IEC TR 61760-3-1

Edition 1.0 2022-06

TECHNICAL REPORT

colour inside

Surface mounting technology –
Part 3-1: Standard method for the specification of components for through hole reflow (THR) soldering – Guidelines for through hole diameter design with solder paste surface printing method

IEC TR 61760-3-1:2022-06(en)

IEC Secretariat
3, rue de Varembé
CH-1211 Geneva 20
Switzerland

Tel.: +41 22 919 02 11
info@iec.ch
www.iec.ch

About the IEC
The International Electrotechnical Commission (IEC) is the leading global organization that prepares and publishes International Standards for all electrical, electronic and related technologies.

About IEC publications
The technical content of IEC publications is kept under constant review by the IEC. Please make sure that you have the latest edition, a corrigendum or an amendment might have been published.

IEC publications search - webstore.iec.ch/advsearchform
The advanced search enables to find IEC publications by a variety of criteria (reference number, text, technical committee, …). It also gives information on projects, replaced and withdrawn publications.

IEC Just Published - webstore.iec.ch/justpublished
Stay up to date on all new IEC publications. Just Published details all new publications released. Available online and once a month by email.

IEC Customer Service Centre - webstore.iec.ch/csc
If you wish to give us your feedback on this publication or need further assistance, please contact the Customer Service Centre: sales@iec.ch.

IEC Products & Services Portal - products.iec.ch
Discover our powerful search engine and read freely all the publications previews. With a subscription you will always have access to up to date content tailored to your needs.

Electropedia - www.electropedia.org
The world's leading online dictionary on electrotechnology, containing more than 22 300 terminological entries in English and French, with equivalent terms in 19 additional languages. Also known as the International Electrotechnical Vocabulary (IEV) online.

IEC TR 61760-3-1

Edition 1.0 2022-06

TECHNICAL REPORT

Surface mounting technology –
Part 3-1: Standard method for the specification of components for through hole reflow (THR) soldering – Guidelines for through hole diameter design with solder paste surface printing method

INTERNATIONAL
ELECTROTECHNICAL
COMMISSION

ICS 31.190

ISBN 978-2-8322-3928-5

CONTENTS

INTERNATIONAL ELECTROTECHNICAL COMMISSION

____

SURFACE MOUNTING TECHNOLOGY –

Part 3-1: Standard method for the specification of components for through hole reflow (THR) soldering – Guidelines for through hole diameter design with solder paste surface printing method

FOREWORD

1) The International Electrotechnical Commission (IEC) is a worldwide organization for standardization comprising all national electrotechnical committees (IEC National Committees). The object of IEC is to promote international co-operation on all questions concerning standardization in the electrical and electronic fields. To this end and in addition to other activities, IEC publishes International Standards, Technical Specifications, Technical Reports, Publicly Available Specifications (PAS) and Guides (hereafter referred to as "IEC Publication(s)"). Their preparation is entrusted to technical committees; any IEC National Committee interested in the subject dealt with may participate in this preparatory work. International, governmental and non-governmental organizations liaising with the IEC also participate in this preparation. IEC collaborates closely with the International Organization for Standardization (ISO) in accordance with conditions determined by agreement between the two organizations.
2) The formal decisions or agreements of IEC on technical matters express, as nearly as possible, an international consensus of opinion on the relevant subjects since each technical committee has representation from all interested IEC National Committees.
3) IEC Publications have the form of recommendations for international use and are accepted by IEC National Committees in that sense. While all reasonable efforts are made to ensure that the technical content of IEC Publications is accurate, IEC cannot be held responsible for the way in which they are used or for any misinterpretation by any end user.
4) In order to promote international uniformity, IEC National Committees undertake to apply IEC Publications transparently to the maximum extent possible in their national and regional publications. Any divergence between any IEC Publication and the corresponding national or regional publication shall be clearly indicated in the latter.
5) IEC itself does not provide any attestation of conformity. Independent certification bodies provide conformity assessment services and, in some areas, access to IEC marks of conformity. IEC is not responsible for any services carried out by independent certification bodies.
6) All users should ensure that they have the latest edition of this publication.
7) No liability shall attach to IEC or its directors, employees, servants or agents including individual experts and members of its technical committees and IEC National Committees for any personal injury, property damage or other damage of any nature whatsoever, whether direct or indirect, or for costs (including legal fees) and expenses arising out of the publication, use of, or reliance upon, this IEC Publication or any other IEC Publications.
8) Attention is drawn to the Normative references cited in this publication. Use of the referenced publications is indispensable for the correct application of this publication.
9) Attention is drawn to the possibility that some of the elements of this IEC Publication may be the subject of patent rights. IEC shall not be held responsible for identifying any or all such patent rights.

IEC TR 61760-3-1 has been prepared by IEC technical committee 91: Electronics assembly technology. It is a Technical Report.

The text of this Technical Report is based on the following documents:

Draft	Report on voting
91/1734/DTR	91/1773/RVDTR

Full information on the voting for its approval can be found in the report on voting indicated in the above table.

The language used for the development of this Technical Report is English.

This document was drafted in accordance with ISO/IEC Directives, Part 2, and developed in accordance with ISO/IEC Directives, Part 1 and ISO/IEC Directives, IEC Supplement, available at www.iec.ch/members_experts/refdocs. The main document types developed by IEC are described in greater detail at www.iec.ch/standardsdev/publications.

A list of all parts in the IEC 61760 series, published under the general title *Surface mounting technology*, can be found on the IEC website.

