ZHX1200 Solderability Hints

TABLE OF CONTENTS

ZHX1200 Solderability Hints 3
Appendix 4
Recommended 6

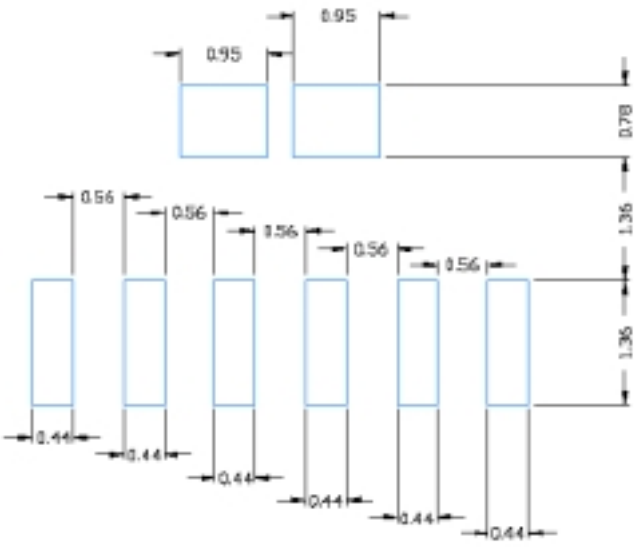
Information Integrity

The information contained within this document has been verified according to the general principles of electrical and mechanical engineering. Any applicable source code illustrated in the document was either written by an authorized ZiLOG employee or licensed consultant. Permission to use these codes in any form besides the intended application, must be approved through a license agreement between both parties. ZiLOG will not be responsible for any code(s) used beyond the intended application. Contact your local ZiLOG Sales Office to obtain necessary license agreements.

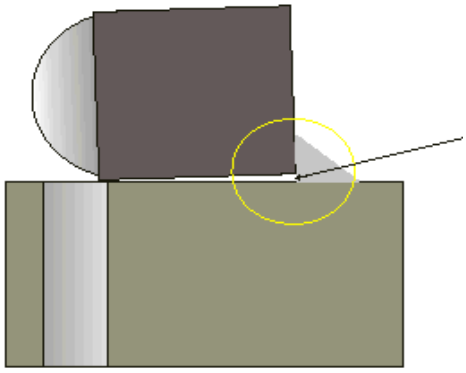
Document Disclaimer

ZHX1200 Solderability Hints

1. The connection pads on the ZHX1200 are copper paste filled and do not wet as easily as the connection backside contact and the shield ground tab.
2. The PCB solder pad spacing for the IrDA module need to be optimized to match the ZHX1200 module connection pads. They should not be longer than they have to be since this might make it harder to better control the movement of the module along the length of the pads during reflow.
3. ZiLOG manufacturing partners have found that having solder paste underneath the entire connection pads of the ZHX1200 has some potential solderability issues which they resolved in their pre-production runs. They have shared their solder stencil aperture design with ZiLOG. Their current solderability defect rate is less than 0.5% with their approach. The technique is to position the solder paste so the backside contacts wets quickly. See below and the appendix for details of their analysis and recommended solder stencil design.

