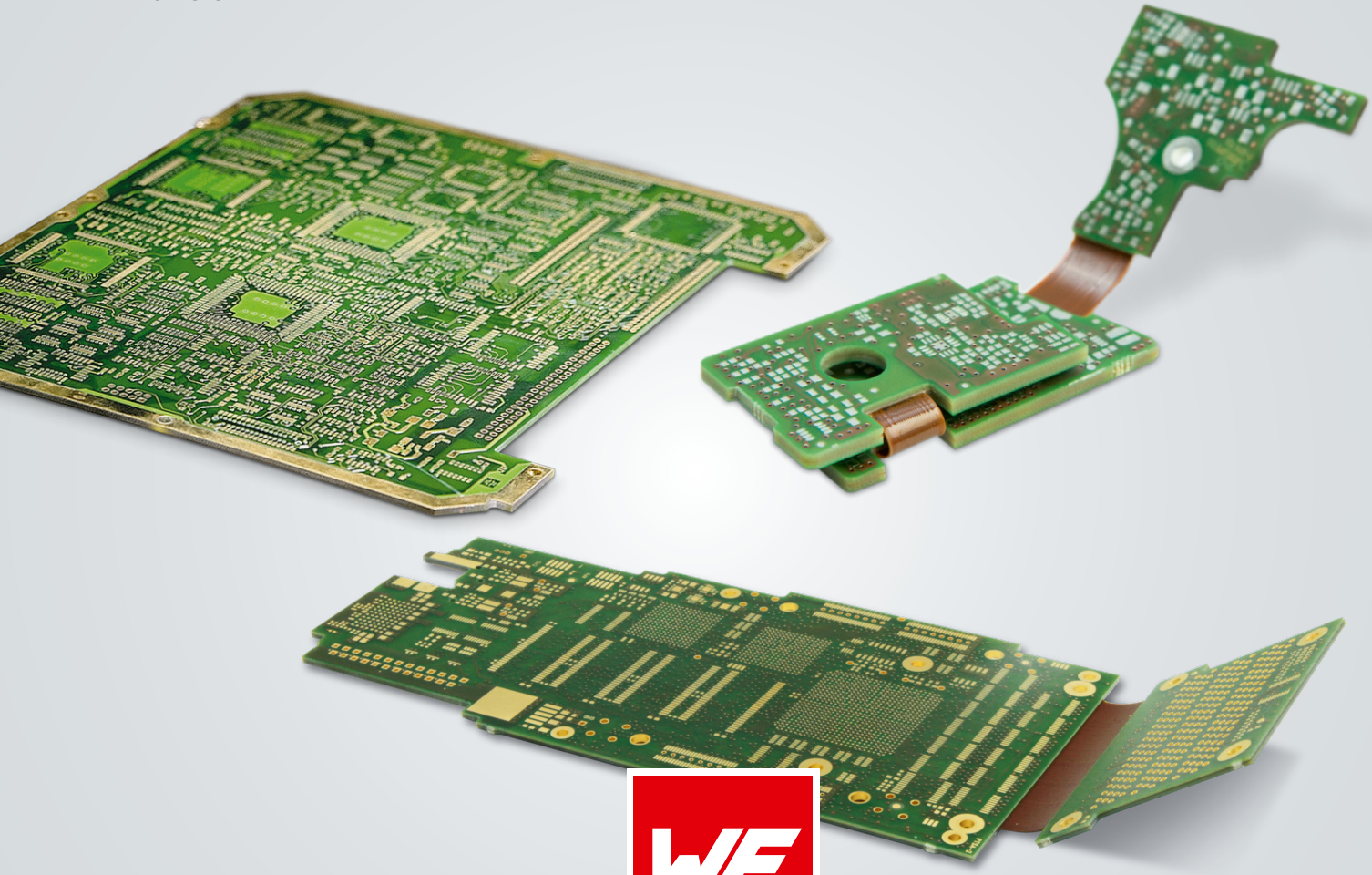


TECHNICAL DELIVERY SPECIFICATION FOR PCBS

WÜRTH ELEKTRONIK MORE THAN YOU EXPECT

INTRODUCTION

Dear customer,

This Technical Delivery Specification aims to make the communication between you and Würth Elektronik Circuit Board Technology more effective and more straightforward. If your PCB specifications are based on this recommendation, we can supply you faster and more reliably. **The data and parameters in the following document do not represent the complete performance portfolio. Please contact us if you have any requirements that go beyond the scope of this document.**

As one of the leading printed circuit board manufacturers in Europe, we supply you with printed circuit boards in all common and many innovative technologies:

- Basic
- Flexible Solutions
- Embedding Technology
- High Current Technology WIRELAID
- Printed Polymer
- HDI Microvia
- Thermal Management
- Wire Bonding
- Stretcheable, Conformable Circuits

A detailed description of the various technologies, design guides, webinars, layer stackups etc. can be found on our website at **www.we-online.com/pcb**

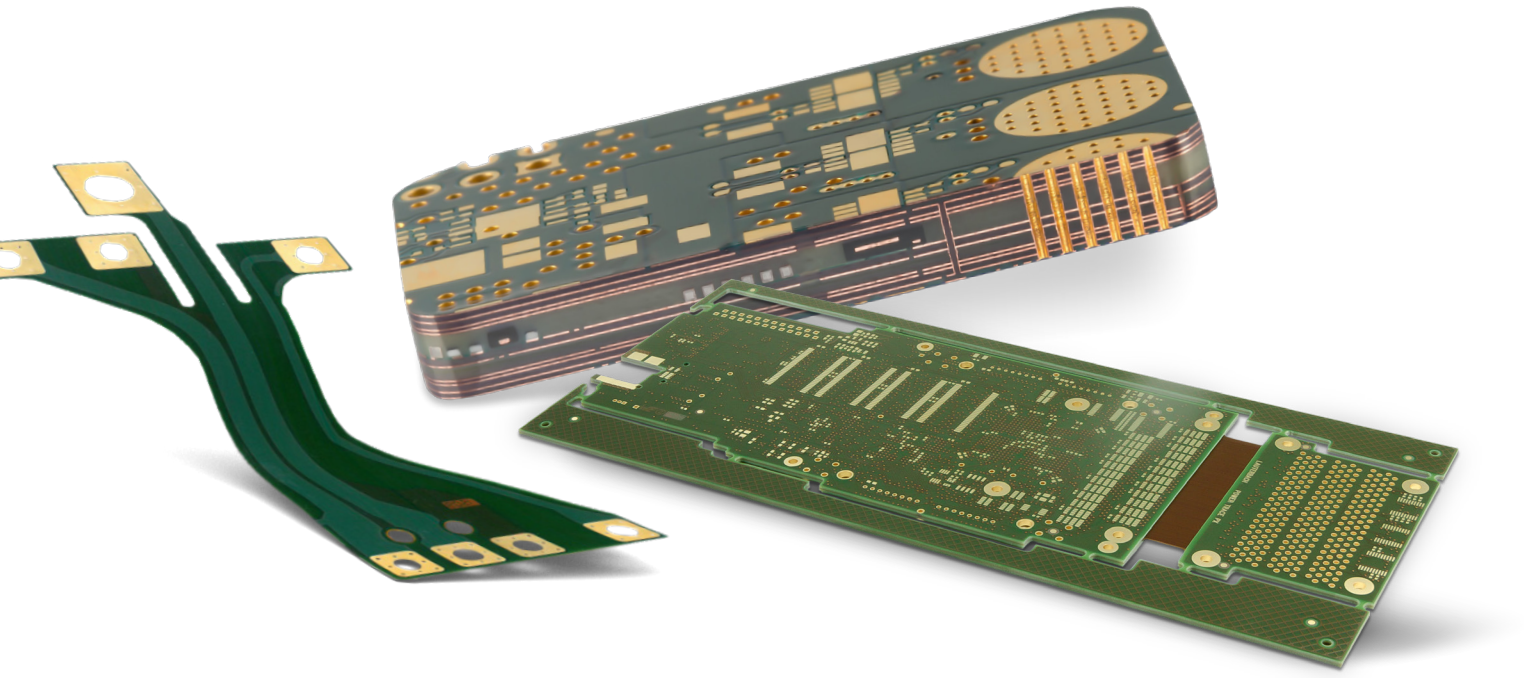
Disclaimer:
All information in this document describes the current status at the time of writing. Errors and omissions are reserved.