The committee has decided that the contents of this document will remain unchanged until the stability date indicated on the IEC website under webstore.iec.ch in the data related to the specific document. At this date, the document will be

- reconfirmed,
- withdrawn,
- replaced by a revised edition, or
- amended.

IMPORTANT – The "colour inside" logo on the cover page of this document indicates that it contains colours which are considered to be useful for the correct understanding of its contents. Users should therefore print this document using a colour printer.

SURFACE MOUNTING TECHNOLOGY –

Part 3-1: Standard method for the specification of components for through hole reflow (THR) soldering – Guidelines for through hole diameter design with solder paste surface printing method

1 Scope

This Part of IEC 61760 supplements IEC 61760-3 to describe examples of solder paste supply methods, the relationship between the terminal position tolerance and the through hole diameter, and provides guidelines for the design of printed circuit boards with solder paste surface printing method, including specific examples.

2 Normative references

The following documents are referred to in the text in such a way that some or all of their content constitutes requirements of this document. For dated references, only the edition cited applies. For undated references, the latest edition of the referenced document (including any amendments) applies.

IEC 61760-3:2021, *Surface mounting technology – Part 3: Standard method for the specification of components for through-hole reflow (THR) soldering*

3 Terms and definitions

For the purposes of this document, the terms and definitions given in IEC 61760-3 and the following apply.

ISO and IEC maintain terminological databases for use in standardization at the following addresses:

- IEC Electropedia: available at http://www.electropedia.org/
- ISO Online browsing platform: available at https://www.iso.org/obp

3.1
stencil aperture
stencil opening
opening area for solder paste printing in the stencil

4 Solder paste application methods

4.1 General

The amount of solder paste required for through-hole reflow soldering is much greater than for surface mount. In the industry marketplace, various methods to supply the solder paste are utilised. All these methods have specific advantages and disadvantages and are depending upon the board lay-out, the solder material to be used, the trough-hole components to be inserted. Other points to be considered are production efficiency, cost and quality requirements.

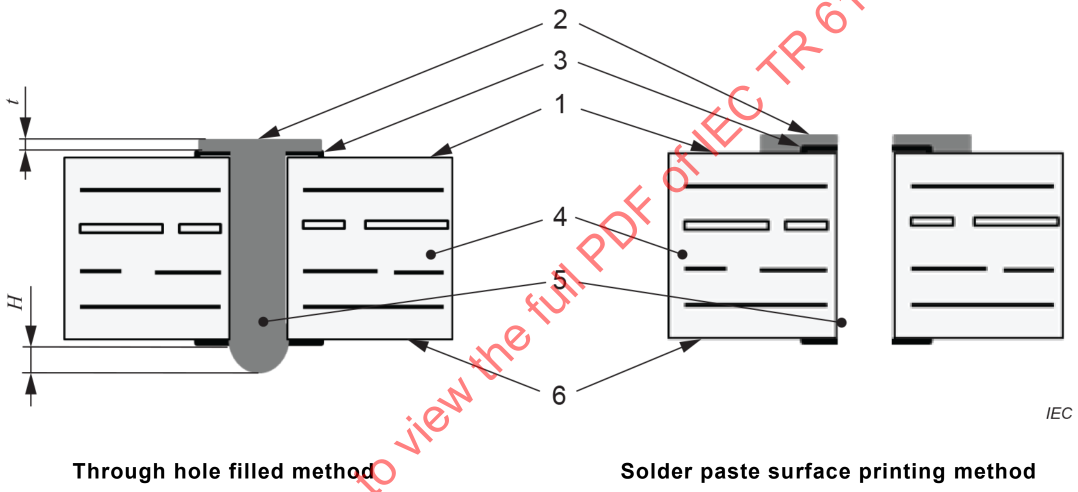
The most common method used is printing solder paste into plated through-holes and over the pad surface during normal surface mount paste printing (solder paste surface printing method, see 4.3). The size of the aperture in the solder paste stencil may be adjusted to allow paste to fill the hole, cover the pad or extend over the surface of the solder mask to obtain the solder volume required.

Another method is the solder paste surface printing method (through hole vacant method), see 4.3.

Refer to Figure 1 to Figure 4 for comparison of both methods.

In this document, the features (advantage and disadvantage) of the solder paste surface printing reflow technology are introduced from the following two perspectives:

- Solder paste printing stencil aperture design guidelines (see Clause 5);
- Influence of position tolerance between component terminals and through holes on through hole diameter and stencil aperture design guidelines (see Clause 6).



key

1	top side (assembly side)
2	solder paste
3	Land
4	circuit board
5	through hole
6	bottom side
t	solder paste thickness
H	solder paste protrusion height

