

Printed wiring boards —

Part 3: Guide for the design and use of printed wiring boards

Committees responsible for this British Standard

The preparation of this British Standard was entrusted by the Electronic Components Standards Policy Committee (ECL/-) to Technical Committee ECL/19, upon which the following bodies were represented:

- Association of the Electronics Telecommunications and Business Equipment Industries (EEA)
- British Telecommunications plc
- Electrical and Electronic Insulation Association (BEAMA Ltd.)
- Electronic Components Industry Federation
- ERA Technology Ltd.
- GAMBICA (BEAMA Ltd.)
- Institute of Metal Finishing
- Ministry of Defence
- National Supervising Inspectorate
- Printed Circuit Interconnection Federation

This British Standard, having been prepared under the direction of the Electronic Components Standards Policy Committee, was published under the authority of the Standards Board and comes into effect on 30 August 1991

© BSI 06-1999

First published April 1984
Second edition August 1991

The following BSI references relate to the work on this standard:
Committee reference ECL/19
Draft for comment 89/33879 DC

ISBN 580 19939 8

Amendments issued since publication

Amd. No.	Date	Comments

Contents

	Page
Committees responsible	Inside front cover
National foreword	iii
Introduction	1
1 Scope	1
2 Normative references	1
3 Materials and surface finishes	2
3.1 Materials	2
3.2 Metallic surface finishes	5
3.3 Non-metallic surface finishes	6
4 Assembly	11
5 Dimensions	11
5.1 Datum reference	11
5.2 Outline dimensions of printed boards	11
5.3 Board thickness	11
5.4 Dimensions of holes	13
5.5 Dimensions of slots and notches	13
5.6 Dimensions of conductors	13
5.7 Dimensional stability	17
6 Electrical characteristics	17
6.1 Resistance	17
6.2 Current-carrying capacity	19
6.3 Insulation resistance	23
6.4 Voltage proof	24
6.5 Other electrical characteristics	26
7 Mechanical characteristics	26
7.1 Adhesion of the conductive pattern	26
7.2 Flatness	27
8 Miscellaneous	28
8.1 Soldering	28
8.2 Delamination	29
8.3 Flammability	29
9 Packaging of printed boards	32
9.1 General	32
9.2 Packaging materials	32
9.3 Procedures	33
Annex A (informative) Determination of size of access window in permanent protective coating	38
Figure 1 — Example for contact zone and inspection mask	34
Figure 2 — Methods of reinforcing lands	35
Figure 3 — Forms of access windows	35
Figure 4 — Access window in permanent protective coating	36
Figure 5 — Datum reference	36
Figure 6 — Examples of recommended and non-recommended conductive patterns	37
Figure 7 — Examples of thermal barrows	37

	Page
Figure A.1 — Smallest access window for the coating as determined by the requirement for the minimum effective solder area (or surface)	38
Figure A.2 — Largest access window for the coating as determined from conductor coverage requirements	38
Figure A.3 — Nominal access window as determined by design	38
Figure A.4 — Example of actual access window as resulting from process tolerances and design	38
Table 1 — A first guide for choosing materials for printed boards	2
Table 2 — Nominal board thickness	12
Table 3 — Nominal hole diameter	13
Table 4 — Tolerances	14
Table 5 — Positional tolerances	15
Table 6 — Minimum peel strength	26
Publication(s) referred to	Inside back cover

National foreword

This Part of BS 6221 has been prepared under the direction of the Electronic Components Standards Policy Committee. It is identical with IEC Publication 326-3:1991, second edition, “*Printed boards — Part 3: Design and use of printed boards*”, published by the International Electrotechnical Commission (IEC).

This Part of BS 6221 is a revision of BS 6221-3:1984 which is withdrawn.

