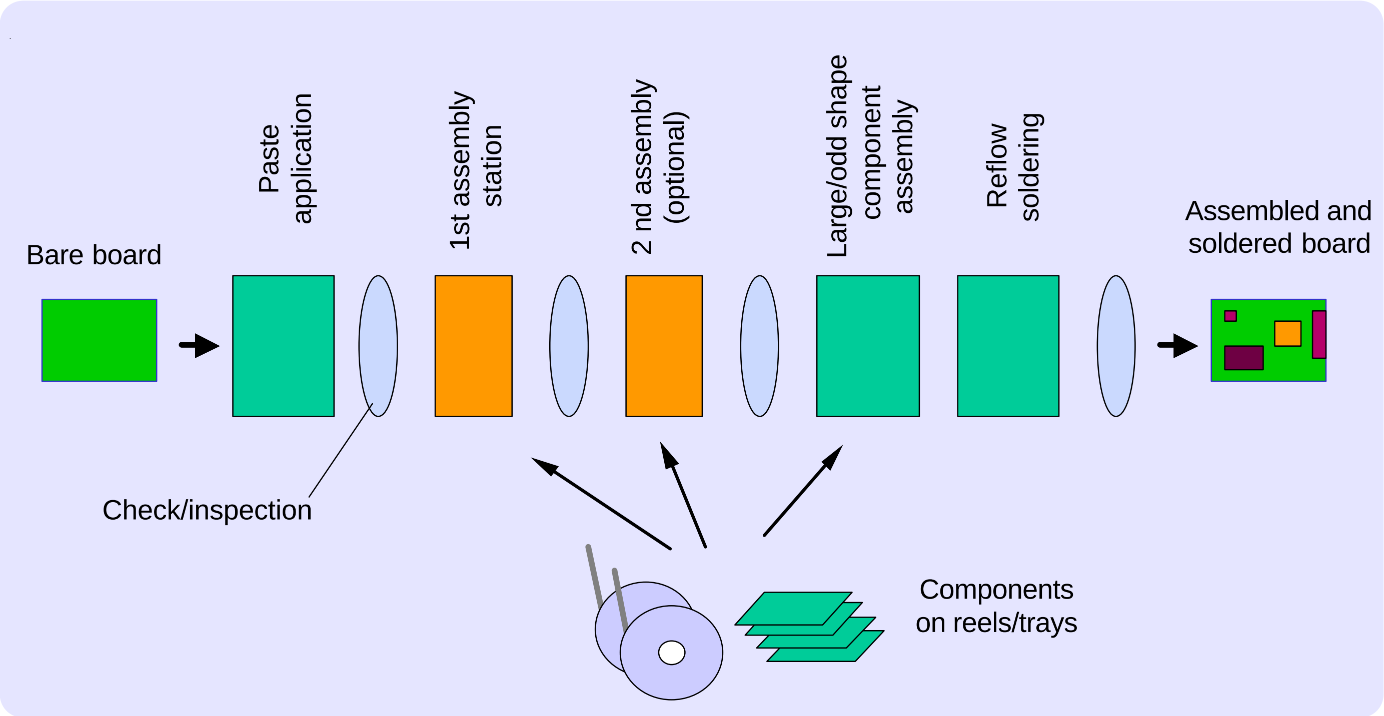

Figure 1
Typical SMT process overview.

The Pin-in-paste (PIP) technology

INTRODUCTION

The principle for PIP is that solder-through-hole component posts are placed in holes with SMT-solder paste and then reflow soldered in the same operation as the SMT components.

Process parameters are hole size, pin size, board thickness, stencil aperture, stencil thickness, paste filling into hole and paste properties.
Compared with wave soldering the process normally shows better result in wetting and less bridging, but for components with thin pins in large holes with thick boards it may sometimes be hard to put enough paste on to get 100 % filling. Industry standard quality requirements have taken these into account, and permit less filling for pin-in paste soldering. (see further in section *Inspection, quality references*).

Especially for connectors in backplane systems the position requirements and the external forces need special attention. Some Pin-in paste connectors are therefore provided with location and in some cases also hold-down features (HDF) for proper adaptation to the SMT process.

