P-Flex®

Flexible Printed Circuit Board (Flex PCB)

P-Flex Specifications

Ver. 5.4.0

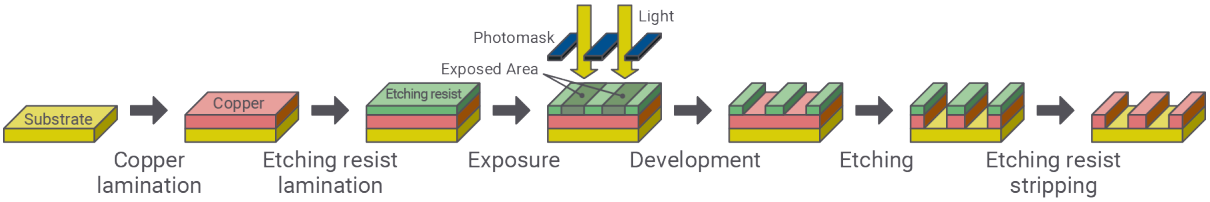
Last updated on: August 5, 2020

TABLE OF CONTENTS

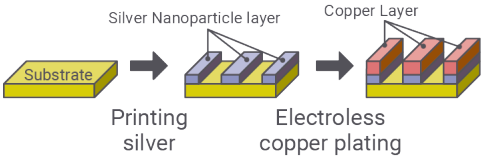
1. OVERVIEW

P-Flex® is a flexible printed circuit board (“PCB”) that is manufactured using inkjet printing and electroless copper plating processes. Unlike conventional PCBs that are manufactured using photolithography, this novel type of PCB does not require any initial manufacturing set-up cost, so it can be supplied to customers at significantly lower prices with far shorter lead time compared to conventional PCBs, especially in small-volume production lots.

Conventional process (Subtractive process)



Our process (Pure Additive™ process)



The use of electroless copper plating reduces the high electric resistance value, solving a weakness of conventional inkjet printing methods, also the adoption of inkjet printing omits the need to use screen mask to produce soldermasks, thus achieving product quality that is virtually identical to that of prototypes of regular flex PCBs, with shorter lead time and at reduced cost.

Unless otherwise specified, the various descriptions in the specifications are of common matters that do not depend on the material and layer composition. The substrates are specified when exceptions apply. Guidelines relating to the design and necessary data are specified separately in the appended guideline [P-Flex Design Guideline]; please check this guideline when designing or placing an order. Data for the various tests will be added sequentially.

2. PRECAUTIONS

When using the product, please follow the precautions that are provided below to properly store and use the product.

- As for the inspection specifications, please refer to the Inspection Specifications that are separately provided.
- Please note that there may be probe marks on the pads that were left during open/short testing.
- When only a small quantity is processed, traces of burn and/or discoloration may appear near the cutting surface, as it involves the use of a laser to cut along the outline. However, these have no negative effect on conductivity or insulating performance.
- Overall warpage may occur in some cases, which is a common issue with any flex PCB but this will not have any negative effect on the connection with the connectors or any other electrical characteristics. If, however, it is necessary to keep a levelled surface, it might be better to attach stiffeners, etc. For more details, please consult our staff.
- As touching the circuit patterns with bare hands will leave fingerprints, we recommend wearing gloves when handling the product. In addition, please make sure that components are mounted onto the product within three months.
- The dimensions guaranteed for this product shall be measured at 20˚C and at 50% humidity. Also, dimensions and measurement values provided as reference data have been measured at a temperature ranging from 15˚C to 20˚C unless stated otherwise.
- When disposing this product, necessary measures must be taken to dispose of it as industrial waste.