Cross-references

International standard	Corresponding British Standard
IEC 97:1970	BS 5830:1979 Specification for grid system for printed circuits (Identical)
	BS 4584 Metal-clad base materials for printed wiring boards
IEC 249-1:1982	Part 1:1983 Methods of test (Identical)
IEC 249-2-1:1985	Section 102.1:1990 Specification for phenolic cellulose paper copper-clad laminated sheet, high electrical quality (Identical)

The Technical Committee has reviewed the provisions of IEC 65, IEC 171, IEC 194, IEC 216, IEC 249-2, IEC 321, IEC 326-2, IEC 512-2 and IEC 695-1-1, to which reference is made in the text, and has decided that they are acceptable for use in conjunction with this standard.

Related British Standards are as follows.

International standard	Related British Standard
IEC 65:1985	BS 415:1990 Specification for safety requirements for mains-operated electronic and related apparatus for household and similar general use
IEC 512-2:1985	BS 5772 Basic testing procedures and measuring methods for electro-mechanical components for electronic equipment Part 2: Supplement 1:1981 Test 4c

IEC 326-2:1990 replaces IEC 326-2:1976. BS 6221 “*Printed wiring boards*” Part 2:1991, “Methods of test” is identical with IEC 326-2:1990 incorporating amendments Nos. 1 and 2.

NOTE *Typographical errors.* In Figure 6, “Examples of recommended and non-recommended conductive patterns” in the first side text, the word “acoided” should read “avoided” and, in the second side text, the word “Larte” should read “Large”.

A British Standard does not purport to include all the necessary provisions of a contract. Users of British Standards are responsible for their correct application.

Compliance with a British Standard does not of itself confer immunity from legal obligations.

Summary of pages

This document comprises a front cover, an inside front cover, pages i to iv, pages 1 to 40, an inside back cover and a back cover.

This standard has been updated (see copyright date) and may have had amendments incorporated. This will be indicated in the amendment table on the inside front cover.

Introduction

IEC 326 is applicable to printed boards, irrespective of their method of manufacture, when they are ready for the mounting of components. It is divided into separate parts covering information for the designer, recommendations for the specification writer, test methods and requirements for the various types of printed boards, for example rigid single-sided, double-sided and multilayer printed boards, flexible single-sided, double-sided and multilayer printed boards and flex-rigid double-sided and multilayer printed boards.

This part of IEC 326 contains fundamental information on the design and the application of printed boards, their technical limitations and the features and characteristics which can normally be obtained and expected.

It is intended to be used for guidance to the designer and the user of printed boards.

Types of printed boards covered by this part of IEC 326 include:

- a) rigid or flexible single-sided printed boards;
- b) rigid, flexible or flex-rigid double-sided printed boards with or without interconnection through the board;
- c) rigid, flexible or flex-rigid multilayer printed boards with interconnections between layers made by plated-through holes.

1 Scope

This part of IEC 326 relates to the design and the application of printed boards, irrespective of their method of manufacture.

This part of IEC 326 gives recommendations to the designer and the user of printed boards on matters relating to the specification, design and application of printed boards.

2 Normative references

The following standards contain provisions which, through reference in this text, constitute provisions of this part of IEC 326. At the time of publication, the editions indicated were valid. All standards are subject to revision, and parties to agreements based on this part of IEC 326 are encouraged to investigate the possibility of applying the most recent editions of the standards indicated below. Members of IEC and ISO maintain registers of currently valid International Standards.

IEC 65:1985, *Safety requirements for mains operated electronic and related apparatus for household and similar general use.*

IEC 97:1970, *Grid system for printed circuits.*

IEC 171:1964, *Fundamental parameters of connectors for printed wiring boards.*

IEC 194:1988, *Terms and definitions for printed boards.*

IEC 216, *Guide for the determination of thermal endurance properties of electrical insulating materials.*

IEC 249-1:1982, *Base materials for printed circuits — Part 1: Test methods.*

IEC 249-2, *Base materials for printed circuits — Part 2: Specifications.*

IEC 249-2-1:1985, *Base materials for printed circuits — Part 2: Specifications. Specification No. 1: Phenolic cellulose paper copper-clad laminated sheet, high electrical quality.*

IEC 321:1970, *Guidance for the design and use of components intended for mounting on boards with printed wiring and printed circuits.*