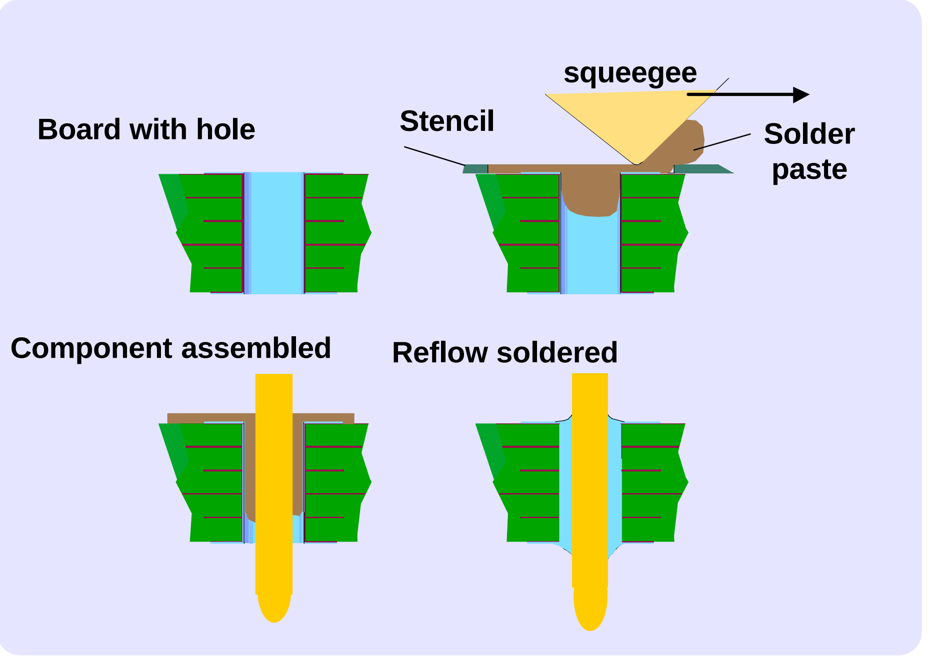

Figure 2
Pin-in-paste process sequence

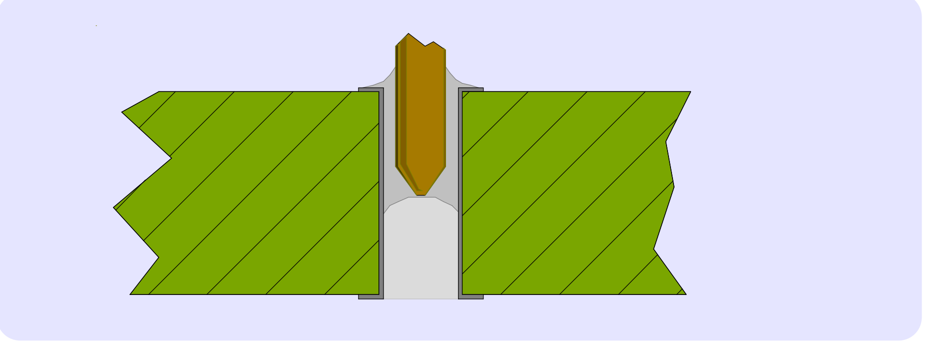

Figure 3
Partial Pin Penetration Soldering (PPPS)

MODIFIED PIP (PPPS)

A variant of the PiP-technology is the modified PiP or PPPS (Partial Pin Penetration Soldering), which in some cases is a more suitable way to make soldering. (Figure 3)
Such case can be for small pitches where the amount of paste by printing is limited, or when signal integrity is requesting small holes, and the automatic assembly (Pick and place) will be easier by use of a shorter solder post.
PPPS is also board thickness independent, so a paste printing aperture is valid for all thicknesses. (Figure 4)
PPPS will also be used if no protrution is allowed at the other side of the board.

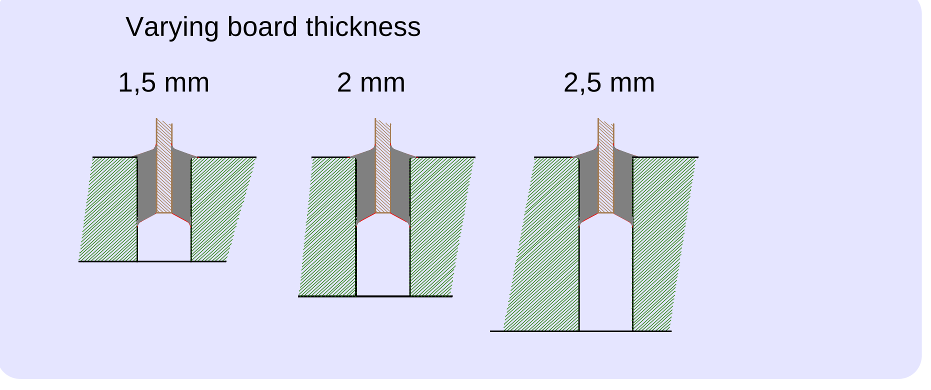

Figure 4
PPPS independency of board thickness

PASTE APPLICATION

The amount of paste necessary to give an appropriate solder joint can be calculated. Normally there is a need to print paste also on the board surface (overprinting), not only in the hole itself, so the stencil apertures are larger than the land of the hole.
Paste applied on the board surface will later be sucked into the hole in the reflow process.

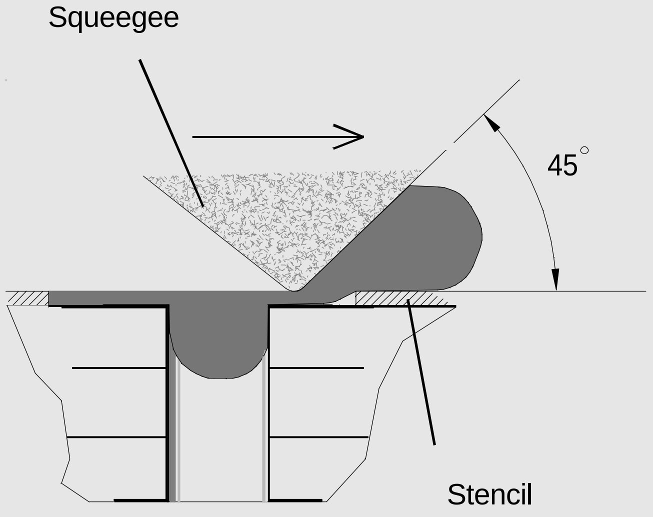

Figure 5
Paste application squeegee

BOARD LAYOUT ISSUES

In order to maximize yield and process setting some rules may be useful:

Make hole lands as small as allowed. This will reduce the needed amount of solder paste

Ensure solder mask is covering all copper, so the melting paste can flow into the hole.

Avoid via holes where paste is planned to be printed

STENCIL APERTURES

In principle a hole grid for solder posts can be printed with paste all over, but then the amount of solder sucked into the holes will differ from joint to joint depending on where the melting of the paste starts.
It is therefore recommended to let each hole have its own defined paste printing shape. In the case with connector solder tail grids, the paste amount is often maximised by the grid size, e.g. 2 x 2 mm. Separating space should normally be equal or more than 0,3 mm. (see figure 6). The apertures in a grid may very well be optimised in size using unsymmetrical placing of paste apertures (see figure 25).

The amount of paste in the hole will depend on which vertical pressure the paste applicator can provide. A normal squeegee will improve the vertical pressure if the angle is reduced from 60° to 45° (see figure 5).
Improved filling can also be achieved with applicators providing a static pressure against the board/stencil.