3. MANUFACTURING SPECIFICATIONS AND VARIOUS CHARACTERISTICS

3.1. MANUFACTURING SPECIFICATIONS

Table 3-1: Manufacturing Specifications

Substrate	Transparent heat-resistant PET film: 50 μm thick, 125 μm thick PI (Polyimide) film: 25 μm thick
Line width / spacing	200/200 μm min., 200/150 μm min. (only PET substrate option)
Hole diameter	0.5 mm min.
Outline-pattern spacing	0.3 mm min.
Copper foil thickness	3 μm (please consult us if you require a thickness of more than 3 μm)
Panel size	388 x 226 mm max.
Wiring layer	Single-sided
Soldermask coating (for PET substrate only)	UV inkjet printing method (transparent) (can also be specially ordered with a waterproof grade soldermask coating) For PET substrates, green, white and black soldermask coating by silk screen printing is also available as a developmental product.
Coverlay pasting (for PI substrate only)	PI film 12.5 μm, adhesion layer 15 μm Only coverlay pasting is available for PI substrates
Legend printing	UV inkjet printing (white)
Surface finish	Oxidation prevention treatment, Electroless nickel gold plating (option)
Outline cutting	Laser cutting
Hole drilling	Laser cutting
Stiffeners	When specifying the thickness of the connector part, film stiffeners can be used to adapt to connectors of 200 μm or 300 μm in total thickness. Component mounted parts and such can be adapted by using FR-4 stiffeners of 0.1, 0.3, 0.5, 1.0, and 1.6mm in thickness. Other stiffeners, electromagnetic wave shield film, double-sided tape and such are also available.
Inspection	Visual inspection + opens/shorts test

*Please consult Elephantech if other manufacturing specifications are preferred to those described above.

3.2. LAYER COMPOSITION

The layer compositions of P-Flex® PET as well as P-Flex® PI are as specified in **Figure 3-1** below. The thickness data shown below are provided for reference purposes only and are not guaranteed values. The standard layer composition is shown in the illustration on the left.

While the substrate for the P-Flex® PET is rather thick, there is no adhesive layer for copper foil or soldermask and so the total thickness is roughly the same as that of a flex PCB with a substrate that is 25 μm thick. P-Flex® PI uses an even thinner substrate that is 25 μm thick.

Nickel gold plating and FR-4 stiffeners are also available as options. The illustration on the right shows the layer structure when all these options have been added. The film stiffeners can be used for the part to insert into the connectors with specified thicknesses of 200 or 300 μm. When selecting a waterproof graded soldermask, the theoretical thickness shall increase from the usual 25 μm to 40 μm. If the gold plating has been selected, the copper layer below the gold plated parts shall be thinner by approximately 0.5 μm due to soft etching of the underlying copper.