CONTENTS

Horizontal line for the metallization of holes with the processes deburring, cleaning and desmear, electroless and electroplating copper deposition

ABBREVIATIONS

CB	Circuit Board
DIN	German Institute for Standardization (**D**eutsches **I**nstitut für **N**ormung)
DM	Plated through hole, center point
EDA	Electronic Design Automation
ENIG	Electroless Nickel Immersion Gold (chem. Ni/Au)
HAL	Hot Air Level (HASL Hot Air Solder Level)
HDI	High Density Interconnect (Microvia technology)
IATF	International Automotive Task Force
IPC	Global association (**www.IPC.org**)
ISA	Industry Standard Architecture (Computerbus Standard)
MBB	Moisture Barrier Bag
NPTH	Non Plated Through Hole
PCI	Peripheral Component Interconnect (Bus-Standard)
PTH	Plated Through Hole
RoHS	Restriction of Hazardous Substances
TDR	Time Domain Reflectometry
Tg	Temperature of glass transition
TDS	Technical Delivery Specification
UL	Underwriters Laboratories Inc.
WE	Würth Elektronik



1 INTRODUCTION

1.1 SCOPE OF APPLICATION

This Technical Delivery Specification (hereinafter referred to as TDS) applies to all types of unassembled circuit boards from Würth Elektronik Circuit Board Technology within the warranty periods. The tolerances are machine and process-related, so they also apply to flexible and rigid-flexible printed circuit boards.
The description of the product characteristics also apply for rigid-flex (partly only in the rigid areas), for flex they are partly not applicable.

For production-oriented development, the application recommendations in the applicable Design Guides and the design requirements of the applicable Design Rules must also be observed.

Therefore, the TDS is not applicable to special products that are manufactured using PCB materials and PCB production techniques but do not follow the Design Guides and Design Rules. Such special products are therefore not printed circuit boards in the true sense.

1.2 PURPOSE

This document describes the qualitative characteristics of the product PCB as well as the performance for processing and application, also the applicable standards. It also regulates the hierarchy of specifications and the procedure in case of deviations or clarifications.

1.3 HIERARCHY OF SPECIFICATIONS

The priority in attention of the different specifications is regulated by the following order of priority:

- Customer circuit board specification
- General customer specification
- IPC-A-600 (see point 1.6)
- WE CBT Technical Delivery Specification (this document).

1.4 DESCRIPTION OF THIS DOCUMENT

The document is structured in three sections:

1. Data exchange
2. Product characteristics
   a. Visible features – exterior, appearance
   b. Nonvisible features – internal (microsection, measuring methods)
   c. Performance for processing and customer application
3. Packing, documentation, after sales

1.5 NORMATIVE REQUIREMENTS FOR PRINTED CIRCUIT BOARDS

IPC (**www.IPC.org**) is a worldwide board and electronics industry association that provides standards for all phases of the product lifecycle (see also IPC Specification Tree).

These standards are used throughout the industry, some examples are listed below:

- IPC-A-600 – Acceptability of Printed Boards
- IPC-TM 650 – Test Methods
- IPC-2221/2222 – Generic Standard on Printed Board Design, IPC-2223 Sectional Design Standard for Flexible Printed Boards, IPC-2226 Sectional Design Standard for High Density Interconnect (HDI) Printed Boards
- IPC-4101 – Specification for Base Materials for Rigid and Multilayer Printed Boards, IPC-4202/03/04 same for flexible and flex-rigid PCBs
- IPC-4562 – Metal Foil for Printed Wiring Applications
- IPC-455x – Specification for solder surface, i.e. Electroless Nickel/Immersion Gold (ENIG) Plating for Printed Boards
- IPC-6012 – Qualification and Performance Specification for Rigid Printed Boards, IPC-6013 – same for Flexible Printed Boards

1.6 STANDARD PERFORMANCE CLASSIFICATION FOR THE PRODUCT PCB

Unless otherwise agreed in writing, WE Circuit Board Technology confirms compliance with the product requirements of IPC-A-600 Class 2, with the exception of the associated standards and guidelines.

1.7 OVERVIEW OF THE TECHNICAL POSSIBILITIES (EXTRACT)

- Maximum circuit board size: usable area 570 mm x 500 mm
- Minimum / maximum PCB thickness (rigid): 0.5 mm / 3.5 mm
- Maximum number of copper layers (standard): 20
- Maximum number of copper layers at 1.6 mm thickness: 10

Design types:

- BASIC (two-layers and multilayer PCBs)
- HDI, Flex & Flex-rigid PCBs
- Thermal management and heatsink PCBs
- Embedding technology

Additional options:

- Impedance manufactured and impedance tested circuit boards
- Wire bonding
- Polymer resistors inside and outside
- Polymer keyboard contacts
- Polymer Potentiometer
- Polymer heating resistors

Please contact us, if you have further requirements!

1.8 PRODUCTION SITE CERTIFICATIONS

We are certified according to:

- DIN EN ISO 50001
- DIN EN ISO 9001
- DIN EN ISO 14001
- ISO / IEC 27001
- IATF 16949
- Würth Elektronik has the label "AEO F, Authorised Economic Operator (Full)".

Circuit boards from Würth Elektronik Circuit Board Technology are type-tested and certified as components with regard to electrical fire and accident hazards by Underwriters Laboratories Inc. (UL):

- Würth Elektronik UL File www.we-online.com/pcbulfileinternational international
- Würth Elektronik UL File www.we-online.com/pcbulfilecanada for Canada

2 FABRICATION DATA

Your fabrication data should be stored in a container format, e.g. ZIP file.

2.1 RECOMMENDED DATA FORMATS

Design:

Preferred	Only by prior agreement
Gerber RS274X	Gerber X2
ODB++	IPC-2581

Application output data:
Target3001; Eagle (must specify the version)

Mechanics:

- Sieb & Meyer or Excellon (drilling and milling)
- Separate drill files for plated and non-plated holes and for each type of partial hole (Microvia, Buried Via)
- Definition of the drill holes, which should be provided with plugging if necessary

We recommend 3.3 metric or 2.4 inch as the ideal number format. For data in 2.3 inch or 3.2 metric format, an offset of up to 50 µm must be taken into account.

The data provided by you describe the dimensions and measurements for the end product. We take care of creating the production data from your layout data. That means:

For the drilling data:

- For the drill holes, you define the final diameter of the holes, we select the suitable tool diameter in increments of 0.05 mm (see chapter 3.7).
- For special cases such as press-fit technology, you define the tool diameter, copper thickness in the PTH and tolerances for the press-fit drill holes or supply the component specifications with the order documents.

For the wiring layout data:
We can widen the conductors, copper and mounting pads to achieve the conductor widths specified by you within the permissible tolerance after the etching process (see Chapters 3.3 and 4.3).

For the solder mask data:
The data for the solder mask should best be generated 1:1 to the solder pads. We adjust the clearances according to our processes so that as far as possible no soldering areas are affected (see chapter 3.5).

For an optional assembly/position printing:
The data for the position printing is trimmed in such a way that free copper areas are not printed if possible. The legibility can be limited by this.

2.2 FILE NAMES

Please use unique designations for your production data.

- Your container file (ZIP file) should contain a unique layer assignment and your article name and revision level.
- The individual file names should contain the layer name (e.g. silkscreen top, solder mask top, l2, l3, bottom, solder mask bottom, silkscreen bottom, etc.).