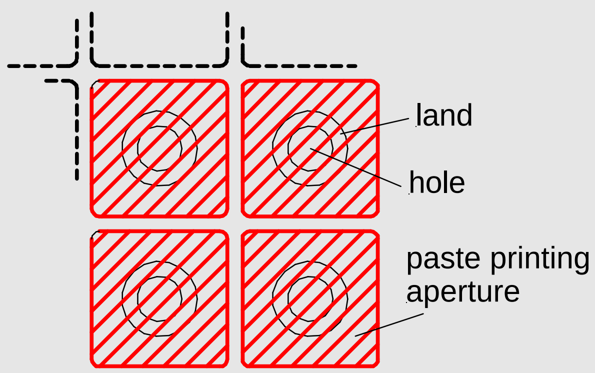

Figure 6
Stencil aperture principle

FILLING DEGREE

The filling degree (Fd) of a standard PIP-joint can be calculated according to following:

Fd = P/V   P= metal volume in applied paste
V= volume to fill (hole - post)

P = {(Y * X* T) + [(D/2)^2 * p] * Z}* m

m= metal volume in paste.
(normally about 50% = 0,5)
Z =paste filling in hole, dimension from board surface and into the hole

Remark that for PPPS the volume V is only the part of the hole which is penetrated by the solder post (penetration depth).

Calculations will show that the length "Z" is important for the filling degree, as well as not having the hole oversized.

Additional paste can also be applied with dispensing units. As SMT-lines are not often equipped with dispensing units this will be used in special cases. In addition to the possibility to add more paste, the paste can also have different properties (viscosity, metal content) to give better filling degree.

Screen-printing stencils may also have partially thicker areas to increase the paste volume at the connector positions (stepped stencils). Some limitation must be taken into account as squeegee moving direction, connector positions, more expensive stencil etc.

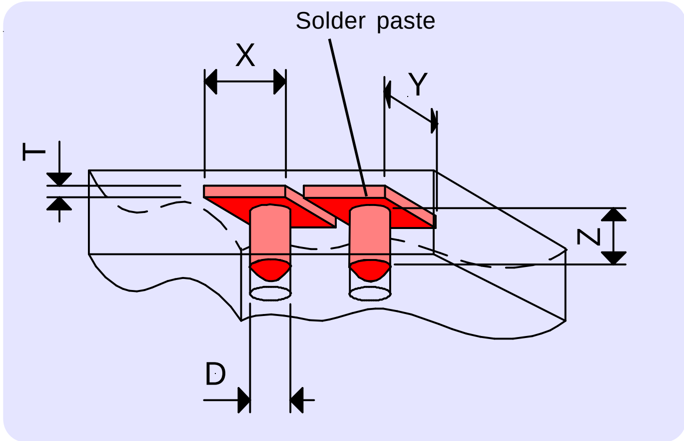

Figure 7
Paste volume calculation definitions

COMPONENT FEEDING AND PACKAGE

For SMT components the package is a part of the product. It is important that the component is fed into and presented to the assembly process in a proper way. Normally the preferred package is tape according to EIA 481 specifications. This allows for use in most SMT-equipment. Metral connector tape is shown in figure 8.
Most SMT machines can also handle components from tray, and in some cases also from tubes.

Figure 8
Tape-and-reel packaged connectors

COMPONENT PICKING AND ASSEMBLY

The assembly equipment normally has a vacuum nozzle as picking and handling tool. Special designed mechanical grippers could also be used.
To allow for a proper use of the vacuum nozzle the component must be provided with a flat surface of enough area, and this surface must be parallel to board mounting plane.
It is recommended to use vacuum nozzle without rubber edge in order to allow the connector to self-correct position during the final part of the mounting cycle.
The handling cycle normally includes a vision inspection, where the position of the component is controlled, and parameters given to the control system to adjust position for the board assembly.

The vision system may also be able to check features like solder terminals etc. A connector, often having features for locating to board holes, shall normally have visible reference features to control the position.
For connectors, in general the placement on the board shall be made with only z-movement (vertically). This is to avoid interference with already assembled adjacent connectors.
If the connector assembly needs force due to locating/fixing features, a board support has to be installed.

A large component/odd shape component assembly machine principle is shown in figure 10.