Figure 1 – Solder paste application method

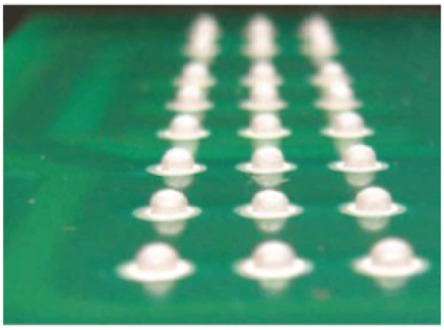
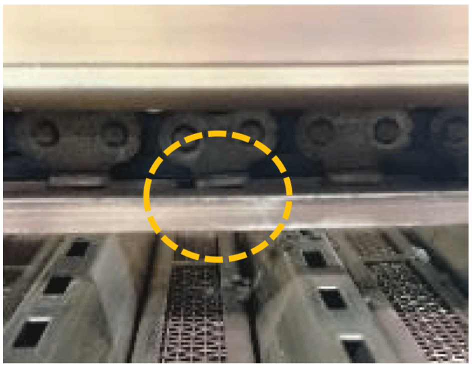
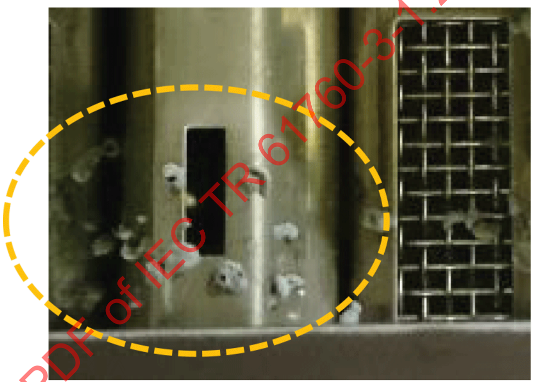


Figure 2 – Through hole filled method (bottom side view)



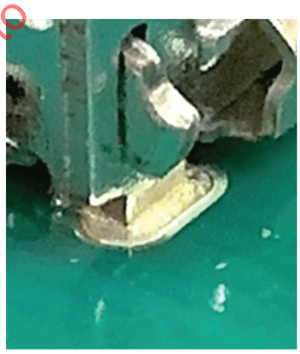
Reflow oven side view



Reflow oven top view

Figure 3 – Solder paste fallen inside the reflow oven (through hole filled method)

a) After print



b) After reflow (top side)

c) After reflow (bottom side)

Figure 4 – Solder paste state of solder paste surface printing method

4.2 Through hole filled method

4.2.1 Overview

As the name indicates, the through hole filled method is a method of filling the through hole with the required amount of solder paste using a printing method or a dispenser.

4.2.2 Advantages

The advantage of this method is that it is applicable in cases where the solder paste surface printing method does not provide enough space for solder printing, such as when the distance between adjacent through holes or between through holes and adjacent component electrodes is close.

4.2.3 Disadvantages

A disadvantage of this method is that the solder paste can drop down during the surface mounting process. The reason for this is that the solder paste filled in the through hole is pushed down by the terminal when the component terminals are inserted.

In addition, the amount of filled solder paste varies depending on various printing conditions, such as printing speed, printing pressure and squeegee angle, and it is difficult to check.

4.3 Solder paste surface printing method

4.3.1 Overview

The solder paste surface printing method is a method of filling a through hole by melting solder of which the solder paste printed in vicinity of the through hole on the circuit board by reflow heating and then spread wetting it to the through hole sides and component terminals.

This method has been popularized recently as a means of reducing of dropping solder paste, which is a disadvantage of the hole-filling method. However, since the through hole is not filled with solder paste, it is necessary to ensure that the volume (actually, about twice the volume to include flux, etc.) of the solder paste as a surface printing area in the vicinity of the component arrangement. Another disadvantage is that if the distance from the through-hole is increased to secure the printing area, it is difficult for the printed solder paste to move into the through hole during the melting phase.

4.3.2 Advantages

The amount of solder paste required can be adjusted by designing the stencil opening area (see Figure 4).

The risk of solder paste dripping into the reflow oven during the soldering process is reduced (see Figure 3).

Since the amount of solder paste depends on the stencil aperture, the amount of solder paste can be easily adjusted and stabilized.

4.3.3 Disadvantages

Adequate printing area of solder paste on components mounting surface is required (see Figure 4).

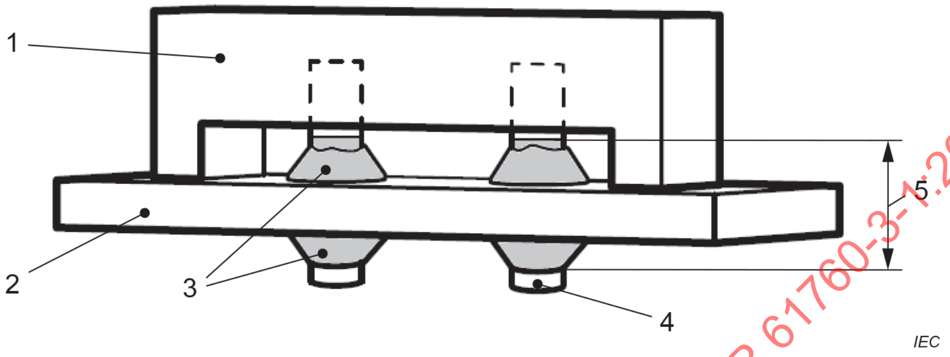
It can be difficult to secure the amount of solder paste supply (see Figure 4).

The solder paste does not move to the through holes when the solder paste is melted if the supply position of solder paste is separated from the through holes by more than a certain distance.

5 Solder paste printing stencil aperture design guidelines

5.1 General

The required solder amount for a component terminal is the sum of the clearance volume in between through hole and the component terminal, and the solder fillet volume on the component terminal of top and bottom side (see Figure 5).



key

1	component body
2	circuit board
3	solder fillet
4	component terminal
5	terminal solder wetting length

Figure 5 – Conceptual view of solder wetting of component terminals after reflow soldering

During reflow process, The printed solder paste will shrink to almost a half of the printed volume. See Annex B, Formula (B.1).