IEC 326-2:1976, *Printed boards — Part 2: Test methods.*

IEC 512-2:1985, *Electromechanical components for electronic equipment; basic testing procedures and measuring methods — Part 2: General examination, electrical continuity and contact resistance tests, insulation tests and voltage stress tests.*

IEC 695-1-1:1982, *Fire hazard testing — Part 1: Guidance for the preparation of requirements and test specifications for assessing fire hazard electrotechnical products — General guidance.*

3 Materials and surface finishes

3.1 Materials

3.1.1 General

The engineer concerned with the design of a printed board should choose suitable materials taking into consideration:

- process to be used (subtractive, additive, semi-additive);
- type of board (single-sided, double-sided, multilayer, rigid, flexible or flex-rigid);
- electrical properties;
- mechanical properties;
- special properties, for example flammability and burning characteristics, machinability, flexibility, etc.

The process to be used determines whether a metal-clad base material (subtractive process) or an unclad base material (additive or semi-additive process) should be used.

Consequently, the material for printed boards will be:

- copper-clad* synthetic resin bonded sheet or *copper-clad* polymeric film, where the conductive patterns are obtained by selective removal of the unwanted portions of the conductive foil, or
- unclad* synthetic resin bonded sheet or polymeric film, where the conductive patterns are obtained by selective deposition of conductive material on unclad base material.

In Table 1, qualitative criteria are given for choosing materials for printed boards. The table does not fully cover all materials, but only those in general use.

Table 1 — A first guide for choosing materials for printed boards

	Rigid printed boards				Flexible printed boards		
	Phenolic resin bonded paper	Epoxide resin bonded paper	Polyester resin bonded glass mat	Epoxide resin bonded glass fabric	Polyester film	Polyimide film	Fluorinated ethylene propylene film (FEP)
Mechanical properties	0	0/+	+	++	NA	NA	NA
Electrical properties	0/+	+	+++	++	+++	++	?
Resistance to high temperature during use	+	0/+	+	++	0/++	+++	?
Resistance to high humidity during use	0	0	+	+	+	+	++
Resistance to soldering + temperature	+	+	+	++	—	0/+	0
Where: ? means that insufficient data are at the moment available to fill this place in the table; — may cause problems under certain conditions; 0 = fair, will usually cause no problems for most applications; +, ++, +++ = good, very good, excellent; NA = non applicable.							

Preferably, the materials standardized in the IEC should be used. IEC 249-2 contains specifications for rigid and flexible copper-clad base materials and for bonding sheet materials for use in the manufacture of multilayer printed boards.

If there is no recognized specification available for the required material, a suitable specification sheet detailing its properties should be prepared.

This should be done preferably

- a) using the methods of test in IEC 249-1;
- b) following the layout and format of IEC 249-2;
- c) in conjunction with the material supplier.

Where special properties are essential, they should be defined and specified in conjunction with the material supplier.

3.1.2 General description of materials for printed boards

Where maximum operating temperatures are quoted in the following descriptions, they are intended only for guidance, and do not imply that abrupt changes in performance or in rates of ageing will occur if they are exceeded.

Furthermore, it should be noted that certain material properties may be influenced by factors, such as the design of the printed board (for example board thickness, amount and distribution of metal, number of layers, solder resist, etc.) and production processes (for example laminating process of multilayer printed boards) to such an extent that the readily processed printed boards exhibit properties which differ considerably from the original material properties.

For a precise definition of the thermal characteristics of materials, refer to IEC 216.

3.1.2.1 Copper-clad base materials for rigid printed boards

Phenolic resin bonded paper

This material is manufactured in various grades. Most grades are suitable for use at temperatures up to approximately 70 °C to 105 °C, depending on the grade and the thickness although long-term operation at temperatures towards the higher end of this range may cause a degradation of some properties. Excessive heating, however, can result in carbonization and, in the areas affected, the insulation resistance may fall to a very low level; sources of such heat are, for example, hot resistors.