Beispiel für Lagenbezeichnungen

Example of layer names: "XXX" stands for your article description, "Rev" for revision

XXXRev.TOP (Toplayer)
XXXRev.L2 (Inner layers L2)
XXXRev.L3 (Inner layers L3)
XXXRev.BOT (Bottom layer)
XXXRev.SMT (Solder resist Top layer)
XXXRev.SMB (Solder resist Bottom layer)
XXXRev.LPT (Legend print Top layer)

2.3 INCONSISTENCY OF DATA

If we detect contradictory information in your layout data and dimensional drawings/production plans, we will consult with you for clarification.

TIP:
Please define in your PCB documentation who can be contacted for technical queries regarding the PCB in question. Clarifications can thus be made quickly and directly.

2.4 DELIVERY PANEL DESIGN, REGISTRATION MARKS

We manufacture printed circuit boards

- As single PCB or
- In a delivery panel with routed tabs and/or v-scoring.

If you do not make any specifications in this regard, we will clarify with you in the quotation phase, taking into account the size and capacity utilisation of the production format, whether the printed circuit board will be manufactured individually or in a delivery panel.

If fiducial marks and locating holes for the delivery panel frame are defined in your data, we will insert them accordingly. Fiducial marks will be circumferentially cleared back from the solder resist coating by 1000 µm.
If there are no fiducials or locating holes in your data, we will insert our standard fiducials (3 pcs. each 1.0 mm round; 1 mm circumferentially clear of solder resist) and standard locating holes (3 pcs. each 2.1 mm). We choose the position at our discretion.

EXAMPLES OF DELIVERY PANEL DESIGN WITH V-SCORED AND WITH MIXED MILLED/V-SCORED PCB CONTOUR:

Figure 1: Delivery panel v-scored (left) and mixed milled/v-scored contour (right)

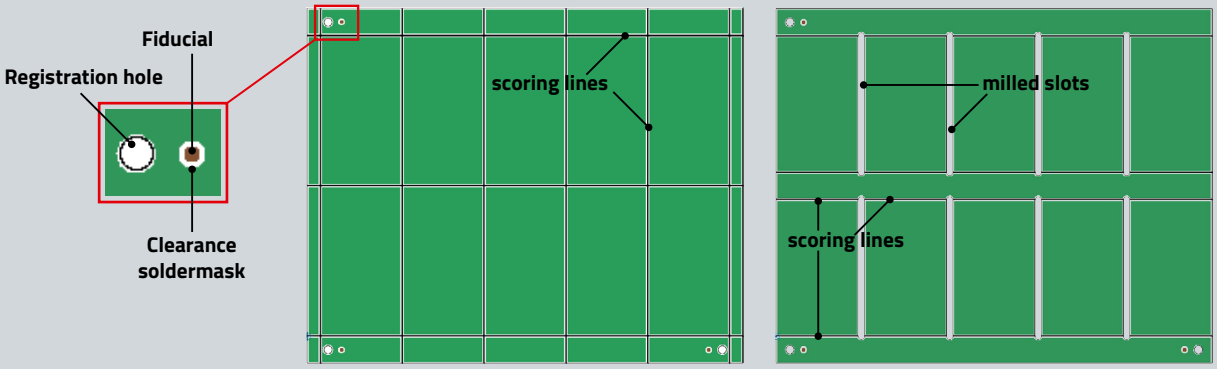
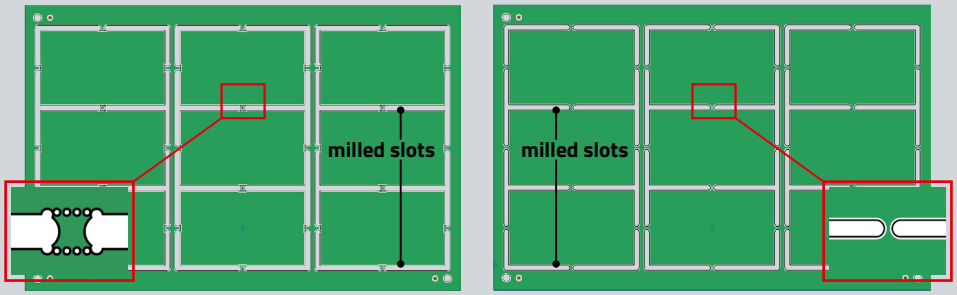


Figure 2: Negative predetermined breaking points on the left, positive retaining webs on the right



There are two different types of connection for the milled contour:

- Negative, inside positioned webs with predetermined breaking points, which can also be broken by hand, see figure 2 left. If you would like us to design the delivery arry for you, please mark the permissible areas in your layout and ensure that there is sufficient copper clearance in these areas.
- Positive protruding webs, which are separated with a separating tool (e.g. "Hector" depaneling tool), see figure 2 right.

DEPANELISATION ADVICE:

Please avoid the introduction of stress during the separation process by using suitable separation processes so that sensitive components and also the printed circuit board are not damaged. For flexible and rigid-flex PCBs, the flex material must be cut through at the webs before breaking in order to avoid delamination in the PCB. Predetermined breaking holes alone are not sufficient for this! Please ask for these options, we will be pleased to provide you with an offer including a delivery array proposal. **Under no circumstances should you use side cutters for cutting off the webs!**

3 EXTERNALLY OBSERVABLE CHARACTERISTICS

Here the IPC-A-600 chapter 2 applies.
This chapter describes "... those characteristics which are observable from the surface. This includes those characteristics that are external and internal in the printed board but visible from the surface".

These are the criteria for assessing the following visible characteristics:

- Edges and surface
- Conductive pattern
- Metallized and non-metallized holes, annular ring on the outer layers
- Identification
- Solder mask, additional prints
- Dimensions and tolerances, position tolerances
- Flatness
- Repair

The IPC-A-600 regulates the aspects not dealt with in detail below.

3.1 SOLDER SURFACES

To ensure solderability, exposed copper surfaces are coated with a soldering surface.

Solder surface	Electro less Nickel Immersion Gold STANDARD	Hot Air Levelling	Hot Air Levelling lead-free	Immersion Tin
	ENIG	HAL (HASL)	HAL (HASL) lead-free	Immersion Sn (iSn)
Standard	IPC-4552	According IPC-6012	According IPC-6012	IPC-4554
Thicknesses	4 – 7 µm Ni; 0,05 – 0,12 µm Au	Covered to 40 µm	Covered to 40 µm	0,8 – 1,2 µm
Solderability	12 months	12 months	12 months	6 months
Properties, remarks	▪ Universally applicable surface ▪ Flat surface ▪ No copper removal through nickel layer ▪ Soldered connection is formed on Ni ▪ Not recommended for press-fit technology! ▪ Suitable for US-bonding with Al wire	▪ Not RoHS compliant (exception: see EU Directive 2011/65/EU) ▪ Not allowed for Microvia	▪ High differences in thickness, only limited suitability for SMT ▪ Increased copper removal, depending on layout ▪ Not allowed for Microvia	▪ Flat surface ▪ Sensitive to physical contact ▪ Suitable for press-fit technology ▪ Layer thickness > 1 µm required for multiple soldering ▪ Rapid processing in the assembly process required

➔ On request, we can also supply your printed circuit boards with other surfaces such as Immersion silver, ENEPIG, OSP, EPIG or DIG. Please provide us with your specification for checking.

➔ **Please contact us for special requirements such as sliding contacts.**

3.2 CONTACT SURFACES

Contact surfaces, for example for direct connectors, are manufactured as follows:

Electro-plated Nickel Gold / Hard gold	ep. Ni/Au	3 – 7 µm Ni; 0,8 – 3 µm Au	**Not solderable!**

For electroplating, these surfaces must be electrically connected to the edge of the printed circuit board or the edge of the delivery panel.