5.2 Amount of solder paste

5.2.1 Needed amount of solder paste

The solder paste surface printing method assumes stencil printing, and the stencil aperture geometry is determined from the amount of solder to be supplied as described in 5.1.

The principle, how to calculate the amount of solder paste to be supplied is shown in Formula (1). See Annex A for details of the calculation methods for solder fillet volume of the round terminal.

$$A = (B - C) \times x \tag{1}$$

where

A is the amount of solder paste to be supplied (mm^3);

B is the solder volume after reflow sum of to fill the through hole and required top and bottom side solder fillet volume (mm^3);

C is the volume of the component terminal in a through hole (mm^3);

x is the volume coefficient of solder paste after melting: see Annex B;

5.2.2 Printing volume of solder paste

The printing volume of solder paste is the product of printed area and the printed thickness.

Please note that the printed solder paste thickness can vary from the stencil thickness depending on printing conditions. In addition, the actual amount of solder paste can be higher than the calculated value due to the solder paste entering the through hole during stencil printing. Thus, it is recommended to check the actual soldering result and adjust the stencil aperture if necessary.

In the following, two types of solder paste printing are described, single area printing (see Figure 6) and slit printing (see Figure 7).

Besides these two methods, other pattern design and printing process approaches are used for optimization of the soldering process, which are not described in this document

Single area printing can have the same disadvantages of the through hole filled method (see Table 1). As a result, slit area printing type is utilised.

For single area printing, the minimum volume of the printing solder paste is obtained by the Formula (2). The solder volume entering the hole during printing is not considered in this formula.

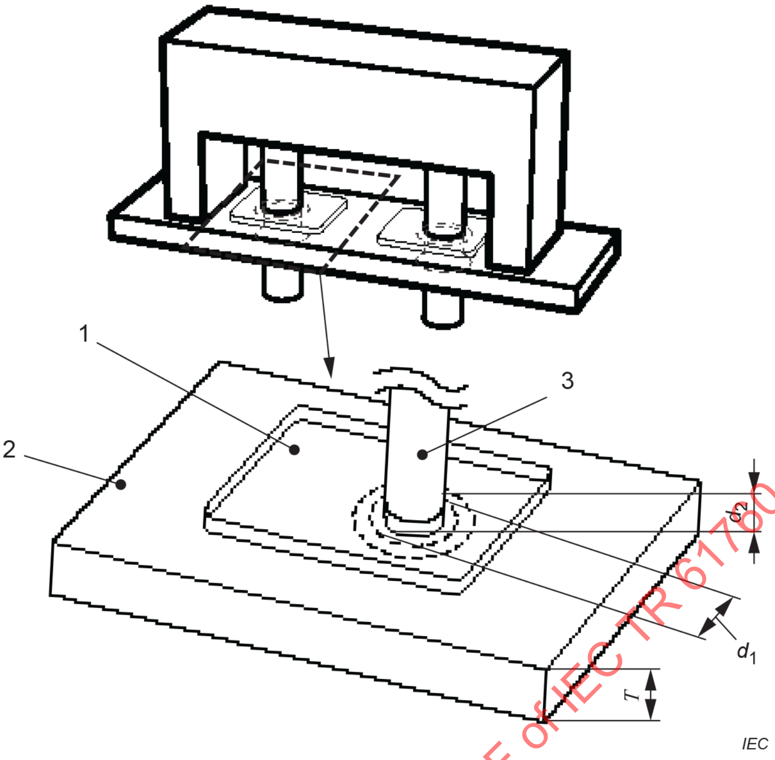
$$E = F \times t \quad (2)$$

where

E is the amount of printing solder paste (mm^3);

F is the printing area (mm^2);

t is stencil thickness (mm).



Key

1	solder paste
2	circuit board
3	component terminal
d_1	through hole diameter
d_2	component terminal diameter
T	circuit board thickness

Figure 6 – Single area printing

For slit printing, the amount of the printing solder paste is obtained by the Formula (3).

$$E = (F - G) \times t \tag{3}$$

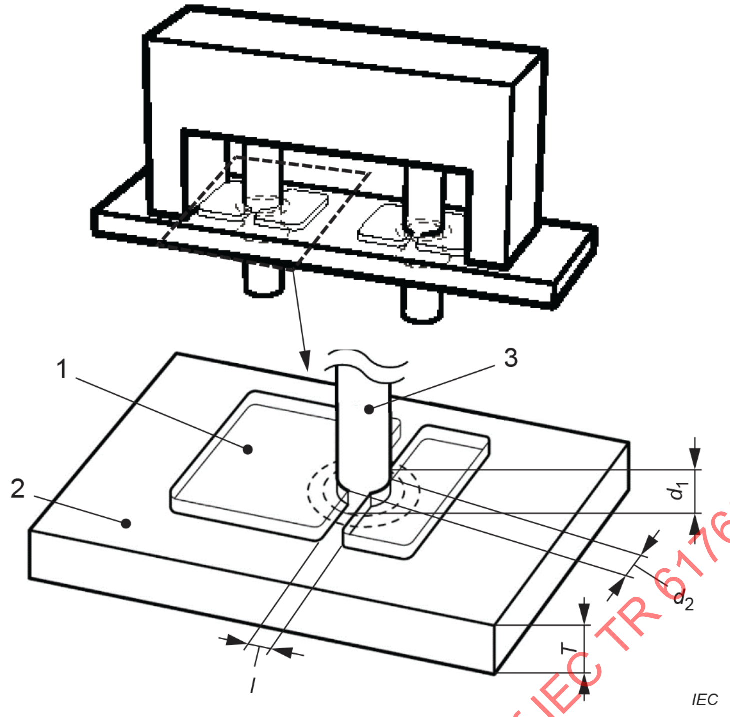
where

E is the amount of printing solder paste (mm^3);

F is the printing area (mm^2);

G is the slit area (mm^2);

t is stencil thickness (mm).