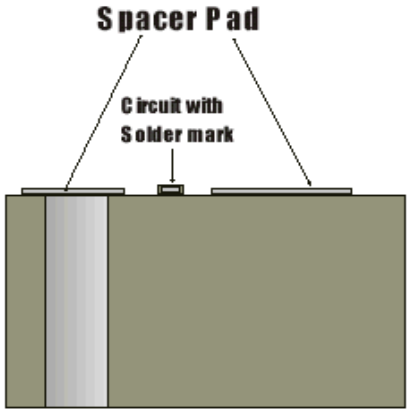


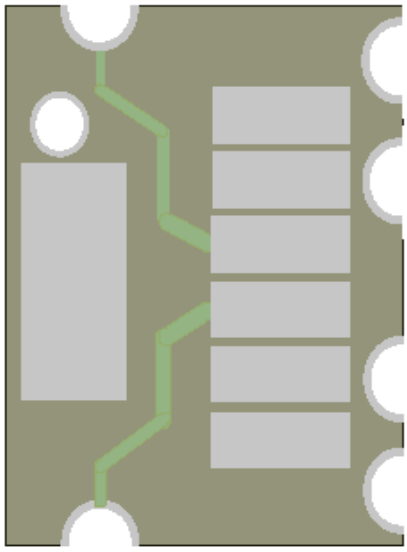
Appendix

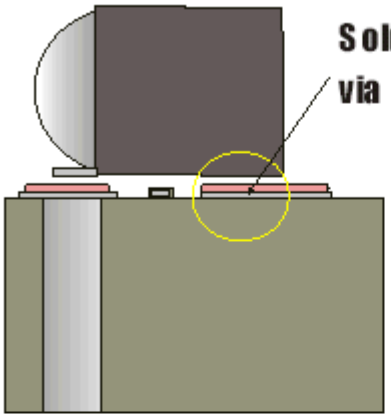


1. Gap between PCB and module

Since PCB design has traces between the ground tab pad and the connection pad plus solder mask to cover the traces, total height of the trace and solder mask is higher than the connection pads and ground tab pad. After the module mount on the spacer, and reflow, there will be a small gap between the spacer and the module. The gap has effect on the quality of solderability and affect filleting.



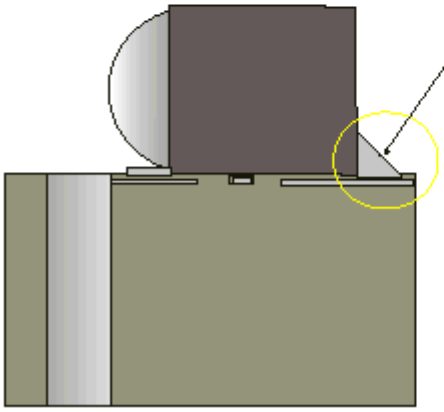




The pict
are on v
stencil a

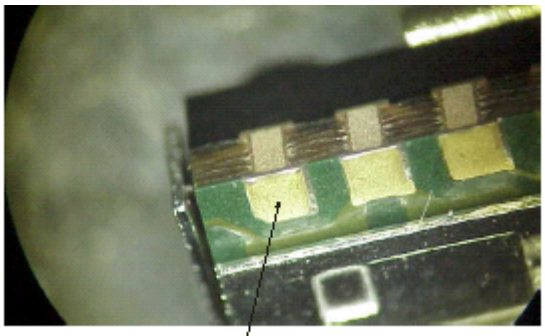
2. Misalignment

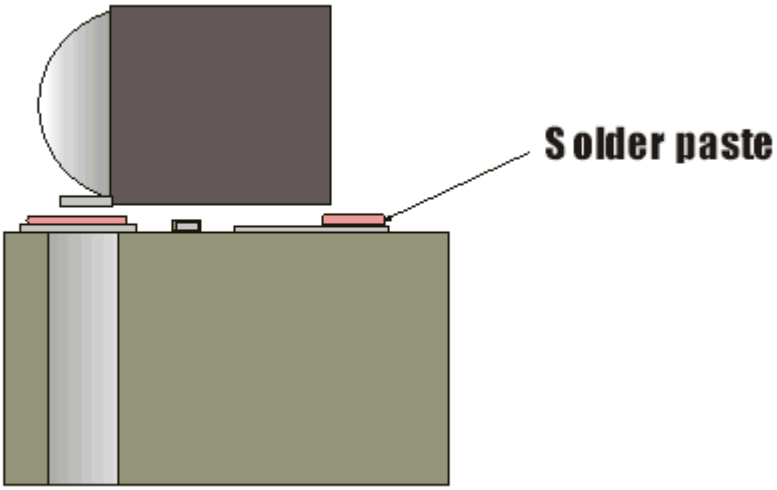
Since the connection is copper filled, solder will connect to the outside pad more than the via hole.



The pictu

If the stencil had big apertures in the via hole area, after SMT process there will be misalignment and gap between the spacer and the module.





Recommended

1. Reduce solder paste stencil aperture.
2. Stencil thickness should be 7 to 8 mils.
3. See stencil drawing below.

Manufacturing Partner's Temperature Profile Using Multicore Sn63/Pb37 CR36 AGS 89.5