Within the normal temperature range, a severe darkening can occur and this might not be due to carbonization. Sunlight also darkens these materials. In these instances no loss of properties is involved.

Exposure to high humidity significantly reduces the insulation resistance of these materials although this condition reverses when the humidity falls to a lower value.

Epoxide resin bonded paper

This material has certain improved properties, both electrical and non-electrical, compared with phenolic resin bonded paper, including better machinability and mechanical properties. It is suitable for use at temperatures up to approximately 90 °C to 110 °C, depending on the thickness.

Polyester resin bonded glass mat

Most mechanical properties of this material are lower than those of materials based on glass fabrics but generally higher than those of paper-based materials.

It is, however, highly resistant to impact. Its electrical properties are good over a wide frequency range and these are maintained when it is exposed to high humidity. Its resistance to tracking and arcing is dependent on the grade selected. Most grades are suitable for use at temperatures up to approximately 100 °C to 105 °C.

Epoxide resin bonded glass fabric

The mechanical properties of this material are better than paper-based materials particularly with regard to flexural strength, resistance to impact, dimensional stability in all three major axes, flatness and greater resistance to the thermal shock caused by soldering. The electrical properties of this material are also good. Most grades can be used up to temperatures of approximately 130 °C and are less affected by adverse environmental conditions (humidity).

3.1.2.2 Copper-clad base materials for flexible printed boards

Certain properties may be modified significantly by the adhesive system used.

When printed boards contain both flexible and rigid sections in the same board, materials used for rigid printed boards (3.1.2.1), flexible printed boards (3.1.2.2) and multilayer printed boards (3.1.2.3), may be combined in a single construction.

Polyester film

The characteristic for which this material is generally used is its flexibility. Its useful feature is that it can be heat set to form retractable coils. Provided that suitable adhesives are employed, this material can be used at temperatures up to approximately 80 °C to 130 °C, depending on the grade. When soldering, considerable care is needed as the film tends to soften and distort at soldering temperatures.

It has excellent electrical properties and these are well maintained when the material is exposed to high humidity.

Polyimide film

This material has good flexibility and can be safely soldered provided absorbed moisture is first removed by pre-heating. Normal adhesive bonded types can be used at up to approximately 150 °C continuous operating temperature, but a special fusion-bonded type, using an intermediate film of fluorinated ethylene propylene (FEP), can be used at temperatures up to approximately 250 °C.

A special purpose no adhesive version is available for higher temperatures. The electrical properties of polyimide are excellent but may be affected by moisture absorption.

Fluorinated ethylene propylene film (FEP)

This film is usually combined with polyimide or glass fibre to form laminates with good flexibility and stability at soldering temperatures not exceeding 250 °C, but it may also be used unsupported. The material is a thermoplastic with melting temperature of about 290 °C. It has excellent resistance to moisture, acids, alkalis and organic solvents. Its main disadvantage is that at laminating temperatures, during processing, the conductors are liable to move.

3.1.2.3 Adhesives for flexible printed boards

Adhesives specified for bonding of coverlayers and layers of flexible multilayer printed boards may be thermosetting or thermoplastic. The adhesive selected should be compatible with the material to be bonded and with the performance requirements of the flexible printed board.

The choice of a suitable adhesive depends on factors such as type of flexible printed board, through connection requirements, bending requirements (static/dynamic), operating temperature, humidity, cost, etc.

3.1.2.4 Coverlayer materials for flexible printed boards

Coverlayers are used in flexible printed boards to capture surface conductors and to improve and/or maintain electrical properties of the flexible printed board.

The coverlayer and adhesive system is usually the same as used in the base material. The coverlayer selected should be compatible with the materials used and with the performance requirements of the flexible printed board.

For selecting coverlayer materials see **3.3.3**.

3.1.2.5 Materials for multilayer printed boards

Multilayer printed boards consist of alternate layers of conductive patterns and insulating material with conductive patterns in more than two layers. They are built up with individual thin printed boards (single-sided or double-sided) bonded together with insulating bonding sheets. These bonding sheets consist of sheet material, for example glass fabric, impregnated with a semi-cured resin which will be cured to the final stage when the multilayer printed board is being laminated.