Key

1	solder paste
2	circuit board
3	component terminal
d_1	through hole diameter
d_2	component terminal diameter
T	circuit board thickness
l	slit width

Figure 7 – Slit printing

5.2.3 Designing amount of printing solder paste

The amount of printing solder paste (E) becomes equal to the required amount of solder paste to be supplied (A).

6 Influence of terminal position tolerance on through hole diameter and stencil aperture design guidelines

6.1 Terminal position tolerance

The terminal position tolerance is the allowable displacement of the centre of the component terminal from the design centre position.

According to IEC 61760-3:2021, 4.6.5.2 the position tolerance of each pin tip should be specified to a value between 0,2 mm and 0,4 mm in diameter.

6.2 Relationship between terminal position tolerance and through hole diameter

6.2.1 General

For the same terminal shape and dimension, a larger terminal position tolerance will require, a larger hole diameter and a larger area of the printed solder paste. The smaller the terminal position tolerance gets, the smaller printing area can be designed. If enough space is available in the vicinity of the THR components, the degree of freedom to select the position tolerance increases. For higher pin count or narrower pitch applications, normally tighter position tolerances, e.g. 0,3 mm or 0,2 mm, are applied.

The examples described in 6.2.2, 6.2.3 and 6.2.4 illustrate how to calculate the terminal position tolerance needed for specific circuit board designs. In the 6.2.2, 6.2.3 and 6.2.4, the relationship between the terminal position tolerance and the through hole diameter is shown for a terminal diameter of 1,0 mm.

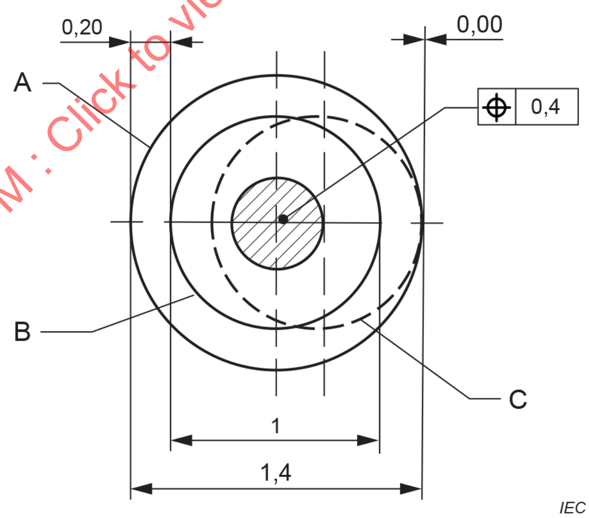
6.2.2 Terminal position tolerance of 0,40 mm

Example 1 is as follows:

– through hole diameter: 1,40 mm;
– terminal diameter: 1,0 mm;
– position tolerance: 0,40 mm;
– design gap on each side: 0,20 mm.

If the actual displacement of the terminal is up to the maximum terminal position tolerance of 0,20 mm, the clearance between the terminal and the finished hole wall becomes 0,00 mm (see Figure 8). Considering the insertion variation of the mounting machine, an enlargement of the through hole diameter will become necessary.

Dimensions in millimetres



Key

A circuit board through hole diameter

B terminal position (centre of position tolerance)

C terminal position (maximum displacement to terminal position tolerance)

Figure 8 – Terminal position tolerance 0,40 mm, through hole diameter 1,40 mm (one side clearance 0,20 mm)

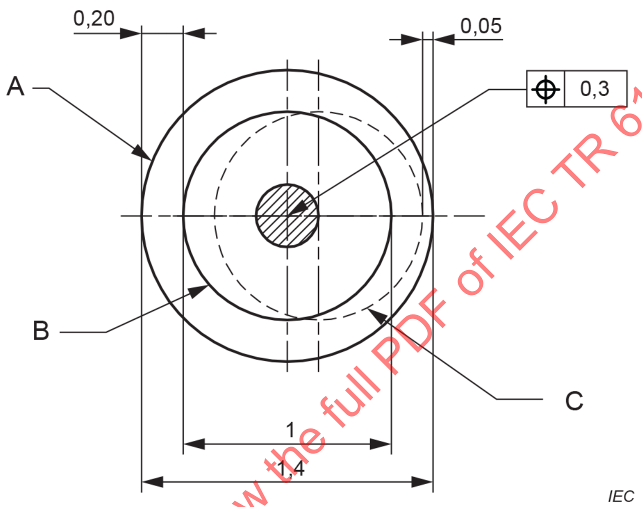
6.2.3 Terminal position tolerance of 0,30 mm

Example 2 is as follows:

- through hole diameter: 1,40 mm;
- terminal diameter: 1,0 mm;
- position tolerance: 0,30 mm;
- design gap on each side: 0,20 mm.

A minimum clearance of 0,05 mm between the terminal and the finished hole wall can be achieved, even if the actual displacement of the terminal is up to the maximum terminal position tolerance of 0,15 mm (see Figure 9).

