# Conformal Selective Wave Solder Carriers – Design Rules

The design of an exclusion fixture is “non trivial”.

As well as checking the process suitability, calculations need to be done to ensure:

- Carrier stability and longevity – Euler Buckling of the masking due to differential heating
- Minimal thermal deflection – caused by the thermal gradient across the carrier
- Optimal solder flow – maximally sculpting locally to the PTH while maintaining overall stiffness

These calculations are only aids and we have found that successful designs evolve with experience.

Our first exclusion fixture was over 20,000 jobs ago.

## Process Suitability

The viability of this process route crucially relies upon being able to estimate the number and distribution of PTH components:

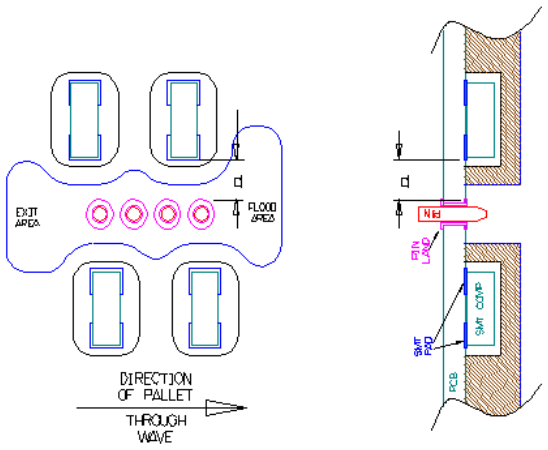
- Which cannot be processed (with and without glue dotting) – These will have to be mini-waved or hand soldered.
- Which are likely to be poorly soldered – these will require further inspection and manual touch up.

This estimation may be done in three ways:

- If a PCB is available (preferably populated) – our sales engineers can rapidly evaluate your board.
- If PCB design data is available we will process, analyse and remotely assess it.
- You can do it using the rules presented below – our customers quickly find that the above two methods are easiest.

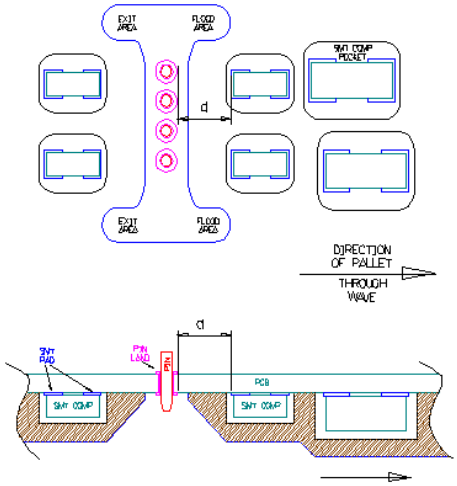
**Pin Land to SMT pad clearance evaluation**

The two figures below each show part of a CSWSC in plan and section views. The right hand figure shows that more clearance
is required when the connector orientation is perpendicular to the wave.




**PTH Components Located Parallel to direction through wave**

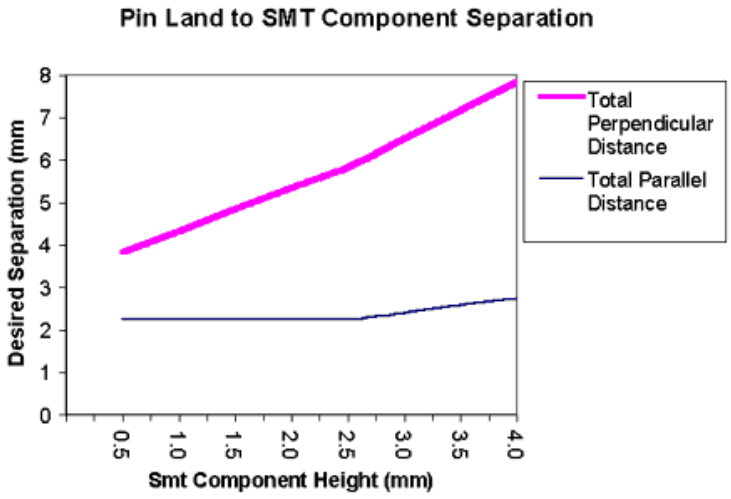
The clearance required between the pin land and SMT pad can be made quite small, as the solder does not have to flow "under" the component pockets.




**PTH Components Located Perpendicular to direction through wave**

A larger separation is required due to the solder having to flow "under" component pockets.

Working your way around the underside of the PCB, identify which components are parallel and perpendicular to the wave and assess the solderability of each PTH connector by comparing the actual separation against the graph right. Ideally you want to be above the line in all cases.



**PCB Design Implications – for Board Designers – or re-spin**

We are often called upon by our customers to help with identifying design respin opportunities.

We will identify problem areas within a board and suggest appropriate movements of components. (Ideally before the PCB is fabricated)

However for board designers reading this, can you remember another four "rules" (to compete with the hundred other rules you have to have floating
around in your head).

- Keep large (height) SMT components away from PTH areas.
- Leave the leading and trailing areas around PTH components as clear as possible.
- DON'T put any SMT components within 3mm (0.12") of any PTH components.
- DON'T put all PTH components in line along one edge of a board – leave some space to allow us to support the masking in the centre of the board.

Reference: Global Datum. (2017). 'Conformal Selective Wave Solder Carriers – Design Rules.' Retrieved from http://globaldatum.co.uk/cswsc-design-rules/