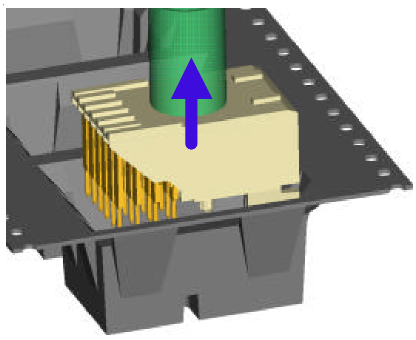

Figure 9
Vacuum nozzle picking connector from tape package

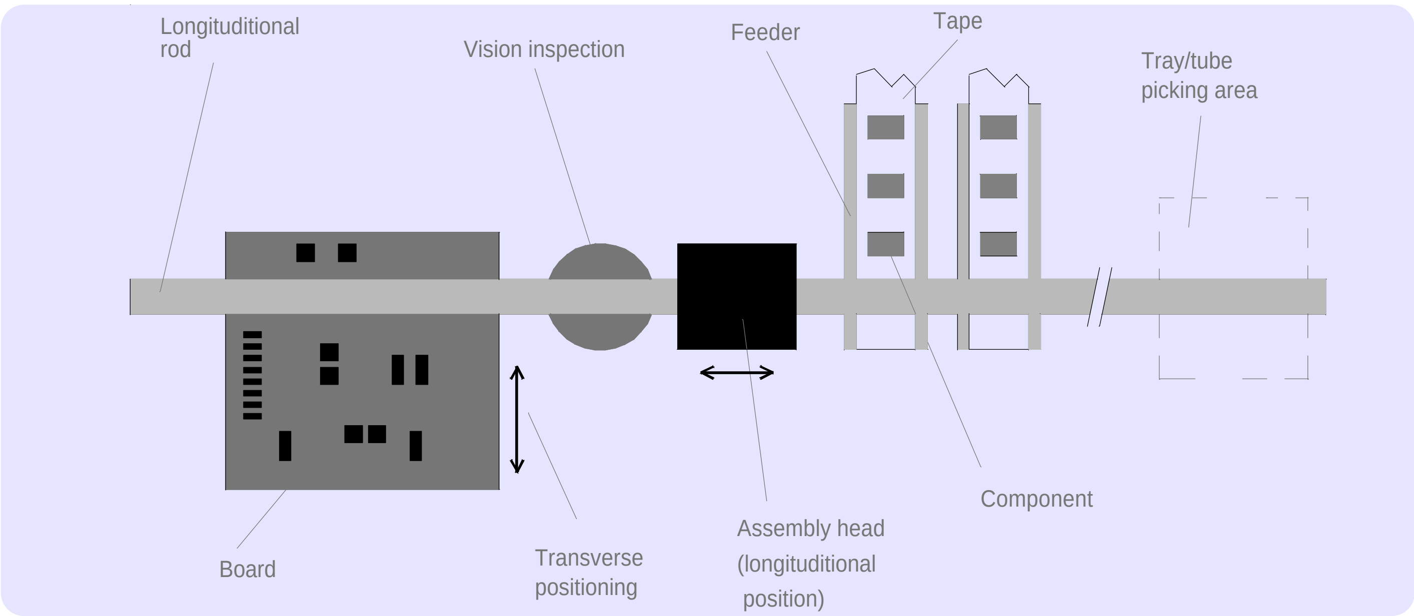

Figure 10
Example of last step component assembly machine (principle)

REFLOW

The assembled board will be fed through the reflow oven. The oven consists of several zones with different temperature. This is to ensure even temperature distribution and minimise risk of component failure due to temperature stresses. A typical temperature profile is shown in figure 11.

During the reflow process the flux content in the paste is evaporating while cleaning the joint surfaces, the small balls of tin (tin/lead) start to melt and the liquid metal pulled towards the clean metal surfaces (e.g. pads and posts) by surface tension. The solder mask on the board prevents the tin to creep out from the pads and remain creating solder balls.

The conveyor speed in the oven is normally 500 to 1000 mm/min.

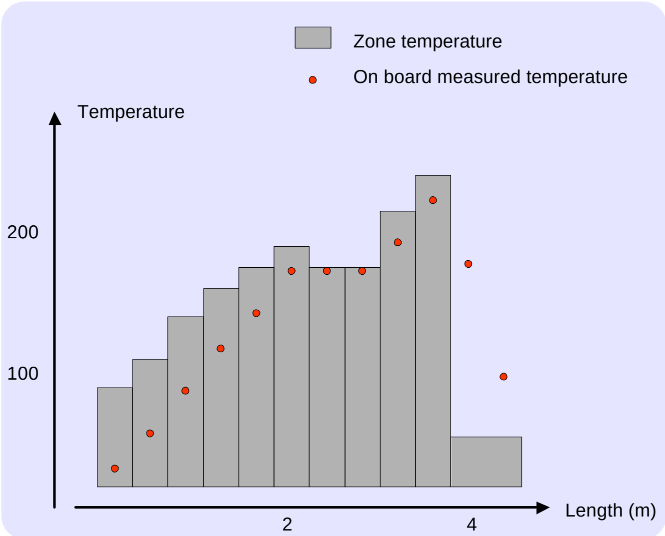

Figure 11
Reflow oven temperature zones

INSPECTION, QUALITY REFERENCES

The most used quality standards published by IPC (The Institute for Interconnecting and Packaging Electronic Circuits), outline the acceptance of solder joints with minimum 75 % filling degree.

The quality is mainly secured by process control. The main tools for controlling the process are x-ray and micro sectioning. For judging and approval of soldering in tuning the process parameters the filling degree at a micro sectioned soldering as expressed in figure 12 and 13 can be helpful.

Quality/inspection criteria must be considered differently than for wave soldering. The most important is that the amount of solder paste is controlled by the process by specifying hole, post, paste, stencil aperture and paste applicator service parameters. The remaining inspection can be reduced to check proper assembly and wetting.

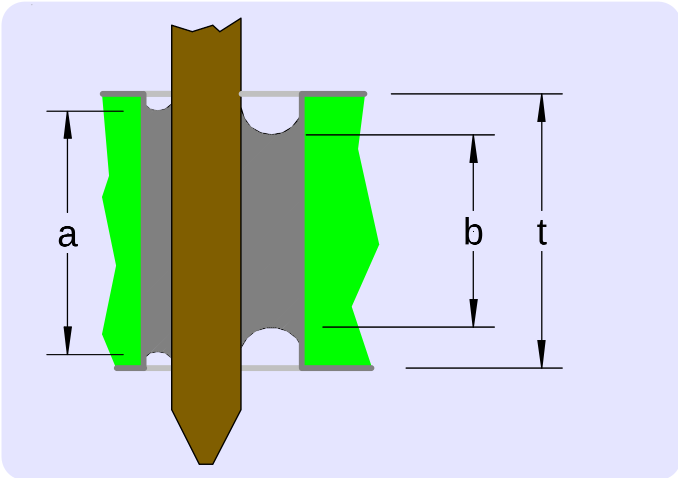

Figure 12
Filling degree calculation units

Calculation 1 (for figure 12)
(a+b)/2t ³ 0,75 (for t less or equal to 2.4 mm)

If voids are indicated, the whole filling shall be measured (b1) and the effective unsoldered high (b2) of the void deducted from the dimension as shown in figure xxx.
Voids not touching any or only one of the sides (pin or hole) shall not be taken in account.