Copper-clad epoxide resin bonded glass fabric

The copper-clad base materials used for the individual thin printed boards are basically the same as those used for single and double-sided printed boards. Usually they are thinner than the materials used for single and double-sided printed boards and their thickness is standardized in ranges instead of a few fixed values. They have the same basic properties as the relevant materials described above.

Epoxide resin impregnated glass fabric bonding sheets

The bonding sheets consist of sheet material (for example glass fabric impregnated with a semi-cured resin) which will be cured to the final stage when the multilayer printed board is being laminated. They show their final properties, therefore, only after lamination. It should be noted, however, that the production processes and the design of the multilayer printed board may influence the material properties considerably.

3.1.2.6 *Special materials and new materials*

Besides the materials described here there are special materials and new materials on the market which are not or have not yet been standardized.

NOTE An example of a special material is the silicone resin bonded glass fabric which is suitable for temperature up to approximately 180 °C.

Since development is in progress, no general description of special materials and new materials can be given here. For the use of these materials, consultation with the supplier of the material is necessary.

3.1.3 *Some particular properties*

3.1.3.1 *Machinability*

The material standards do not contain details for the machinability. They state only that the laminates shall be capable of being punched, sheared or drilled without delamination, in accordance with the manufacturer's recommendations. The machinability of the various materials may, however, be different. Some of the materials are even available in different grades of machinability. For example, certain materials may be punched at room temperature while others can only be punched at elevated temperatures. It is, therefore, necessary to observe the supplier's recommendations.

3.1.3.2 *Flammability*

Some of the materials are available, with defined flammability. There are different grades of flammability. Their details are given in the relevant specification, for example in IEC 249-2.

It should duly be noted, however, that the flammability characteristics of the base material are given for guidance only and may differ considerably from those of the fully processed printed board. The design of the printed board (for example, board dimensions, amount and distribution of metal, number of layers, etc.) has a big influence on the flammability characteristic. Normally, the printed board will be better than the base materials alone, i.e. the fire risk will be lower. For further details, see 8.3.

3.2 *Metallic surface finishes*

Metallic finishes are used to protect the metal (copper) surface, to provide solderability and in some processes as a resist against etchant fluids (as in plated-through hole production).

They can also be used as a contact surface for connectors or as a bonding layer for surface mounted devices.

3.2.1 *Materials*

A suitable finish for the conductive pattern should be chosen, depending on the application of the printed board. The type of the surface finish may influence the production process, the production costs and the properties of the printed board, for example shelf life, solderability, contact properties.

Examples of widely used surface finishes are given below:

a) *Copper (without additional plating)*

It is used for all kinds of printed boards without finish requirements. Usually a temporary protective coating is applied. Recommended values for the thickness of copper plating in plated-through holes are given in 5.4.2.

b) *Tin*

It is used to preserve solderability. Thicknesses of 5 µm to 15 µm are normally applied.

c) *Tin-lead (electroplated or solder)*

It is used to preserve solderability. The thickness obtained depends on the process applied. When electroplated, the thickness of the tin-lead as deposited lies usually between 5 µm and 25 µm. Where electroplated tin-lead is fused or where tin-lead is applied by solder bath or hot roller coating, the thickness may locally be below 1 µm. These areas are mainly located at the transition between land and hole-wall. The solderability at this transition may be inferior to that at other areas.

The eutectic composition of tin-lead with 63 % tin, remainder lead has the lowest melting point. In practice an acceptable range of compositions is 55 % – 75 % tin, remainder lead.

Solderability of tin-lead will gradually be impaired by storage.

Excess tin-lead plating or solder may be removed by hot air or hot oil spray.