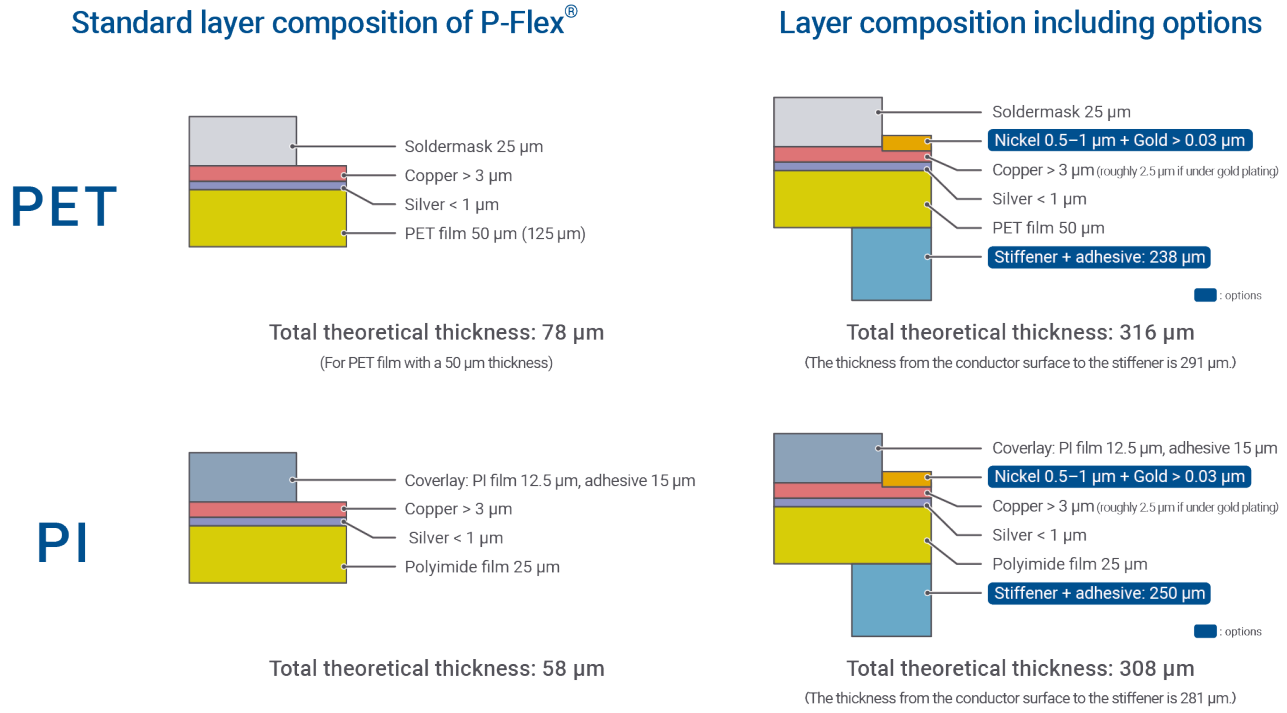


Figure 3-1: Layer composition of P-Flex®

3.3. VARIOUS CHARACTERISTICS

Table 3-2: Various Characteristics

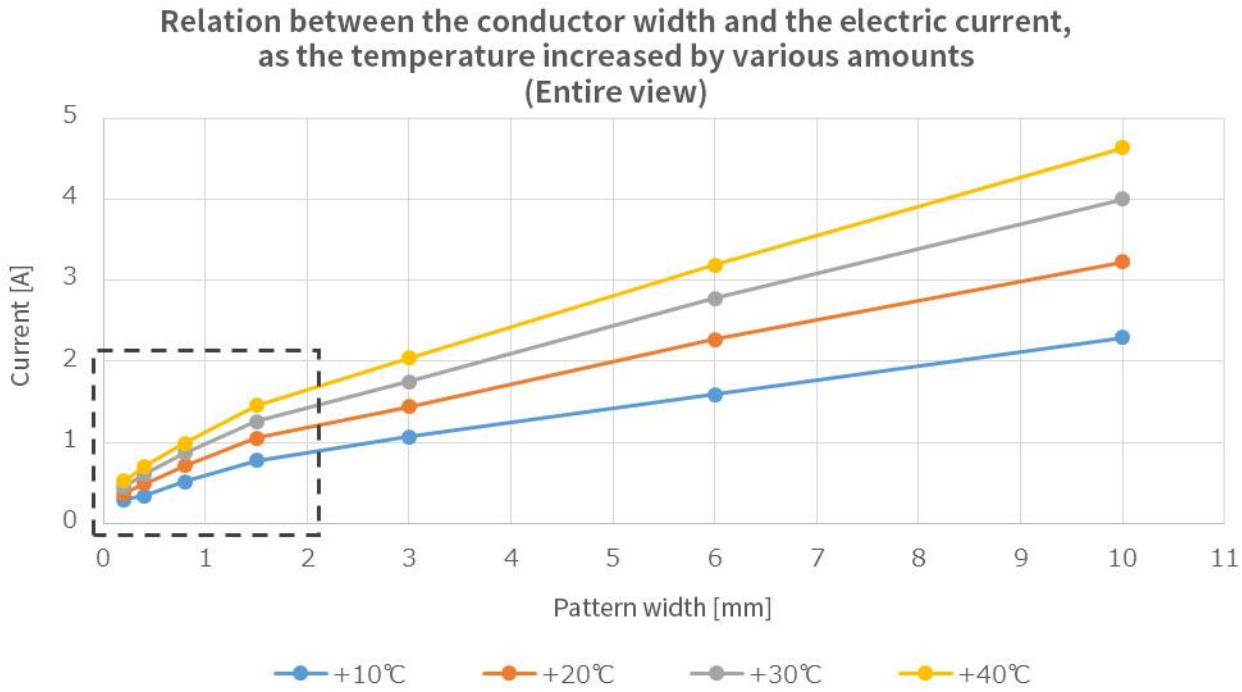
Thickness tolerance of copper foil	It should not be less than the specified foil thickness.
Sheet resistance	6.7 mΩ / Sq. (at a copper foil thickness of 3μm), which is inversely proportional to the copper foil thickness.
Operating temperature	Between -20°C and +105°C
Reflow heat resistance	PET: 200°C for five seconds PI: 260°C for ten seconds

*The copper foil thickness specified here is confirmed by calculating the copper foil thickness from the electric resistance value of the test pattern on the same artwork as the product. Here, the volume resistivity of electroless copper foil is set to 2.0x10^-8Ωm.