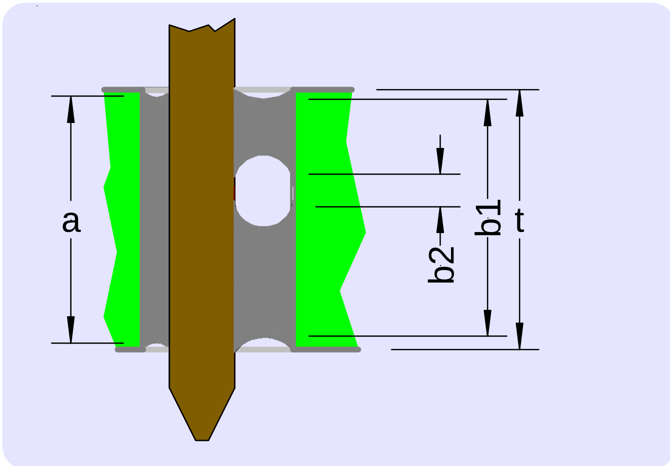

Figure 13
Filling degree with voids

Calculation 2 (for figure 13)
[a+(b1-b2)]/2t ³ 0,75 (for t less or equal to 2.4 mm)

Calculation for PPPS-joint can be made, but intention is to have more close to 100 % filling. Voids may be present, but with less paste/solder the risk for voids is reduced.
Try to avoid to ”overfill” PPPS-joints, as the risk for voids will increase, as well as for blow-outs.
See figure 14.

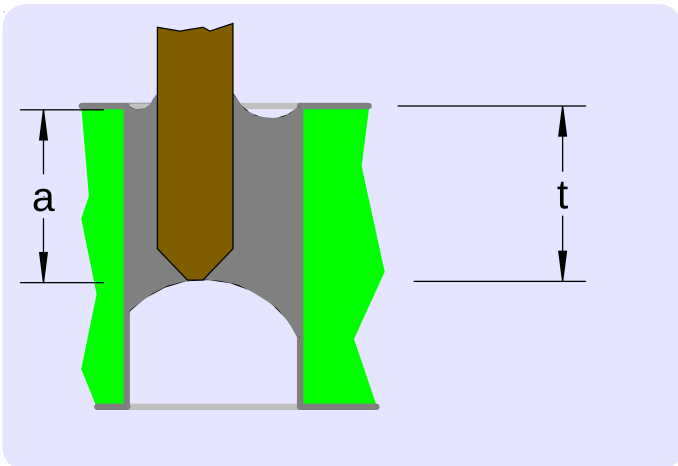

Figure 14
Typical approved filling with PPPS.

MICRO-SECTIONING OF SOLDER JOINTS

The following figures from actual Pin-in paste soldering give some example of the results.

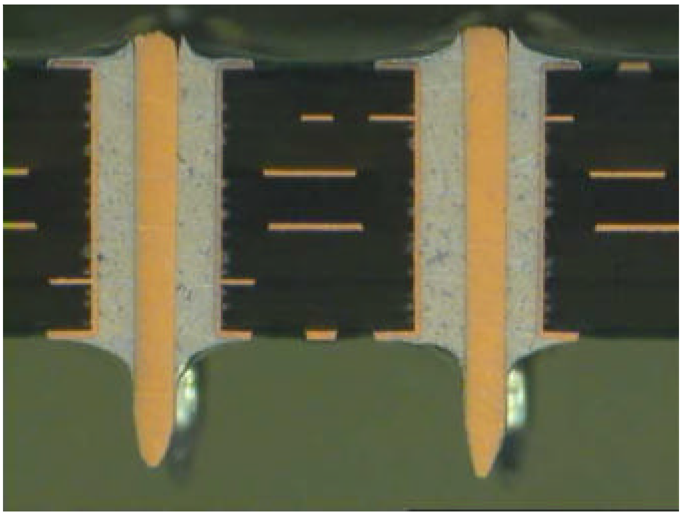

Figure 15
Solder joints in 1,7 mm thick board. Paste screen aperture 1,7x1,8 mm. 100% filling and approved wetting

Filling degree according to calculation below gives the average filling as the soldering very seldom is symmetric (as shown in IPC-requirements)

Board thickness t =2 mm (in this case equal to ”b”).
a = 1,2 mm
b = 2 mm
Filling degree (a+b)/2t

(1,2+2)/4 = 0,8 (80%).

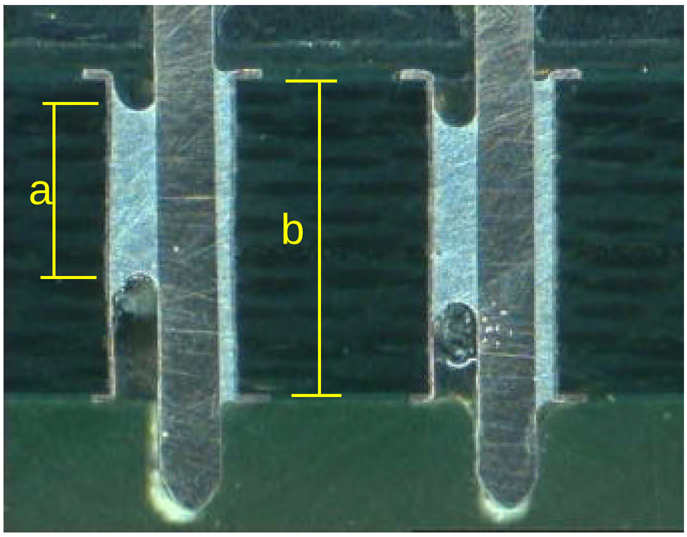

Figure 16
Solder joints with partial filling.

According to quality standard used in industry (IPC) it is not need for solder post to have protrusion on thicker boards than 2,3 mm. For thick boards, or in general if solder posts are shorter than the board thickness (compare PPPS), soldering not necessary need to go through the hole as indicated in figure17. Filling degree could than be calculated as having a board thickness of 2,3 mm, or having the board thickness equal to solder post hole penetration. This issues is up to each user to evaluate and decide on.

PPPS may be used on board with less thickness than 2,3, but in that case no quality standard exist at time for the issue of this application guide.