3.3 TOLERANCES OF CONDUCTOR PATTERN FOR OUTER LAYERS

Conductor pattern features on the outer layers consist of base copper plus copper build-up from the "through-hole plating" and "galvanic reinforcement" processes. As standard, conductors and connection surfaces for surface mounting are modified in the same way in order to counteract under-etching during the structuring of the conductive pattern. The acceptable tolerances in the dimensions of pads and conductors are minus 20 percent.

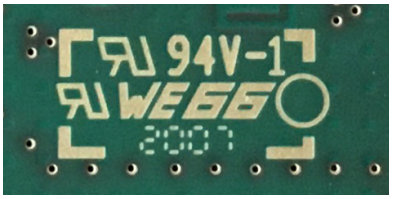
3.4 IDENTIFICATION, LABELLING

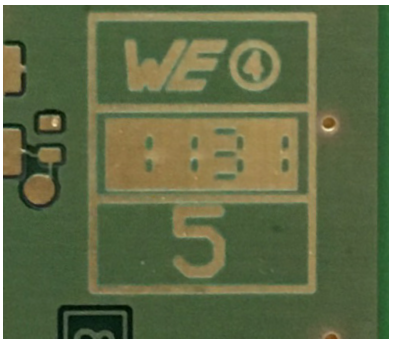
Each printed circuit board must be clearly identifiable with regard to the manufacturer.

- As a standard, your PCB receives a WE logo including date code (yy/ww) for tracking purposes.
- Data Matrix Code according to ISO/IEC 16022 with dynamic contents for individual marking is possible on request. **Learn more here: www.we-online.com/DMCflyer**
- If a UL marking is required, a suitable area must be provided in the data. As standard, the UL marking consists of manufacturer's identification plus UL type designation and factory identification. Optionally, the flammability class and the cURus logo can be included.
- Please note that our UL marking includes the WE logo as standard. This is not advertising. The logo is an official part of the UL marking.
- Alternatively, the Würth Elektronik UL file number "E76251" can be used instead of the WE logo, which requires more space.

Markings are possible in different ways. For all markings, the minimum permitted character heights defined below must be kept.

Examples:





Minimum character height as				
Copper (Size depends on the base copper)		**Clearance in solder mask**		**Legend print Colour white**
Base copper		**Preferred on base material**	**Over copper, NOT HAL!**	**On soldermask**
18 µm	**≥ 1,0 mm**	**≥ 1,0 mm**	**≥ 1,5 mm**	**≥ 1,5 mm**
35 µm	**≥ 1,5 mm**			
70 µm	**≥ 2,0 mm**			
105 µm	**≥ 2,5 mm**			

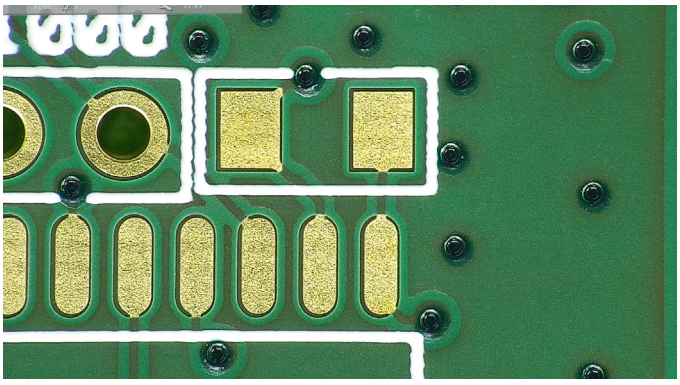
3.5 SOLDERMASK

Our standard for soldermask is green photosensitive soldermask that meets the requirements of IPC-SM-840 Class T and H. Other colors such as white, black, blue, red or yellow can also be applied on request. Depending on the production location, the execution is as a colored solder mask instead of the green solder mask or as an additional print over the standard green solder mask. Please ask for these options, we will provide you with a quote.

Normally we receive solder mask data without oversize, i.e. 1:1 to the circuit image. The necessary modification is carried out by Würth Elektronik. Areas that may not be modified by us must be clearly described.

3.6 PLUGGING OF VIAS IPC-4761, Type III-a

Vias can be plugged on one side with a non-conductive material that partially penetrates the via (approx. 1/3). Please describe in your fabrication data which drill holes and from which side they should be plugged.



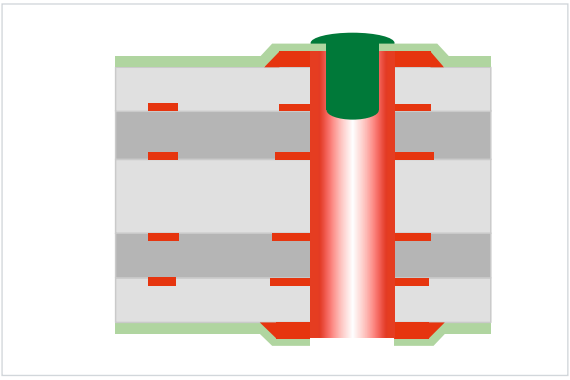
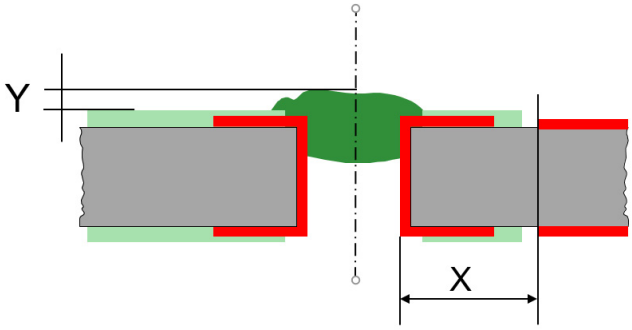


Figure 3: Vias with Plugging (IPC-4761 Type III)

Diagram of the the techincal implementation:

If the minimum distance **X** cannot be kept in your design, check whether the holes can be designed as IPC-4761 Type VII (filled and capped) (see



	Standard	Higher requirements
Minimum distance **X** plugging to solder areas on Top + Bottom! ▪ PTH diameter ≤ 0.3 mm ▪ PTH diameter 0.35 mm to 0.7 mm	 ≥ 0,25 mm ≥ 0,30 mm	
Maximum height of plugging **Y**	70 µm	60 µm

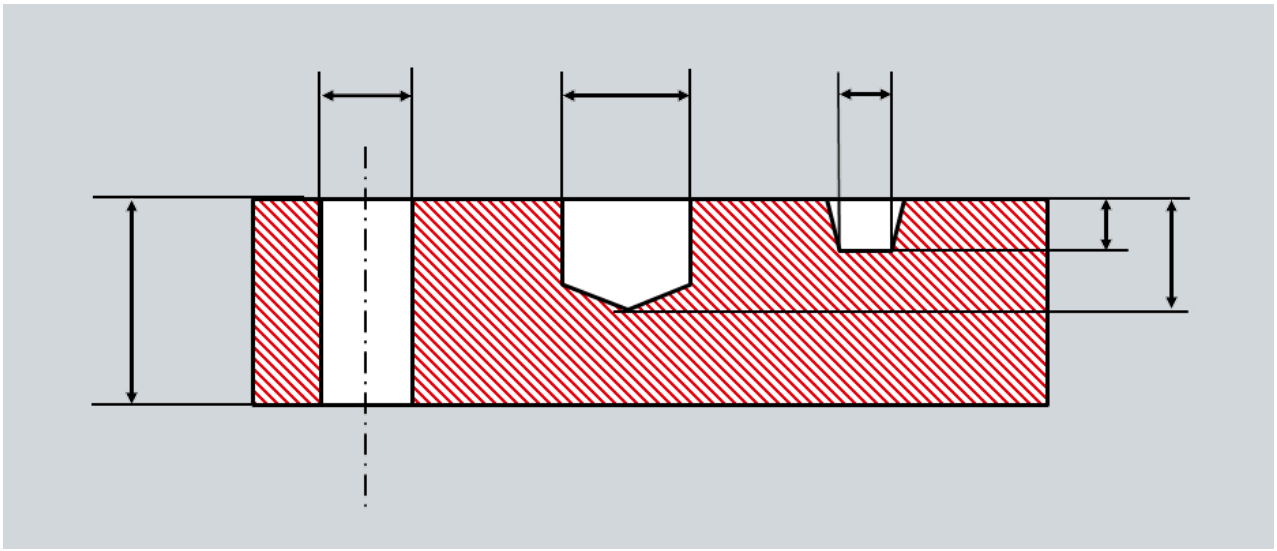
3.7 MECHANICS, GENERAL CHARACTERISTICS

3.7.1 MECHANICAL DRILLING, LASER DRILLING

"Aspect Ratio" with drilling:
Ratio of drill depth to drilling tool diameter

Example:

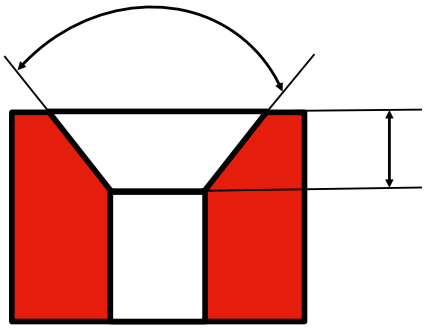
$$\textbf{Aspect Ratio} = \frac{\text{Drill depth}}{\text{Drilling tool diameter}} = \frac{1{,}6\ \text{mm}}{0{,}2\ \text{mm}} = \mathbf{8}$$



Feature	Standard	Higher requirements
Aspect ratio PTH	≤ 8 (not for HAL surface)	≤ 10
Aspect ratio Microvia	≤ 0,65	≤ 0,8
Aspect ratio blind via mechanically drilled	≤ 0,8	—
Ø Drill tool	0,25 mm to 6,00 mm	0,20 mm
Ø Tool increments	Tool Ø ≤ 4,5 mm : 0,05 mm	
	Tool Ø > 4,5 mm : 0,10 mm	
Slot, minimum width	0,8 mm	—
Slot, minimum length	0,8 mm	—

3.7.2 COUNTERSINKING

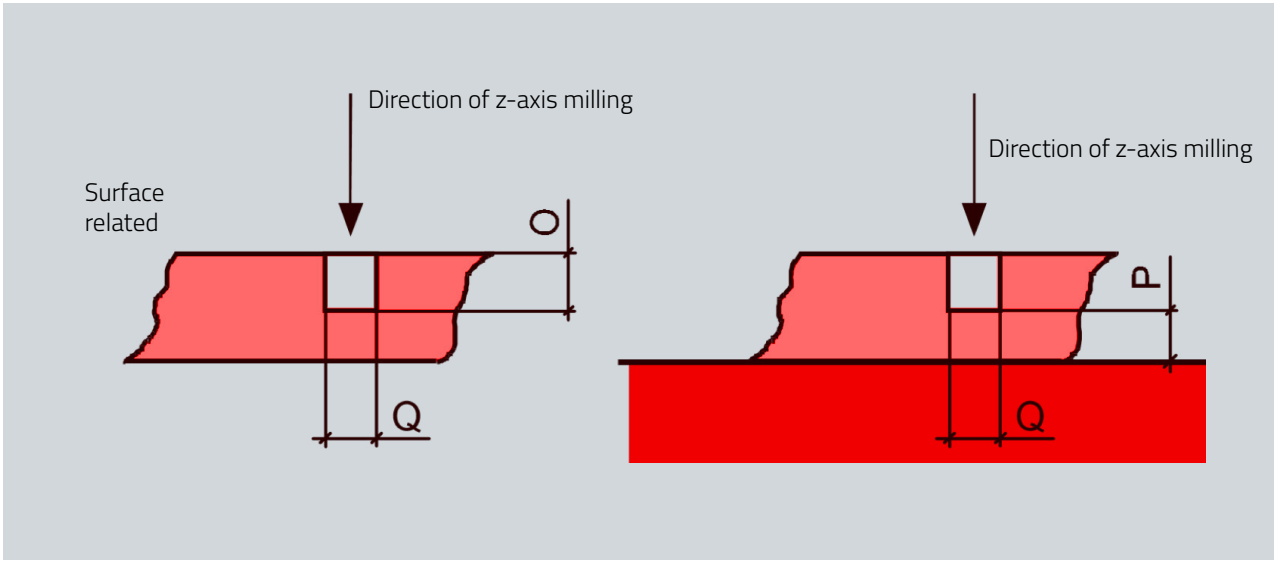
Feature	Standard	Higher requirements
Version	Countersunk head DIN 74m	Countersunk head DIN 74f
Other versions	Flat countersunk for cylinder head screw	—
Angle	40° / 60° / 90°, tolerance ± 1°	—
Depth tolerance	±0,1 mm	—



3.7.3 MILLING

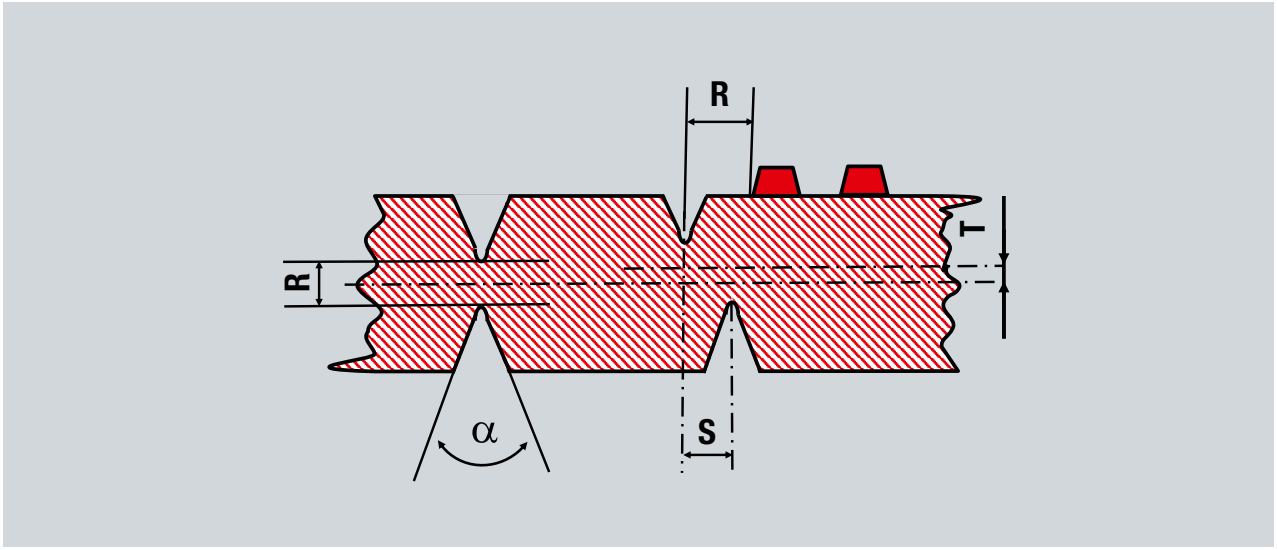
Feature	Standard	Higher requirements
Ø Tool	0,8 to 2,4 mm	0,5 to 0,7 mm > 2,4 mm
Ø Tool increments	Generally, increments of 0.1 mm	
Minimum distance copper-outline	0,23 mm for inner and outer layers	
Tolerances: see chapter 3.8		

3.7.4 Z-AXIS MILLING



Tolerances: see chapter 3.8

3.7.5 V-GROOVE MILLING



Feature	Standard	Higher requirements
Circuit board thickness	0,7 to 3,2 mm	0,4 to 0,6 mm > 3,2 mm
Groove angle α	30° ± 5°	45° ± 5°
Residual web thickness R	0,3 mm ± 0,10 mm	—
Max. parallel offset S	± 0,10 mm	—
Max. center offset T	± 0,10 mm	—
Min. distance groove to copper	depending on residual web and PCB thickness, see next table	
Groove with jump scoring Dive Zone: / Release zone:	10 ± 0,5 mm / 10 ± 0,5 mm	8 ± 0,5 mm / 8 ± 0,5 mm
Groove on one side only	possible	