3.4. REFERENCE DATA ON AMPACITY

The data that show the relation between the current and the increase in temperature are provided in **Figure 3-2** below, as reference data regarding ampacity. The thickness of the copper foil is 3 μm. However, please note that these values are provided for reference purposes only and should not be viewed as guaranteed values.

As for the testing method that was used to gather the data, voltage was applied starting at room temperature so that the specified rise in temperature could be achieved (i.e., +10–+40℃), while the current was measured and recorded. To run an electric current, a test piece was set up on top of two fixation jigs like a bridge. Then it was made sure that nothing was contacting the area where measurements would be taken, with a cover put over the area to shield it from any air movements caused by air-conditioners, etc., so that only natural heat-induced convection could occur.



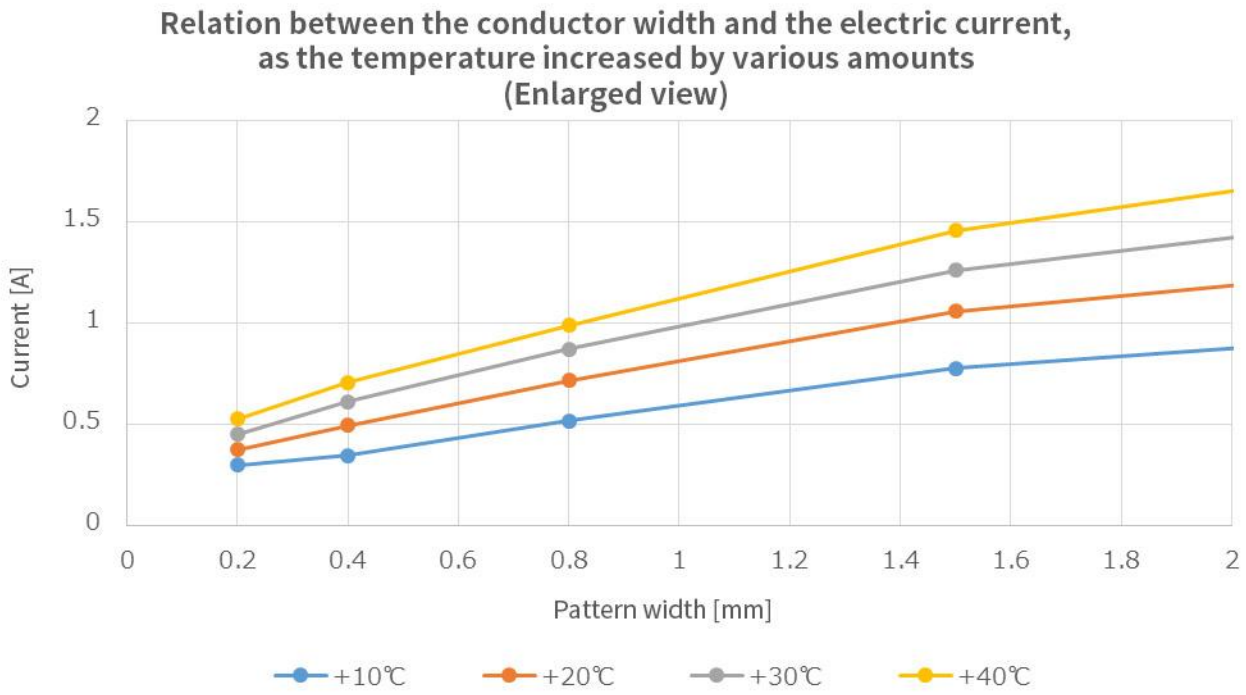


Figure 3-2: Relation between the conductor width and the electric current value, as the temperature increased by various amounts (3μm Copper foil thickness)

3.5. REFERENCE DATA ON THE MINIMUM BENDING RADIUS

Reference data regarding the minimum bending radius is provided in **Table 3-3**: Reference data on the minimum bending radius below, which can be used when bending FPCs with metal molds, etc. The Copper foil thickness is 3μm and the substrate is PET. However, it must be noted that the data is provided for reference purposes only and is not a guaranteed value.