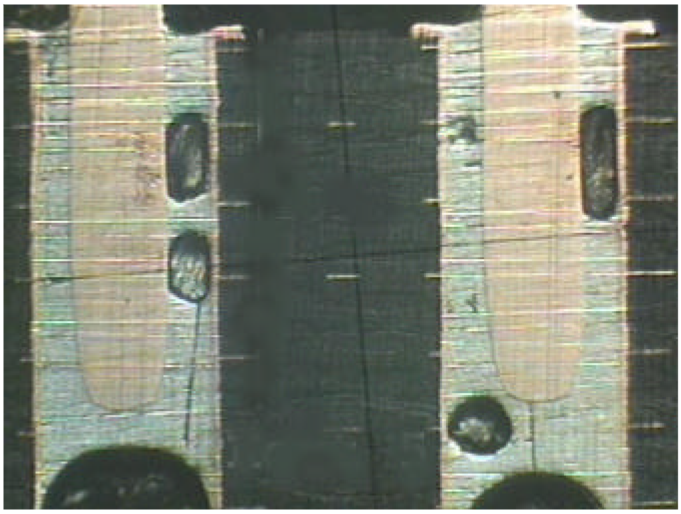

Figure 17
Solder posts shorter than board thickness

Pin-in paste connectors

GENERAL

Connectors, especially such for backplane and cables, like Metral®, Sofix®, DensiShield and d-subs brings some specific requirements for appropriate SMT-compatibility.
These are:
- Ability to carry forces
- Position after assembly
  Vertical (board level related)
  Horizontal (board position related)

Special attention has been taken in the development of SMT compatible connectors to fulfil these requirements.

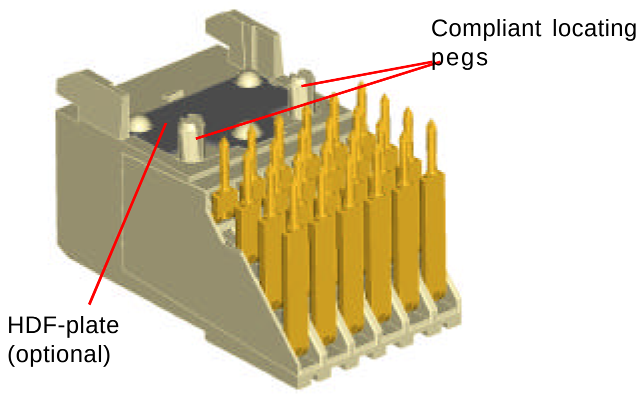

Figure 18
Hold-down and location features at Metral

ABILITY TO CARRY FORCES

The forces exposed to the connectors comes from handling, mating and unmating. Normally locating pegs and the soldering of the tails carry those forces well. In some cases depending on the connector design, additional features for force retention is used. On Metral® connectors an optional hold-down feature (HDF) has been developed (see fig 18), for other connectors different types of posts or harpoons are used.

The HDF shall be soldered to a corresponding pad at the board, and the harpoon is soldered in a plated through mounting hole.

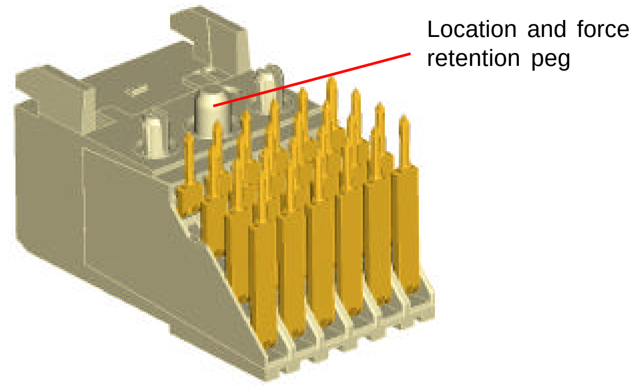

Figure 19
Location and force retention peg at Metral® TINT PIP

POSITIONING

Normally SMT components are self-aligning to their pads by allowing them to float. For a connector with its references to the board pattern/surface, housing placed on the board surface and sometimes rather heavy, it is often necessary to control the position. A compliant locating peg, made in the housing as a plastic peg, or added as a metal clip, has to cope with both limitations in mounting force (typically max 20 N) and to keep the connector in place after mounting prior to soldering (typically minimum 5 N).

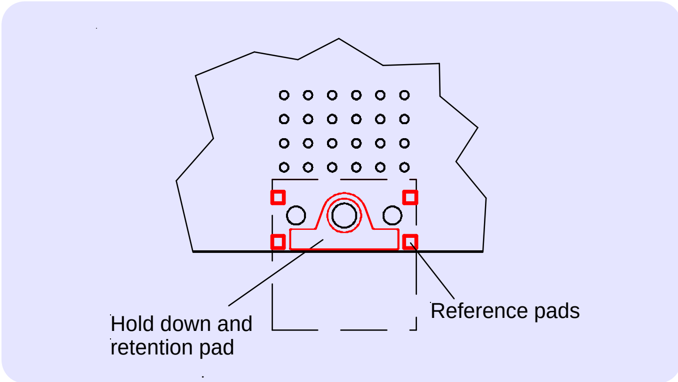

Figure 20
Hold down and reference pads at Metral® connectors

For positioning versus board surface, the recommended layout for Metral® connectors with HDF are provided with 4 level reference pads (1x1 mm) touching the housing in each of the corners (see figure 20). These pads shall be surface treated (tin, gold, OSP etc) the same as the pads of the whole board, but no solder paste shall be applied. The reference pads bring the connector to the reference level of the board (primary side). This is important for co-planarity if the connector has a hold-down feature (HDF) as described above. Also in case of mixed Metral on same board edge it is recommended to have reference pads also on those not having HDF.
Another version of a hold down feature is the counterweight design, shown in figure 21
Read also in clause "Hold down features"

