

Stencil Order Report

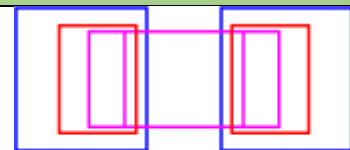
Customer Name	#####	Date	12/15/2015
Customer Part#	SPM-009227	Order#	165512
Metal Thickness	4 Mil	Metal Type	Datum Laser PhD
Coating Type	None	Stencil Type	Paste

Order Special Instructions

We would like to have a new stencil with a thickness of 4 mils and use home plate apertures to eliminate solder balls.

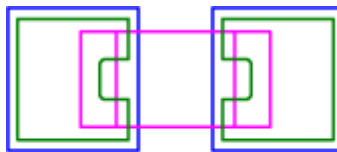
Design Details Requiring Approval

We found that SMT land pads of 0603 chips were made bigger than recommend as shown in the picture to the right. Using the same size for apertures with home plate design will most likely not eliminate solder balls as extra solder would be released underneath the chip terminals.



Red: Recommended Pads
Blue: Designed PCB Pads
Magenta: 0603 chip layout

To resolve the solder balls issue, we recommend applying 15% reduction to the SMT pads and to use U-Shape design for the apertures, this type of design proved better results than standard home plate apertures for resolving solder balls as it prevents releasing solder paste at the center of the chip terminals. Resulted apertures as per this recommendation would be the green pads in the picture below and that's what we've applied to attached modified gerber:



Green: Stencil Apertures
Blue: Designed PCB Pads
Magenta: 0603 chip layout

Please advise if you agree with this design or would like to change it.

FYI Design Details

We found many locations in the provided paste layer were modified, apertures were made 1:1 with paste for such locations, and for all non-modified ones we applied standard reduction as per your profile.

Other Notes

Please note that the size mismatch issue was only found for 0603 chip components, if you are facing solder balls with other chip packages please let us know.

Customer Feedback

Thank you very much and you can proceed with this solution.
 The solder balls appear on 0603 size only, no special consideration is needed for other chips.
 All other modifications are approved, you can start production.

Name #####