It should be noted, however, that the dimensional characteristics (for example flatness) of a printed board may be affected by thermal exposure (such as solder fusing).

d) *Gold*

Usually on a diffusion barrier layer, for example nickel, is often used for switch and edge board contacts. The characteristics necessary for gold as a contact surface, such as thickness, hardness, wear resistance, contact properties, etc., depend on many factors, (see in **3.2.3** the general remarks on printed contacts).

Occasionally gold is deposited on portions of the conductive pattern other than contacts. Care should be taken when the portions are intended to be soldered. Soldering on the gold may cause severe problems for the solder joint as well as for the solder bath due to the gold alloying with the tin-lead.

e) *Other finishes*

For example, palladium, rhodium on nickel and gold on tin-nickel are also used for printed contacts. The general remarks on printed contacts given in **3.2.3** should be observed.

3.2.2 Adhesion, thickness, porosity

Adhesion and thickness of any plating on the conductive pattern may be verified using Test 13a or 13b (adhesion) and Test 13f (thickness) of IEC 326-2. Care should be taken, however, in specifying the porosity Tests 13c, 13d or 13e, since feasibility of application and confidence level of conclusions to be drawn from the test results are very limited.

3.2.3 Printed contacts

Where printed contacts are used, care should be taken to apply a type of plating compatible with the plating on the mating counterpart contact. No general rule can be given since the appropriate plating depends on several factors most of which are interrelated, for example:

- type of plating of the counterpart
- design of counterpart (shape, contact pressure, etc.);
- endurance, number of operations expected;
- electrical requirements (for example contact resistance);
- mechanical requirements (for example insertion/withdrawal forces);
- environmental conditions.

The metal surface of the printed contact should be smooth and free from defects likely to cause reduction in either electrical or mechanical properties. If necessary, this may be verified by visual examination, Test 1 of IEC 326-2. Where only a limited contact zone is important, an inspection mask may be used, an example is given in Figure 1.

3.3 Non-metallic surface finishes

Non-metallic coating materials are used to protect the printed board. Additionally solder resists are used to prevent wetting of conductors except on areas deliberately formed for connection purposes.

3.3.1 General

Improper cleaning may give rise to loss of adhesion when the coated assembly is exposed to high humidity conditions. Loss of adhesion then usually starts manifesting itself by the occurrence of discrete spots or patches visible at the interface of the coating and its substrate, revealing a separation between them (“mealing”).

It is most important that the printed boards are properly cleaned before applying any kind of coatings. The insulation resistance of printed boards with organic or inorganic contamination cannot be improved by a coating.

Coatings, unless properly chosen and applied, may lead to degradation of the printed board with respect to flammability, insulation resistance, electrical characteristics at high frequency, etc.

3.3.2 Temporary protective coating

3.3.2.1 Temporary coating preserving solderability

Coating used to preserve the solderability of the conductive pattern. It is usually applied where the surface of the conductive pattern is not covered by a well solderable metallic surface finish maintaining its good solderability for the necessary period of time. Example: bare copper.

Depending on the material used, the temporary protective coatings may be removed prior to soldering or may serve as flux. Temporary protective coatings which are not removed prior to fluxing are based on a resin that is soluble in the solvent of the flux.

Extensive drying and/or long storage or extensive application of heat, for example as during vapour phase soldering of the printed boards, may “cure” some resin-based coatings to a point where they no longer sufficiently dissolve in the short time between fluxing and soldering and where soldering results are degraded.

The thickness of resin-based coatings is normally lowest at the transition area between hole-wall and land. The solderability of plated-through holes may degrade there more rapidly with time than in the other areas.

For these reasons, the compatibility of the coatings with the intended processing, for example drying, fluxing, soldering or fusing method, has to be carefully considered.

3.3.2.2 Temporary solder resist

Coating usually applied by screen printing prior to the soldering operation and intended to cover defined portions of a printed board to prevent solder wetting of the conductive pattern in that area.

Example: temporary solder resist on circuit areas with noble metal as surface finish.

In addition, such coatings may also be used to protect those areas from being damaged during processing and storage.

Temporary solder resist may be removed by peeling-off or by immersion in appropriate solvents, depending on the resist used.