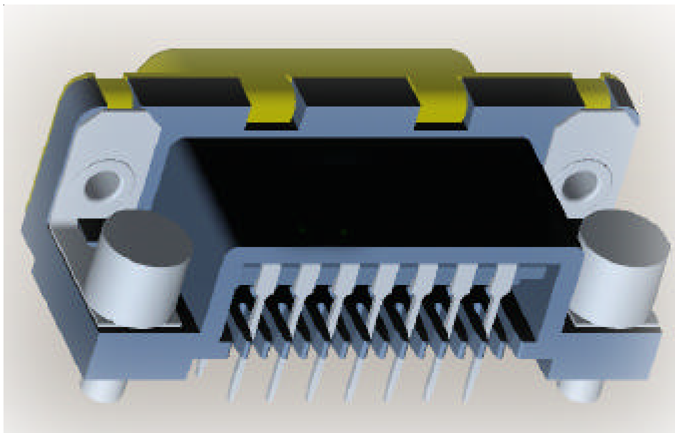

Figure 21
Counterweight type of hold-down feature(D-sub)

BOARD LAYOUT

The recommended board layout is shown on the Customer Drawings of each product (If not available, ask for it from your local sales office or look into www.fciconnect.com).

Pay attention to the hole size tolerances for the compliant locating pegs (often +0/-0,05 mm) as the control of retention force at assembly is maintained by those holes.

SOLDER MASK APPLICATION

To avoid disturbance in the reference surface of connector/board it is recommended that solder mask is never printed on the reference pads underneath the housing. To achieve that the area can be without solder mask. See figure 22. (only applicable at Metral® connectors when using reference pads).

Solder mask has to be applied on all surfaces where solder paste is printed except on the solder pads.

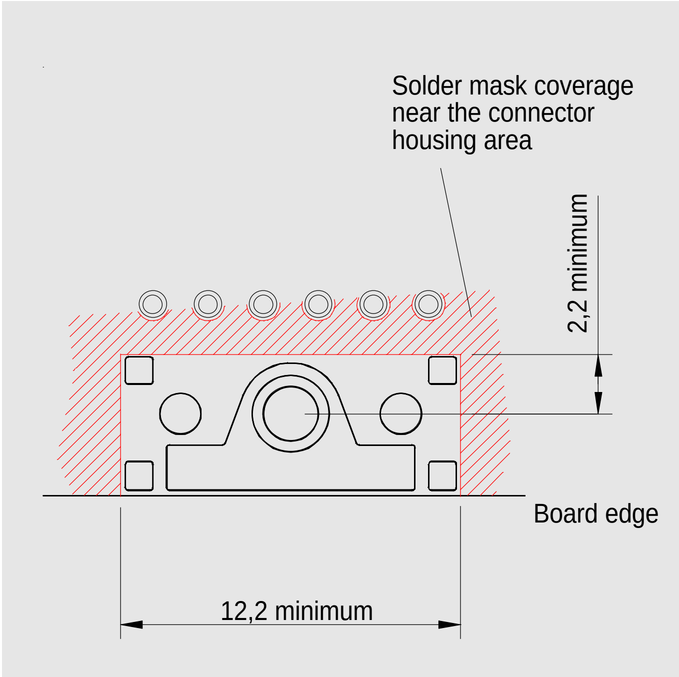

Figure 22
Recommended solder mask opening

SOLDER PASTE APPLICATION

Solder paste at the hole grid is recommended as in figures 23 for 4 row Metral® connectors. It is preferred to utilize the area with rectangular shaped pads, maximum sizes as shown.
For other connector types see actual customer drawing or application specification.

It is important that the paste pad configuration is adapted to actual connector, board thickness, hole size, paste applicator, stencil thickness etc. What is recommended below and/or at our customer drawing is general advice.

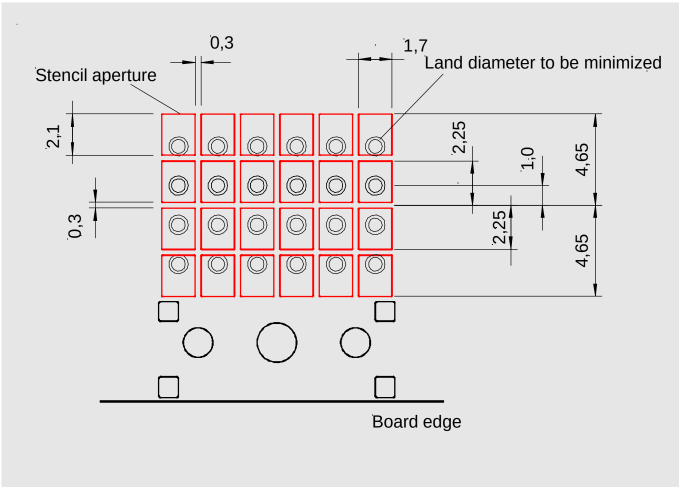

Figure 20
Proposal for maximum amount of paste applied at the pin grid of the 4-row Metral connectors.

Some caution needs to be taken if the aperture edge of the screen printing stencil is resting on the land of a hole; leakage is a risk. It is therefore recommended to minimise land diameter (Nominal hole diameter + 0,4 mm) . Apertures could also have a shape avoiding the land (see figure 24).

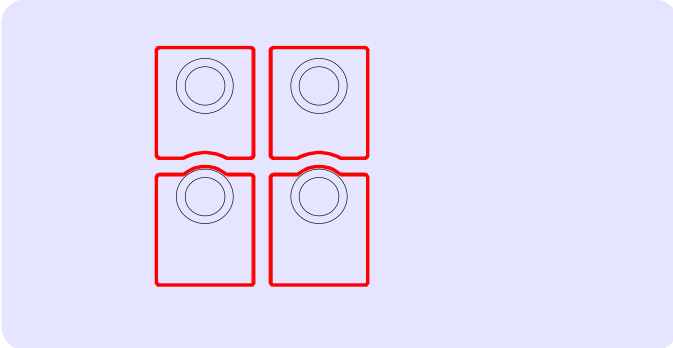

Figure 24
Screen printing stencil apertures to reduce paste leakage. Lands to be minimized (Nominal hole diameter + 0,4 mm)

UTILIZATION OF THE AREA FOR PRINTING PASTE

In some cases, especially on connectors with few solder posts, the area can be utilized as much as possible, as long as the melting paste have a distance to be sucked into the hole less than 2 mm. In figure 25 the maximum paste printing is shown for Sofix® Shielded I/O 8 pos connector.