Dimensions in millimetres



Key

A circuit board through hole diameter

B terminal position (centre of position tolerance)

C terminal position (maximum displacement to terminal position tolerance)

Figure 9 – Terminal position tolerance 0,30 mm, through hole diameter 1,40 mm (one side clearance 0,20 mm)

6.2.4 Terminal position tolerance of 0,20 mm

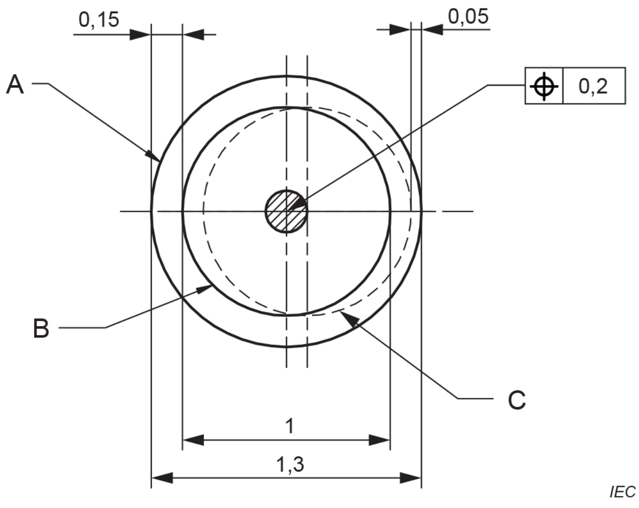
Example 3 is as follows:

- through hole diameter: 1,30 mm;
- terminal diameter: 1,0 mm;
- position tolerance: 0,20 mm;
- design gap on each side: 0,15 mm.

In the case of a terminal position tolerance of 0,20 mm, the trough hole diameter can be reduced to 1,30 mm.

A minimum clearance of 0,05 mm between the terminal and the finished hole wall can be achieved, even if the actual displacement of the terminal is up to the maximum terminal position tolerance of 0,10 mm (see Figure 10).

Dimensions in millimetres



key

A circuit board through hole diameter

B terminal position (centre of position tolerance)

C terminal position (maximum displacement to terminal position tolerance)

Figure 10 – Terminal position tolerance 0,20 mm, through hole diameter 1,30 mm (one side clearance 0,15 mm)

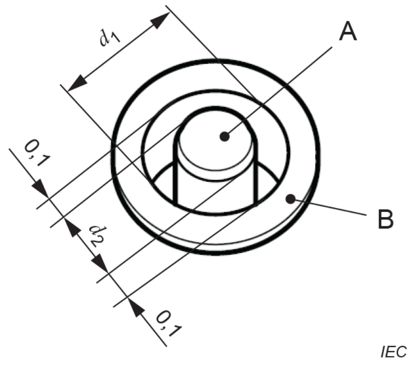
6.3 Circuit board though hole diameter design guidelines for THR components

6.3.1 Circuit board though hole diameter design

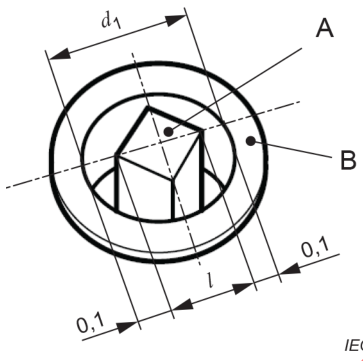
The minimum through hole diameter of a circuit board, to insert a THR component, is determined by considering the diameter of component terminals, the terminal position tolerance, and the position accuracy of the pick-and-place machine.

According to IEC 61760-3:2021, 4.6.5.2, a minimum clearance between terminal and finished hole size of 0,2 mm (0,10 mm on one side) is required for the diagonal line or diameter of the terminal (see Figure 11). A minimum clearance between terminal and finished hole size of 0,3 mm (0,15 mm on one side) can be set, when the mounting accuracy of the mounting equipment is taken into consideration.

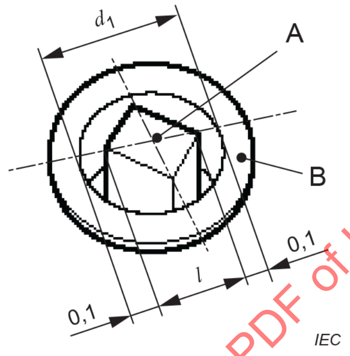
Dimensions in millimetres



a) **Round terminal**



b) **Square terminal**



c) **Rectangular terminal**

Key

A	component terminal
B	land
d_1	through hole diameter
d_2	component terminal diameter
l	component terminal diagonal line dimension

Figure 11 – Circuit board through hole diameter and component terminal diameter/diagonal line dimension

6.3.2 Relationship through hole diameter and solder paste supply

As the through hole diameter increases, the amount of solder paste to be supplied increases (i.e., the printing area of solder paste increases), and the restriction on the arrangement of components in the vicinity of the THR component becomes more sever. There is a limitation on the travel distance to the land during solder paste melting. Therefore, it is necessary to consider the aperture geometry of the stencil and the through hole diameter of the circuit board.

