

Capillary test of flux penetration

Test method to determine the effectiveness of the capillary behaviour of flux.

Introduction

The first demand for soldering is that the parts to be soldered are clean. Therefore we need a flux on all parts to be soldered. Often it is difficult to prove if the flux did penetrate sufficiently in capillaries between hole and lead. Specially if those holes are covered on the topside of the PCB by the component body.

Test method

The capillary behaviour is depending on the gap dimensions, the wetting capacity and the amount of flux.

If one of these factors is inadequate the flux will not be able to cover all parts that must be soldered.

To control if the flux does come in contact with all parts a thin paper tissue can be placed between the component and the PCB topside. If this tissue becomes wet than the flux has risen to the top of the board.

Since the tissue may create a distance between the component and the PCB there is no guarantee that the flux will also penetrate sufficiently when no tissue is present. For a quick check, the component should be removed after fluxing, but before that, the flux on the underside of the PCB and at the lead should be wicked away, e.g. by using a tissue. This is necessary to prevent the excess flux at the underside of the lead to be dragged into the hole, since this could give false information about the flux penetration.

How to use this test to optimize fluxer settings

In case one has a poor solder rise in the connection, one issue could be lack of flux. With this method one can easily control if the flux did rise sufficiently. Also one can see if too much flux was applied when the tissue is soaked with flux. In this way the method can be used to optimized the fluxer settings. This is specially the case when a dropjet fluxer is used, where one has the possibility to adjust the fluxer settings individually for each part that must be fluxed.

Flux amount

For correct fluxing the calculations in the appropriate documents (in Technical Reference) can be used. However often a practical approach is to start to apply the maximum allowable amount of flux and reduce until a reasonable amount of residues is left on the PCB and still a good solder result is maintained.

Vitronics Soltec

DISCLAIMER

All content is subject to periodic review and may be changed without notice.
Vitronics Soltec BV assumes no obligation for content contained herein.

COPYRIGHT VITRONICS SOLTEC BV

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording or otherwise, without the prior written permission of Vitronics Soltec BV. This publication remains the property of Vitronics Soltec BV and may not be passed, loaned or given to any third party.

Vitronics Soltec BV reserves the right to make changes in design and specifications without notice.