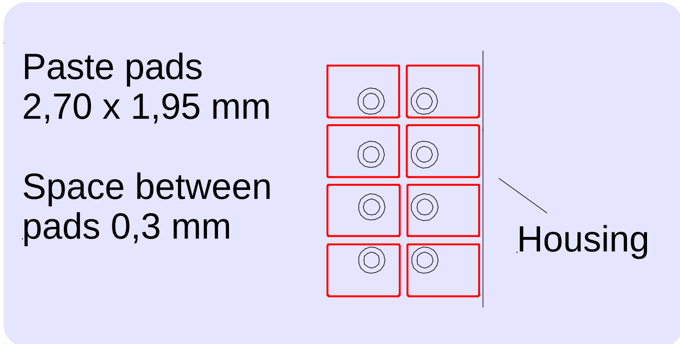

Figure 25
Maximum use for printing paste with unsymmetrically placed solder paste pads.

PASTE APPLICATION CLOSE TO THE CONNECTOR

Some of the connector housings are provided with a recess to allow for paste to be applied for soldering close to or even under the housing. A recess is also normally present for a hole having a harpoon type mounting. The customer drawing is providing the actual information. Minimum distance to a housing is 0,1 mm for automatic assembly (Pick and place), and 0,3 mm for manual assembly.

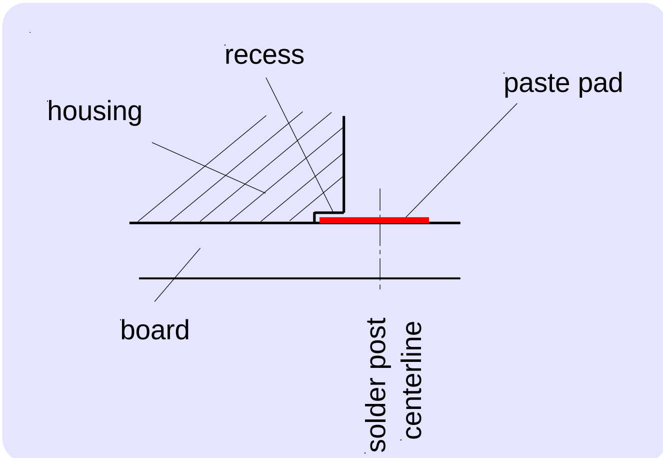

Figure 26
Housing recess for solder paste

GUIDING WAFER STAND-OFF

Some connectors are equipped with a guiding wafer to ensure proper true position of the solder tails. To give space for solder paste to move and reflow correctly, such guiding wafer has stand-offs placed in the space between the proposed solder paste pads.
The position of the stand-offs can be found on the customer drawing of the connector.
To avoid disturbance in the paste pads it is recommended to make small adjustments in the apertures as shown in figure 27 and 28.

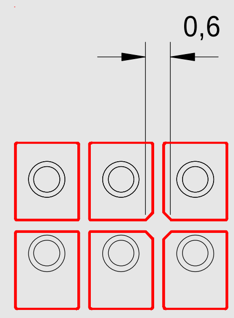

Figure 27
Reduction of paste pad in the intersection (recommended) due to wafer stand-off

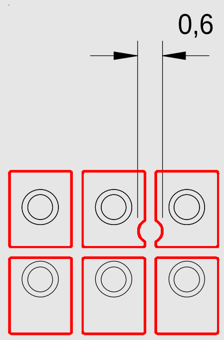

Figure 28
Reduction of paste if stand-off is not fitting in the intersection

HOLD DOWN FEATURES

Hold down features (HDF) are in some cases shaped as seen in figure 15. A corresponding pad at the board for soldering to the HDF-plate is recommended in the actual customer drawing. The paste printing stencil apertures are recommended as in figure 29. In the case with a hole in size 2 mm it is not recommended to print over the hole, but make the apertures divided in smaller portions. It is however important that it will be soldered almost around the whole hole.
A special variant of a hold down feature is the counterweight used on the d-sub connector range. See Figure...
This is effectively reducing the risk for the connector to tilt over the board edge.

For paste printing on harpoon holes, often of 2,5 mm diameter or more, it may be necessary to have a beam in the stencil to avoid too much leakage of paste through the hole.

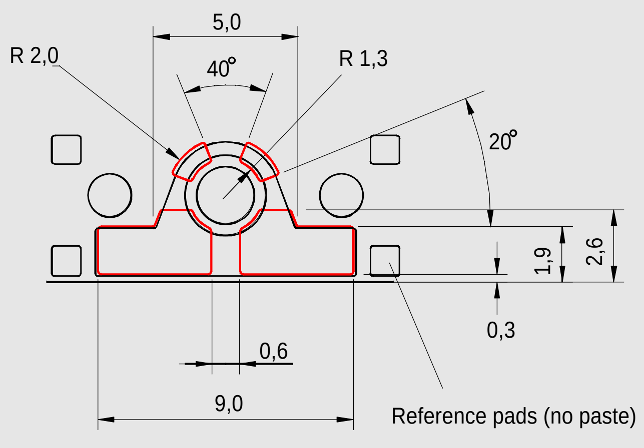

Figure 29
Proposal for paste application on the hold-down feature area

Summary

Hopefully this guide help you to get success with the PiP application you are dealing with. But specific information for the product, as well as for process and board issues can not be fully covered, so for further knowledge it is necessary to collect such information.
But for the improvement of this guide comments and proposals are welcome.
Please contact your local FCI-representative.