Table 3-3: Reference data on the minimum bending radius

Minimum bending radius R [mm]	0.5

3.6. REFERENCE DATA ON ANTI-IONIC MIGRATION

Reference data on the anti-ionic migration characteristic of the product is provided in **Table 3-4** below**.** Please note, however, that the data is provided for reference purposes only and these are not guaranteed values. The test pattern used consisted of interdigital electrodes placed at 0.5 mm intervals, to which a voltage of 50 V was applied.

While silver is used in the seed layer, copper coating is subsequently applied using electroless plating all over the surface. Therefore, no ionic migration of silver will occur under normal circumstances.

Table 3-4: Reference data on anti-ionic migration

Condition	Observation of ionic migration
85°C 85%Rh 1440 hours	None observed

3.7. ON ROHS AND REACH

All products we ship comply with the RoHS Directive as standard. Also, we do not use substances that are listed as Substances of Very High Concern (SVHC) by the REACH regulation. If required, we will gladly provide certificates of compliance of RoHS and REACH upon request. Please contact us.

3.8. ON UL CERTIFICATION

The UL94 (flame retardant standard) conformity test has been carried out and the results are as follows.

Table 3-5: Explanation on UL certification

Substrate	UL94
PI	VTM-0 equivalent (Pending certification number)
PET	None.

3.9. PATTERN PEEL STRENGTH REFERENCE DATA

Reference data on peel strength between the substrate and the copper pattern is shown in **Table 3-6**. The peel test is based on JIS-K5600 (cross cut method).

Table 3-6: Peel Strength Reference Data

Substrate	Peel strength
PI	Passed peel test
PET	Passed peel test

4. DATA ENTRY SPECIFICATION

Guidelines relating to the necessary data are specified separately in the appended guideline [P-Flex Design Guideline]; please check this guideline when designing or placing an order.

5. MANUFACTURING STANDARD AND PRODUCT RETURN RULES

5.1. GENERAL DIMENSIONAL TOLERANCE

The general dimensional tolerance is as shown in **Table 5-1**. This general dimensional tolerance shall apply to the distance between any given two separate points, such as the distance between circuit patterns, or the length of a line. In addition, Elephantech has a different set of specifications to be applied to certain parameters pertaining to out-of-bounds pattern width, outline dimensions, location of stiffeners to be applied to the back of PCBs, etc.

However, this does not mean that we conduct dimensional measurements for all products using a dimensional meter with respect to the specified dimensions. Please contact us if you wish for the dimensions to be measured.

Table 5-1 Table of general dimensional tolerance

Length	General dimensional tolerance
Less than 200mm	The larger value between ±0.5% and ±0.05mm
More than 200mm	The larger value between ±0.3% and ±1.0mm

5.2. OUT-OF-BOUNDS AND MISSING PATTERNS, PITS, AND PATTERN FLOATAGE

As the conductive patterns are created using inkjet printing and electroless copper plating processes, certain irregularities may occur such as out-of-bounds patterns and missing patterns, and also pits (i.e., bubble marks) and pattern floatage that may occur during the plating process. In this connection, slightly uneven plating finish may occur around any pits, but it should not be counted as part of those pits. Similarly, out-of-bounds and missing soldermask may also occur as it is created using inkjet printing. The tolerances for these irregularities are as specified in **Table 5-2** below**,** which are expressed as allowable length in proportion to the design dimension in the direction of the width.

However, if any of these types of irregularities should cause practical issues with correct product usage, they will be removed even if they are within the specified tolerance.