Care should be taken that the appropriate removal is complete.

3.3.3 Permanent protective coatings

3.3.3.1 General

Coatings to improve or maintain electrical properties of the printed board, for example the insulation resistance and the breakdown voltage between conductors on the surface of the printed board. They usually consist of durable scratch-resistant material and, hence, protect the board surface from being damaged. They are intended to remain permanently on the printed board during normal operation.

Permanent protective coating may improve or maintain electrical properties of the printed board by:

- retarding the ingress of moisture into the base material;
- preventing contamination being deposited between conductors (for example contamination that would attract moisture);
- acting as dielectric between conductors;
- acting as a protection layer in or over plated-through holes (via-holes) which do not require soldering.

3.3.3.2 Permanent solder resist

Coating applied prior to the soldering operation and intended to cover defined portions of a printed board to prevent solder wetting of the conductive pattern in that area.

Unlike the temporary peel-off or wash-off types, this solder resist is not removed after the soldering operation and serves as a permanent protective coating. It should have adequate protective properties in addition to the characteristics necessary when merely used as a solder resist.

Solder resist intended to be used as a permanent protective coating may also be applied to the component side. In this case it has only the function of a permanent protective coating.

Solder resist may be used for one or more of the following reasons:

- a) to prevent wetting of defined areas;
- b) to prevent bridging between adjacent parts of the conductive pattern;
- c) to concentrate the solder upon the parts of the conductive pattern not covered by solder resist to facilitate and to improve soldering;
- d) to decrease solder consumption and contamination of the solder bath;
- e) to protect the printed board during processing;
- f) to improve or maintain electrical properties of the printed board;

g) to act as an insulating barrier between component body and underlying parts of the conductive pattern.

Where a solder resist is applied to a conductive pattern covered with a material that melts during a soldering operation, for example solder, the solder resist may show wrinkling, blistering or flaking after the soldering operation.

Those effects may be reduced by selectively avoiding the combination of the solder resist over solder or by applying, for example thick layers of solder resist (also acting as heat barrier), thinner layers of solder, narrow conductors and cross-hatching of large conductive areas.

If wrinkling, blistering or flaking is not acceptable, alternative solutions should be employed.

There are two basically different types of solder resist used:

- printing, generally screen printing, where the solder resist is printed in the desired pattern on the printed board;
- photo-definable solder resist, where a special wet or dry film is applied to the printed board and the pattern is created by exposure to light (normally UV) and subsequent developing.

Screen printing is normally cheaper but smaller tolerances may be obtained with photo-definable solder resist (see 3.3.3.4).

Misalignment of access window in the solder resist and the land, as well as deviation in diameter of the land and the access window in the solder resist, may result in partial coverage of the land reducing the area available for soldering. Where necessary, appropriate requirements for dimensions and registration should be specified in the relevant specification.

3.3.3.3 Coverlayers

Insulating protective layers placed on the surfaces of a printed board. It is normally a film or insulating foil applied to a flexible printed board using an adhesive. It may also be a layer applied to a rigid printed board using, for example, prepregs and a lamination process.

Coverlayers cover the entire surface of the printed board except for access windows for soldering or contacting purposes.

Coverlayers of flexible printed boards capture surface conductors and improve or maintain electrical and flexing properties of the printed board. They are usually 0,025 mm thick plus an adhesive, and are considered to be dimensionally unstable. This needs to be considered when defining the minimum annular width for adequate solder land area.

On flexible printed boards, lands with unsupported holes may have to be protected against lifting from the base material either by adding anchoring tabs to the lands concerned or by a circumferential overlap with the coverlayer, as shown in Figure 2.

When closely positioned solder locations make use of individual coverlayer access windows impractical (as in connector configurations), then such access windows can take the forms shown in Figure 3. For unsupported holes, anchoring tabs should be added to the copper lands.

It is not advisable to apply coverlayers to flexible printed boards over areas with metal finishes that melt during soldering operations. The coverlayer can show wrinkling and/or blistering after the soldering operation.