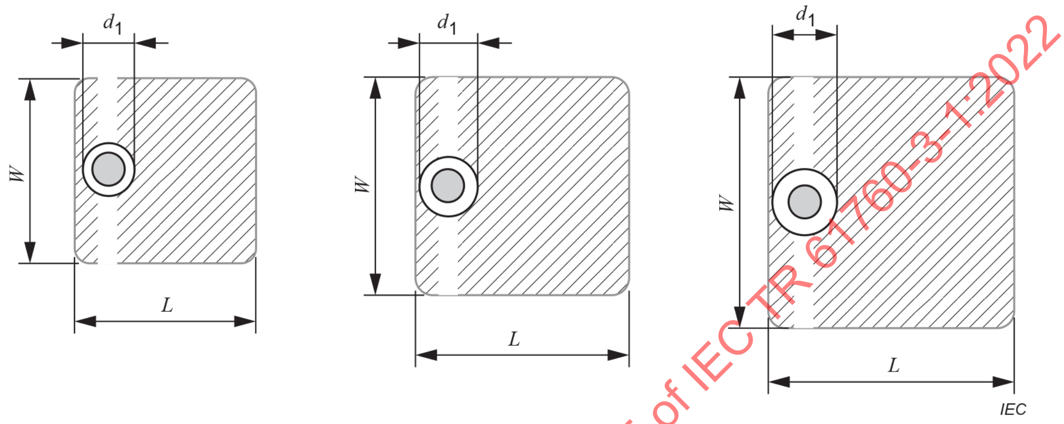
Comparison of the amount of solder paste required by through hole diameter are shown in Figure 12 through Figure 14. These comparisons are made under following conditions:

– Component terminal dimensions, 0,5 mm diameter, 0,5 mm square, or 0,5 mm × 0,25 mm;
– Circuit board thickness, 1,6 mm;
– Stencil thickness, 0,12 mm;
– Stencil aperture gap, 0,5 mm.

NOTE Diagonal line dimensions are 0,7 mm for 0,5 mm square terminal and 0,56 mm for 0,5 mm × 0,25 mm rectangular terminal.

In the case of round terminal with 0,5 mm diameter, the through hole diameter design values are considering only the positional tolerances of 0,2 mm, 0,3 mm and 0,4 mm, i.e., when the through hole diameter is enlarged from 0,8 mm to 0,9 mm and 1,0 mm. When the terminal position tolerance is 0,3 mm and 0,4 mm, the printed area of the solder paste would need to be about 1,39 and about 1,84 times, respectively, compare to the terminal position tolerance of 0,2 mm (see Figure 12).

Dimensions in millimetres



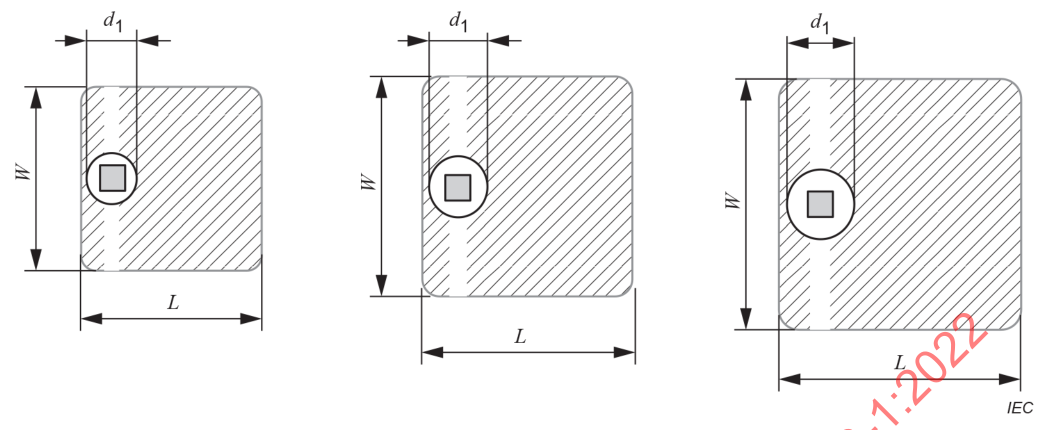
Hatched areas represent solder paste printed areas.

Parameters	Terminal position tolerance		
	0,2	**0,3**	**0,4**
d_1	0,8	0,9	1,0
$L \times W$	2,8 × 2,8	3,3 × 3,3	3,8 × 3,8
Solder paste printing area ratio	1,00	1,39	1,84

Figure 12 – Through hole diameter and solder paste supply amount (printing area) for round terminal

In the case of square terminal with 0,5 mm × 0,5 mm, the through hole design diameter values are considering only the positional tolerances of 0,2 mm, 0,3 mm and 0,4 mm, i.e., when the through hole diameter is enlarged from 1,0 mm to 1,1 mm and 1,2 mm. When the terminal position tolerance is 0,3 mm and 0,4 mm, the printed area of the solder paste would need to be about 1,30 and about 1,63 times, respectively, compare to the terminal position tolerance of 0,2 mm (see Figure 13).

Dimensions in millimetres



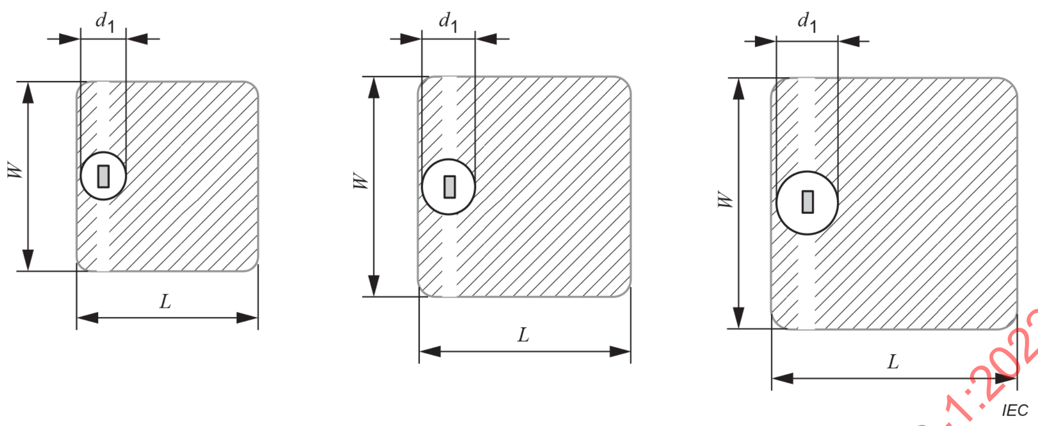
Hatched areas represent solder paste printed areas.