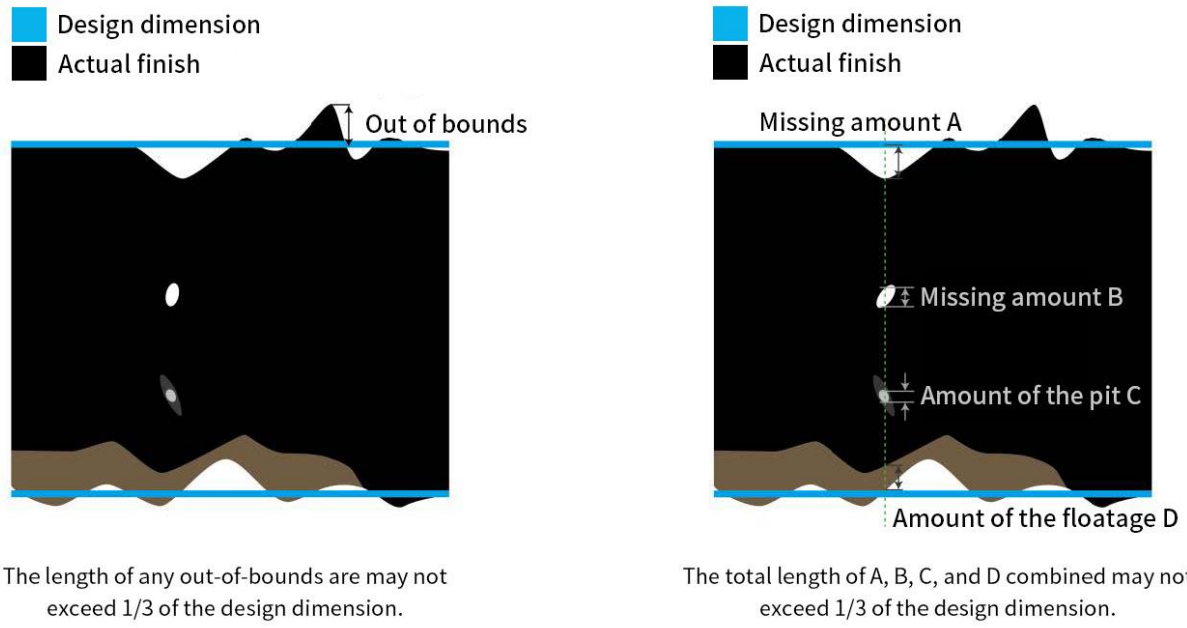


Figure 5-1: Illustrations of out-of-bounds and missing patterns, etc.

Table 5-2: Tolerances on out-of-bounds and missing patterns, etc.

Item	Tolerance
Out of bounds	The length of any out-of-bounds pattern should not exceed 1/3 of the design dimension
Missing pattern, pit, and pattern floatage	The total length of these irregularities combined should not exceed 1/3 of the design dimension or 0.1 mm, whichever value is larger.

5.3. PATTERN BREAKAGE AND SHORT CIRCUIT

No break in the circuit is allowed for any section of the product that is the same width or wider than the minimum pattern width as specified in **Table 3-1**. Likewise, no short circuit is allowed for any section of the product that is the same width or wider than the minimum pattern width as specified in **Table 3-1.**

5.4. INK SPLASH

The product is prone to having the kind of ink splash shown in due to the manufacturing process in which the conductive ink is applied by inkjet printing**.**

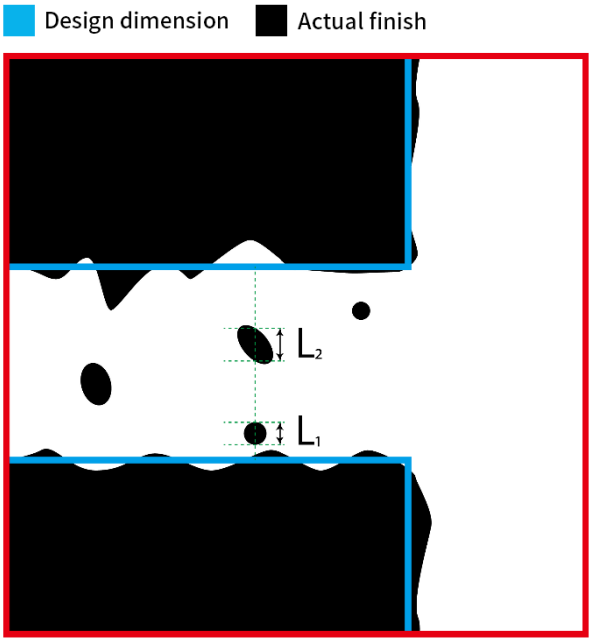


Figure 5-2: Ink splash

No short circuit between patterns that is caused by ink splash is allowed. In addition, with regard to the shortest line connecting any given point on one pattern to another pattern, the tolerance for ink splash is as specified in **Table 5-3** below**.**

Table 5-3: Tolerance for ink splash

Item	Tolerance
Maximum length=max(L)	0.05 mm
Total length=$L_1+L_2+\cdots L_N$	No more than 1/2 of the spacing between conductive tracks
No. of splashes=N	Maximum of five splashes every 0.2 mm

5.5. OTHER TYPES OF IRREGULARITIES

Tolerances for other types of irregularities are as specified in **Table 5-4** below.