NOTE 1 The combination and individual methods are the most costly. The strip method creates a weak spot where the copper and the base material may crack.

NOTE 2 The individual method should be used for flexible printed boards with low density lands.

NOTE 3 Strip or combination methods should be used for flexible printed boards with high density lands.

NOTE 4 The strip method (baring conductors) should always require the addition of conformal coating or encapsulation compounds to provide additional adhesion support for the bare conductors after the assembly of the part to the conductive pattern.

NOTE 5 The combination method (baring conductors) should always require the addition of conformal coating or encapsulation compounds to provide additional adhesion support for the bare conductors after the assembly of the part to the conductive pattern.

3.3.3.4 Tolerances and design of the solder resist pattern or the coverlayer

Product design and end product requirements shall include the necessary allowance for process tolerances on position and dimensions of the access windows in the solder resist or coverlayer.

Normally a certain area that has to be free from solder resist or coverlayer is specified (both in dimensions and in position) as the minimum effective solder area (see Figure 4). When this area contains a hole intended for component mounting, a minimum value for the solderable annular width, as agreed to between user and manufacturer, may be specified instead of or in addition to the tolerances on position and dimensions.

The design width of the access window in the solder resist or coverlayer should be equal to the width of the minimum effective solder area plus a process allowance equal to at least the process tolerance PT1 as agreed with the printed board manufacturer.

In many cases the extent of the coating is left open, depending upon the amount of coverage required by the conductors which are closest to the solder land. When coating of such conductors is specifically required, the design width of the corresponding solder resist or coverlayer area should be equal to the width of the area to be coated plus a process allowance equal to at least the process tolerance PT2 as agreed with the printed board manufacturer.

As a first approximation, PT1 and PT2 can be considered about equal.

Solder resist tolerances

The following guidance relates to glass-epoxide boards which have not been exposed to solder fusing.

For photo-definition type processes, the positional tolerance may vary from 0,1 mm to 0,6 mm depending on the sizes of the exposed product and on the method of registration.

For screen printing type processes, the positional tolerance may vary from 0,4 mm to 1,0 mm.

Coverlayer tolerances

For processes where the pre-punched or pre-drilled coverlayer is laminated, the process tolerance may vary from 0,5 mm to 1,5 mm.

For a practical example of considerations related to solder resist or coverlayer, see Annex A.

3.3.4 Conformal coating

3.3.4.1 General

Conformal coating is an electrically insulating material on a printed board and/or printed board assembly to provide a protective barrier against deleterious effects from environmental conditions. Correctly chosen, and carefully applied, conformal coating will help to protect the assembly from the following hazards:

Humidity, dust and dirt, airborne contaminants (for example smoke, chemical vapours), conductive particles (for example metal chips, filings), accidental short circuit by dropped tools, fasteners, etc., abrasion damage, fingerprints, vibration and shock (to a certain extent), mould growth and decreased flashover voltage at decreased atmospheric pressure.

Conformal coating resins are selected to fulfil the above requirements plus several other minor ones such as transparency (to permit reading component values after coating) and flexibility (to prevent damage to components in temperature cycling).

In some cases, certain varnishes are used as permanent protective coatings. They are applied after the soldering operation and usually to the solder side only.

In addition to their protective properties, they may have other special characteristics. For example, they may be fluorescent to facilitate visual inspection of coverage.

3.3.4.2 Some limitations of conformal coating resins

Because of the essential requirements, certain limitations are inherent in conformal coatings.

These limitations are:

- a) Conformal coating films, being permeable to water vapour and not being formulated with corrosion preventing fillers such as chromates, will not prevent corrosion due to active electrolytic salts on the part being coated or salts trapped on the surface of the part under the coating.
- b) Conformal coating films, being permeable to water, will undergo a reduction in insulation resistance as the thickness of a film increases. This is particularly the case in a fillet of resin around a component (such as an integrated circuit).
- c) Conformal coating resins, being organic and filling the voids between conductors, will cause a marked change in inter-track capacitance ("C").