Parameters	Terminal position tolerance		
	0,2	**0,3**	**0,4**
d_1	1,0	1,1	1,2
$L \times W$	3,6 × 3,6	4,1 × 4,1	4,6 × 4,6
Solder paste printing area ratio	1,00	1,30	1,63

Figure 13 – Through hole diameter and solder paste supply amount (printing area) for square terminal

In the case of rectangular terminal with 0,25 mm x 0,5 mm, the through hole design diameter values are considering only the positional tolerances of 0,2 mm, 0,3 mm and 0,4 mm, i.e., when the through hole diameter is enlarged from 1,0 mm to 1,1 mm and 1,2 mm. When the terminal position tolerance is 0,3 mm and 0,4 mm, the printed area of the solder paste would need to be about 1,21 and about 1,50 times, respectively, compare to the terminal position tolerance of 0,2 mm (see Figure 14).

Dimensions in millimetres



Hatched areas represent solder paste printed areas.

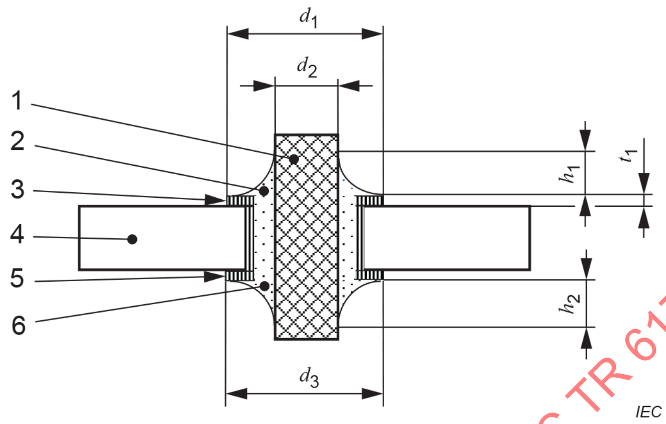
Parameters	Terminal position tolerance		
	0,2	**0,3**	**0,4**
d_1	1,0	1,1	1,2
$L \times W$	4,0 × 4,0	4,4 × 4,4	4,9 × 4,9
Solder paste printing area ratio	1,00	1,21	1,50

Figure 14 – Through hole diameter and solder paste supply amount (printing area) for rectangular terminal

Annex A
(informative)

Calculation method for solder fillet volume of the round terminal

The schematic cross-sectional diagram of the solder joint region is shown in Figure A.1.



Key

1	component terminal
2	solder fillet (top side)
3	land (top side)
4	circuit board
5	land (bottom side)
6	solder fillet (bottom side)
d_1	circuit board land diameter (top side)
d_2	component terminal diameter
d_3	circuit board land diameter (bottom side)
t_1	circuit board land thickness
h_1	solder fillet height (top side)
h_2	solder fillet height (bottom side)

NOTE The shape of the meniscus in this figure is idealized/simplified to enable a simple calculation.

Figure A.1 – Solder joint region cross section

Assuming the solder fillet shape with component terminal can be approximated by the following circular truncated cone (see Figure A.2):

– the base is the same round as the land diameter;
– the top is the same round as the component diameter;
– the height is same as solder fillet height.

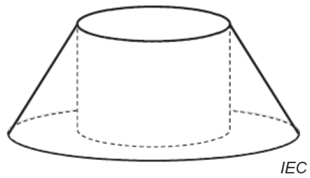


Figure A.2 – Solder fillet approximated by a truncated cone

$$V_1 = \frac{1}{3}\pi h_1^{\,2}(d_1^{\,2} + d_1 d_2 + d_2^{\,2})$$

$$V_2 = \frac{1}{3}\pi h_2^{\,2}(d_3^{\,2} + d_2 d_3 + d_2^{\,2})$$

where

V_1 is the circular truncated cone volume, top side (mm^3);

V_2 is the circular truncated cone volume, bottom side (mm^3);

d_1 circuit board land diameter, top side (mm);

d_2 component terminal diameter (mm);

d_3 circuit board land diameter, bottom side (mm);

h_1 solder fillet height, top side (mm);

h_2 solder fillet height, bottom side (mm).

The volume of the solder fillet can be calculated by subtracting the volume of the above circular truncated cone by the volume of the inside component terminal.

$$V_3 = V_1 - \pi\left(\frac{d_2}{2}\right)^2 h_1$$

$$V_4 = V_2 - \pi\left(\frac{d_2}{2}\right)^2 h_2$$

Thus, the volume of the solder fillets both top side (V_1) and bottom side (V_2) can be obtained using Formula (A.1) and Formula (A.2) respectively.

$$V_3 = \frac{1}{12}\pi h_1\left(d_1^{\,2} + d_1 d_2 - 2d_2^{\,2}\right) \tag{A.1}$$