Table 5-4: Types of other irregularities and their tolerances

Item	Tolerance
Foreign objects	Foreign objects up to a maximum length of 1 mm are allowed so long as they do not cover multiple conductive or non-conductive areas.
Air bubble	Air bubbles up to a maximum length of 1 mm are allowed so long as they do not cover multiple conductive or non-conductive areas.
Scratch	Scratches are allowed so long as they do not cause any broken or short circuits.

5.6. SOLDERMASK

Soldermasks and polyimide coverlays are generally used for flexible PCBs. At Elephantech, we use soldermasks for PET substrates. Specifically, our method prints soldermask ink by UV inkjet and enables maskless manufacturing. For PI substrates, polyimide coverlays are used just as with conventional flexible PCBs. As for design rules, the same soldermasking rules for PET substrates shall apply.

The roles of the soldermask and coverlay are to:

- prevent the solder from flowing onto sections of the wiring where it is undesired
- prevent solder bridging

when mounting parts to the pad by soldering or reflow. As stated above, the role of the soldermask is to assist in component mounting and does not guarantee the absence of pinholes. To completely eliminate pinholes and gain waterproofing, please select a waterproof graded soldermask. Also, soldermasks shall be free of the practical issues of peeling.

Regarding the splashes from soldermask, the tolerance values for ink splashes in **Table 5-3** shall apply correspondingly. In regard to any printing misalignment between the conductive patterns and soldermask, general tolerances are as specified in **Table 5-5** below, so long as no practical issues are caused, such as a pad being completely covered and such. However, the broadening of soldermask is as shown in **Figure 5-3**.

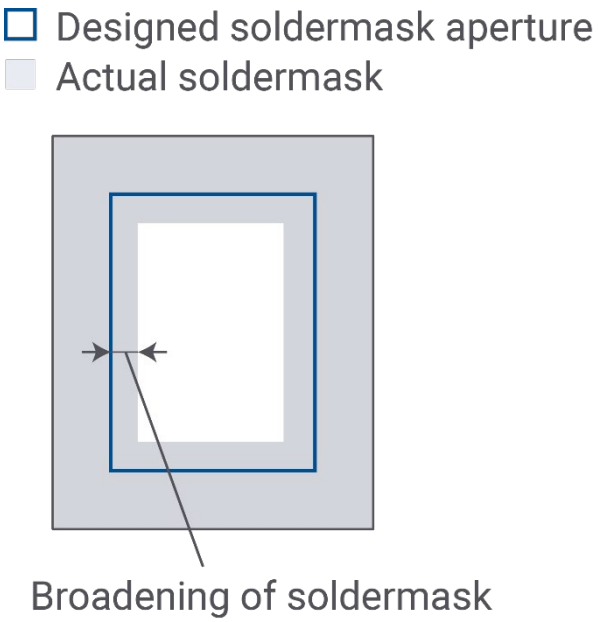


Figure 5-3: Broadening of soldermask

Waterproof graded soldermasks free of pinholes are available as a special specification. There is a possibility that this waterproof graded soldermask may have lower bend resistance and

the soldermask may broaden further than designed due to the pinholes being removed by the overcoating of the soldermask layer. Also, the theoretical thickness shall increase from the usual 25 μm to 40 μm when waterproof graded soldermask is selected.

Design guidelines concerning the soldermask are specified separately in the appended guideline [P-Flex Design Guideline]; please check this guideline when designing.

5.7. LEGEND

This refers to the symbols that are printed on PCBs that are intended to serve as indications for helping the mounting process, which is sometimes referred to as 'silk-screen printing,' etc. The tolerance for any legends being printed off the correct locations in relation to the corresponding circuit patterns is as specified in **Table 5-5.** Design guidelines concerning the legends are specified separately in the appended guideline [P-Flex Design Guideline]; please check this guideline when designing.