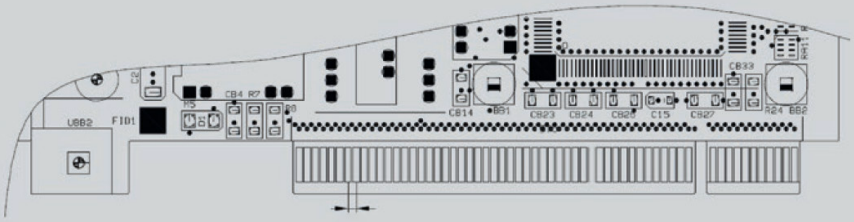
	Material thickness	Distance copper to groove
Values for Groove angle 30°	0,80 mm	≥ 0,40 mm
	1,00 mm	≥ 0,40 mm
	1,55 mm	≥ 0,50 mm
	2,00 mm	≥ 0,60 mm
	2,40 mm	≥ 0,70 mm
	3,00 mm	≥ 0,80 mm

3.7.6 CHAMFERING

Feature	Standard	Higher requirements
Chamfer angle	20° / 30° / 45°	—
ISA Plug standard	45°	—
PCI Plug standard	20°	—

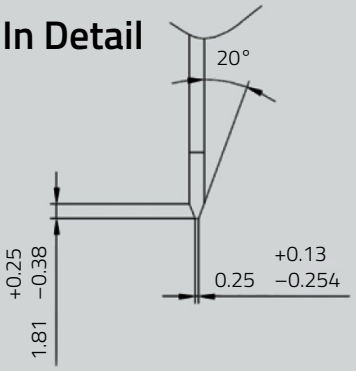
If there are no specifications, we use 45 degrees.
The copper on the outer and inner layers is reset accordingly.

PCI 20°

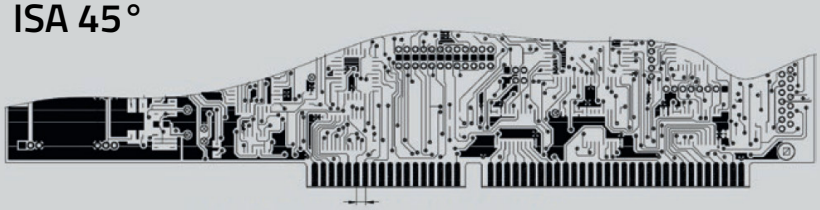


min. 1.27 mm distance from connector tab to connector tab

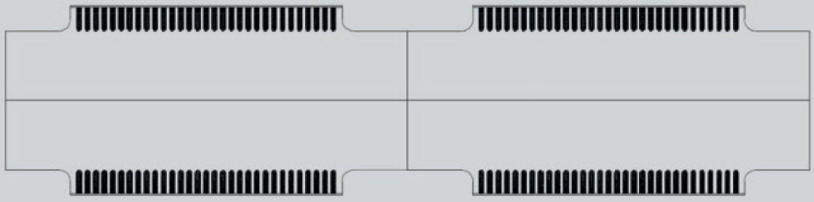
In Detail



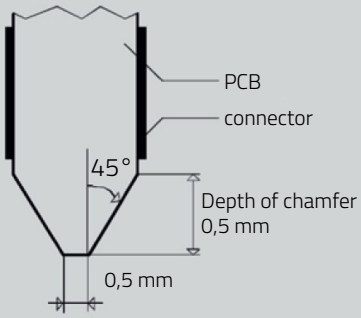
ISA 45°



min. 2.54 mm distance from connector tab to connector tab



In Detail

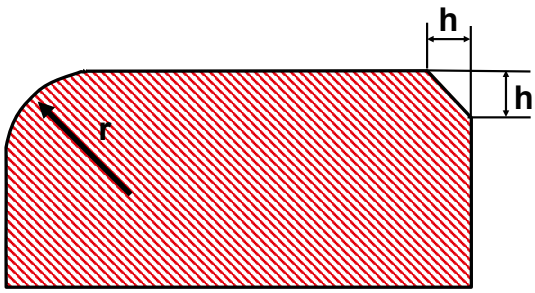


3.8 OTHER NOMINAL DIMENSIONS AND TOLERANCES

For linear dimensions (e.g. external contours), DIN ISO 2768 Part 1 applies:

Dimensions in [mm] for nominal size range in [mm]							
Level of accuracy	0,5 to 3	> 3 to 6	> 6 to 30	> 30 to 120	> 120 to 400	> 400 to 1000	> 1000 to 2000
fine	±0,05	±0,05	±0,10	±0,15	±0,20	±0,30	±0,50
middle	±0,10	±0,10	±0,20	±0,30	±0,50	±0,80	±1,20
For nominal dimensions below 0.5 mm, the dimensions must be indicated at the nominal dimension.							

Limit dimensions for rounding radius [r] and chamfer heights [h]

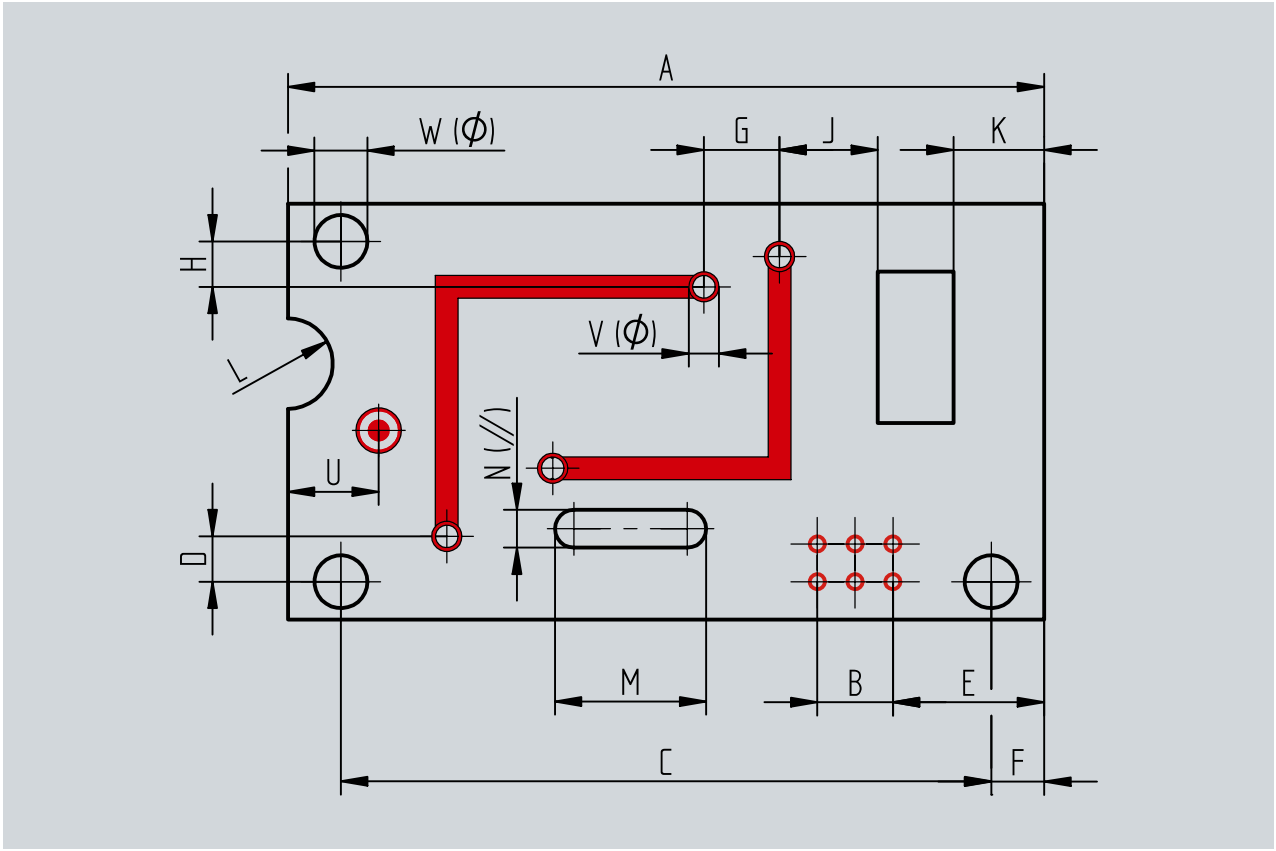


The values in the table are based on DIN ISO 2768 Part 1:

Dimensions in [mm] for nominal size range in [mm]					
Level of accuracy	0,5 to 3	> 3 to 6	> 6 to 30	> 30 to 120	> 120 to 400
fine	±0,2	±0,5	±1,0	±2,0	±4,0
middle					
For nominal dimensions below 0.5 mm, the dimensions must be indicated at the nominal dimension.					

MECHANICAL TOLERANCES

The following illustrations and tables show the dimensions in mm for nominal dimension ranges (based on DIN ISO 2768 m).



Explanations for the identifiers on page 18 and 19:

- **A** External dimensions
- **B** Tolerance between the PTH holes
- **C** Tolerance between NPTH drill holes or locating holes
- **D** Tolerance of the connection and mounting holes to the PTH drilling pattern
- **E** Tolerance of a PTH hole to the next PCB edge
- **F** Tolerance of the nearest board edge from the reference locating hole or an NPTH hole
- **G** Tolerance of the conductor pattern to a PTH hole
- **H** Tolerance of the conductor pattern to an NPTH hole
- **J** Tolerance from a breakout (notch) to a PTH hole
- **K** Tolerance from a cutout (notch) to the nearest board edges
- **L** The roundness deviation of a milling operation (e.g. circular milling, circular nibbling)
- **M** Slotted external dimensions with corresponding tolerance specification
- **N** Parallelism of one slot
- **O** Surface-related milling depth (Z axis)
- **P** Milling depth related to the machine table (Z axis)
- **Q** Milling width (Z axis)
- **U** Position tolerance milling edge to conductor pattern (Fiducial)
- **V** Standard tolerance for PTH drill holes
- **W** Standard tolerance for NPTH drill holes

Identifier	Assignment	Variable	Standard	Advanced	Nominal dimension range
A	Outline	Milling	± 0,10 mm	± 0,075 mm*	0 – ≤ 30 mm
			± 0,15 mm	± 0,10 mm*	> 30 – 120 mm
			± 0,20 mm	± 0,15 mm*	> 120 – 200 mm
			per each further 100 mm ± 0,05	± 0,20 mm	> 200 mm
				± 0,30 mm	> 400 mm
		V-grooving	DIN ISO 2768 fine (unbroken as delivered)	—	—
		Milling + grooving	DIN ISO 2768 fine (unbroken as delivered)	—	—
B	PTH to PTH	Tool-Ø ≤ 6,00 mm	± 0,05 mm	—	—
		Tool-Ø > 6,00 mm	± 0,05 mm	—	—
C	NPTH to NPTH		± 0,10 mm	± 0,05 mm	—
D	PTH to NPTH		± 0,10 mm	± 0,05 mm	—
E	Outline to PTH	Milling	± 0,10 mm	—	0,5 to 6 mm
		Grooving	± 0,15 mm	—	0,5 to 6 mm
		Milling or grooving	± 0,20 mm	—	6 to 30 mm
		Milling or grooving	± 0,30 mm	—	> 30 mm
F	Outline milled to NPTH		± 0,10 mm	—	—
	Outline grooved to NPTH		± 0,15 mm	—	—
G	Conductor pattern to PTH	Tool-Ø ≤ 6,00 mm	± 0,05 mm	—	—
		Tool-Ø > 6,00 mm	± 0,05 mm	—	—
H	Conductor pattern to NPTH	NPTH drilling in PTH program	± 0,05 mm	—	—
		NPTH drilling in outline program	± 0,10 mm	—	—
J	Conductor pattern to PTH	Milling	see Identifier E	—	—
K	Contour to cutout	Outline Milling	± 0,10 mm	—	—
		Outline Grooving	± 0,15 mm	—	—
		Outline Milling + Grooving	± 0,15 mm	—	—
L	Rounding accuracy	Milling	± 0,10 mm	± 0,075 mm	—
M	PTH- u. NPTH slots Length / width	Milling, drilling (nibble)	± 0,10 mm	± 0,05 mm	—
N	Slots NPTH PTH parallelism	Milling, drilling (nibble)	± 0,10 mm	± 0,05 mm	—
O	Milling depth (Z axis)	Milling	± 0,10 mm	± 0,05 mm	Surface related
P	Milling depth (Z axis)	Milling	± 0,05 mm	—	machine table related
Q	Milling depth (Z axis)	Milling	see Identifier E	—	
U	Outline to conductor pattern (Fiducial)	Milling	± 0,15 mm	—	0,5 to 6 mm
		Grooving	± 0,20 mm	—	0,5 to 6 mm
		Milling	± 0,25 mm	—	6,0 to 30 mm
		Milling	± 0,35 mm	—	30 to 120 mm
		Milling	± 0,55 mm	—	120 to 400 mm
		Milling	± 0,85 mm	—	400 to 1000 mm
V	PTH Ø	Drilling	± 0,10 mm	+0,1/–0,05 mm	—
W	NPTH Ø	Drilling	± 0,10 mm	± 0,05 mm	—

* If no other tolerances counteract, e.g. drills to contour and shrinkage/elongation values are ignored.

3.9 REPAIRS

Smaller defects in the solder resist are professionally repaired. Locally limited short circuits on inner and outer layers can be removed. This is followed by a visual inspection of the inner layers or an electrical retest.

4 INTERNAL PRODUCT FEATURES

These product features are located within the printed circuit board, the evaluation is carried out by checking microsections. Here the IPC-A-600 chapter 3 is applied and provides criteria for the evaluation of the following features:

- Dielectric materials
- Conductor pattern inner layers, copper thickness inner and outer layers
- Metallized holes (PTH), annular rings inner layers, copper thickness, microvia
- Filling of drill holes (Filling according to IPC-4761).

The IPC-A-600 regulates aspects not dealt with in detail below.

4.1 BASE MATERIAL

All base materials used are IPC compliant. Please ask us if you have special requirements.

Rigid base materials are specified in IPC-4101 and its specification sheets. For standard FR-4.0 with Tg135, for example, specification sheet 21 applies; higher-grade FR-4.1, for example, in specification sheets 128 (92, 94, 127) for elevated operating temperatures with Tg150 and low CTE.

Our portfolio is rounded off with CEM-1 materials (specification sheets 10, 12, 14, 15, 81) and CEM-3 materials (specification sheets 16, 35) as well as high performance materials with higher temperature resistance or low loss materials for high speed / high frequency applications (IPC-4103).

Flexible base materials are specified in the standards IPC-4202/IPC-4203 and IPC-4204. Our used polyimide foils are always high quality and adhesiveless.

Standard copper foils according to IPC-4562.

4.2 STACKUP

The stack-up plan is the necessary design specification for all multilayer circuits.

It defines
- the materials used with their qualities and dielectric values
- the number of copper layers and their connections through vias
- the thickness of the copper layers, the layer spacing and the total thickness with the permissible tolerance.

Würth Elektronik offers cost- and production-optimized standards for the technologies BASIC, HDI, Flex and Rigid-Flex at the internet site **www.we-online.com/pcb**. We use these if you do not make any specifications. On the website you will also find all standards as digital stackup files for import into your EDA software.

4.3 CONDUCTOR PATTERN INNER LAYER TOLERANCES

Conductor pattern structures on the inner layers are created from the copper foil by means of etching technology. Conductors and copper surfaces are modified in the same way as standard in order to counteract underetching during the conductor pattern structuring process.
The permissible tolerances in the dimensions are minus 20 percent.

4.4 COPPER LAYER THICKNESSES

Minimum acceptable copper layer thicknesses are defined by the IPC standards:

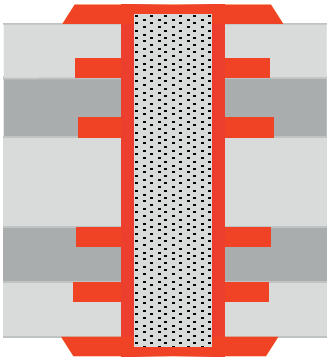
- For internal layers (Minimum Internal Layer Copper Foil Thickness): IPC-6012 3.6.2.14 for rigid PCBs, IPC-6013 3.6.2.15 for flexible and rigid-flexible PCBs.
- For outer layers (Minimum Surface Conductor Thickness after Plating): IPC-6012 3.6.2.15 for rigid PCBs, IPC-6013 3.6.2.16 for flexible and rigid-flex PCBs
- For vias (Hole Copper Plating Minimum Requirements) in Tables IPC-6012 Table 3-4 and 3-5 for rigid PCBs, IPC-6013 Table 3-3, 3-4 and 3-5 for flexible and rigid-flex PCBs.

4.5 FILLING OF PLATED HOLES
IPC-4761, Type VII

For filling according to IPC-4761, Type VII (Filled & Capped Via) we fully fill vias with a non-conductive paste. Please indicate in your fabrication data which holes (diameter) should be filled in this way.

Applications:

- Alternative to plugging IPC-4761 Type III if the minimum distance to adjacent soldering surfaces cannot be maintained
- For thermal vias in soldering surfaces



5 PERFORMANCE FOR PROCESSING AND CUSTOMER APPLICATION

5.1 ELECTRICAL TEST OF PRINTED CIRCUIT BOARDS

The electrical testing of the printed circuit boards is carried out on parallel testers with article-specific needle bed adapters or by finger testers. For the creation of the test program the customer data is converted (CAM Data Test). All of the PCBs are tested, using the adjacency method.

As standard, the PCB or the delivery panel is marked with the following identification after the E-test: Line on the edge with black pen.

The standard process parameters for the electrical test are:

- Test voltage: 10 V
- Lower threshold value for the interruption test: 50 Ω
- Upper threshold value for the short circuit test: 10 MΩ

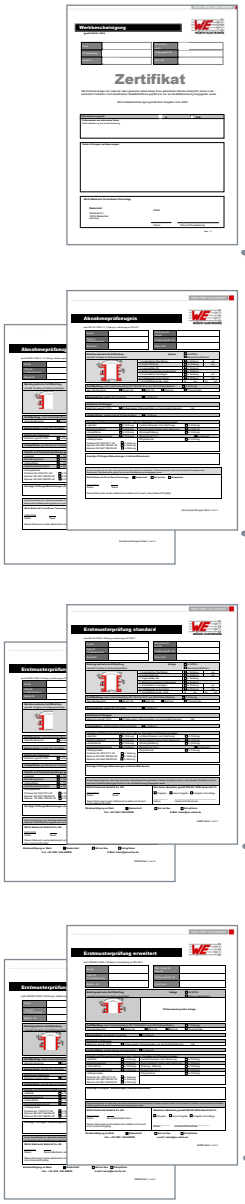
ADDITIONAL OPTIONS

TDR impedance measurement:
It is possible to measure characteristic impedance for specified transmission lines. Test coupons are generated and used for representative measurements.

Coil measurements:
It is also possible to carry out measurements on coils by arrangement.

Please ask for these options, we will be pleased to make you an offer.

5.2 TEST DOCUMENTATION



According to DIN 10204 we offer the following test certificates:

- Certificate of compliance with the order (free of charge)
- Acceptance test certificate (chargeable)
- Initial sample inspection standard (chargeable).
- Initial sample inspection extended (chargeable).

For this inspection documentation, the customer must specify the required inspection characteristics and agree them with Würth Elektronik Circuit Board Technology.

Look and content:

Certificate of compliance with the order
Confirmation of the fulfilment of customer requirements and compliance with certain standards. No documented inspection of certain features.

Inspection certificate
Dimensioned micrographs, detailed notes and measurement results of the most important characteristics regarding product structure, solder resist types and quality.

First article inspection report standard
Dimensioned micrographs, detailed notes and measurement results of the most important features regarding product structure and quality. Customer confirmation of the documented test results and release.

First article inspection report extended
Dimensioned micrographs, detailed target/actual comparison of all product features to be tested in accordance with customer order and customer specification. Customer confirmation of the documented test results and release.

5.3 BAD PARTS IN THE DELIVERY PANEL (X-OUTS)

In the case of delivery panels with several printed circuit boards, there is a statistical probability that individual printed circuit boards do not comply with the specification and will be scrapped off. This probability increases with the complexity of the products. For reasons of the protection of resources, economic efficiency and security of supply, delivery batches with bad parts are always delivered as well.

Bad parts in the delivery panel are marked with a black X on both sides. A maximum of 50 per cent "X-Outs" in the delivery part and 30 per cent over the entire delivery are permissible. Panels with bad parts are packed and marked separately.

5.4 PACKING

Standard packaging is a needled, antistatic PE shrink film with a foil thickness of 70 µm. The packaging units are selected in sensible packaging (usually 20 pieces). These packaging units are packed in cartons with a maximum total weight of 15 kg.

5.5 SURFACE CLEANLINESS

Ionic residues on inner layers before lamination and outer layers before solder resist coating: Max: 1.56 µm/cm² sodium chloride equivalent.

5.6 SOLDERABILITY, DRYING, RESISTANCE IN THE SOLDERING PROCESS

We guarantee the solderability of unpopulated printed circuit boards, depending on the soldering surface, for the duration specified in point 3.1, provided that they are handled and stored properly in their original packaging.

Printed circuit boards basically already have a certain moisture content directly after manufacture due to the manufacturing processes and ambient conditions. Without special treatment, they are therefore never free from moisture that has diffused into the dielectric materials. Over the storage period, the moisture content can increase further under unfavourable storage conditions.

In order to avoid damage during the soldering process, it may be necessary to dry the PCBs in a suitable oven immediately before assembly or for a longer period of time in a dry atmosphere (dry storage cabinet). For flexible and flex-rigid printed circuit boards, this drying is obligatory before soldering!

The necessary drying parameters depend on the PCB design (copper surfaces), the drying equipment and the arrangement of the PCBs therein, the PCB material, the soldering process and the soldering parameters, the latter possibly also multiple and combined from reflow, wave and partial soldering techniques. These drying parameters must be determined and verified at the PCB assembler. Special attention must also be paid to logistics and especially idle times, as the dried PCBs absorb moisture from the environment again. Packaging the PCBs in a Moisture Barrier Bag (MBB) for the supply chain and storage is not sufficient.

6 COMPLAINTS

Würth Elektronik Circuit Board Technology is known as a manufacturer of printed circuit boards with a high quality level. Should you nevertheless have reason for complaint, please contact us with the most important reference data:

You will find the contact persons in the order confirmation and the delivery note:

Reference data for a complaint are:
- Your contact details
- Product data
  (article description, delivery note number, date code)
- Error information and
- Quantity information.

Your account manager
in field sales service

Phone:

@we-online.com

Your inside sales
contact
in internal sales

Phone:

fax:

@we-online.de

Würth Elektronik GmbH & Co. KG
Circuit Board Technology
Salzstraße 21
74676 Niedernhall · Germany
Phone: +49 7940 946-0
cbt@we-online.com

October 2022

www.we-online.com