5.8. OUTLINE DIMENSIONS

As the outline of PCBs is cut using laser, the tolerances for the distance between the opposite sides of the outline and the outline-pattern distance are less than the general dimensional tolerances as shown in **Table 5-5.** It must be ensured by each customer that a minimum outline-pattern distance of 0.3 mm exists. As the outline of PCBs is processed using laser, any laser-cut area and circuit pattern being too close to each other potentially risks leading to burnt or exposed patterns. Please consult us if you require cuts to the copper foil parts.

Any parts of PCBs that require especially high-precision processing, such as holes for mounting components and connector parts, can be created as shown in **5.10 SPECIAL SPECIFICATIONS FOR CONNECTORS** below. Please send an inquiry to Elephantech for any order that involves such high-precision processing needs.

5.9. STIFFENERS

Stiffeners are used to prevent the component-mounted areas of PCBs from bending too much and also to achieve evenness in the thickness around the connector terminals. Concerning the part of connector terminals, the PCBs and the attached stiffeners are cut out together. For all other parts, the PCB and the stiffeners are cut out separately and visually aligned by hand. Tolerance for the deviation between the outlines, legends and stiffeners are as shown in **Table 5-5**.

Design guidelines concerning the stiffeners are specified separately in the appended guideline [P-Flex Design Guideline]; please check this guideline when designing.

Table 5-5: Tolerances on positional and dimensional precision, aside from circuit patterns

Item	Tolerance
Between any circuit pattern and soldermask	±0.2 mm
Between any circuit pattern and legend	±0.7 mm
Outline size	±0.3% or ±0.5 mm, whichever is larger
Between the outline and any circuit pattern	±0.3% or ±0.3 mm, whichever is larger
Between the outline or any legend and a stiffener	±0.7 mm
Soldermask broadening	0.1 mm, (waterproof grading of 0.3mm)

5.10. SPECIAL SPECIFICATIONS FOR CONNECTORS

If high-precision is required for properly aligning any connector sections, etc. with the outline dimensions and stiffeners, the options as shown in **Table 5-6** below are available for manufacturing the connector insertion sections, etc. with the indicated tolerances. For example, when using a connector with a 1.0 mm pitch or a 0.5 mm pitch, an outline tolerance of ±0.12 mm or ±0.07 mm respectively is recommended.

Please consult us beforehand if an even higher precision requirement, such as ±0.05mm tolerance in outline dimensions, is needed for the connector. Design guidelines concerning the connectors are specified separately in the appended guideline [P-Flex Design Guideline]; please check this guideline when designing.

Table 5-6: Special specifications for connectors

Item	Tolerance
Outline size	±0.07 mm
Between the outline and any circuit pattern	±0.07 mm

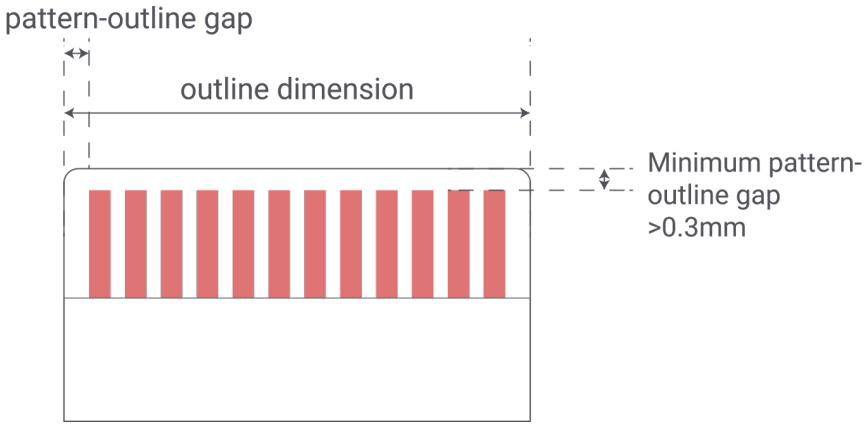


Figure 5-4: Dimensions of the connector part

5.11. PRODUCT RETURN AND REPLACEMENT RULES

If any product unit does not conform to any of the manufacturing specifications provided above, it can be replaced with a new unit, free of charge. In such case, please contact Elephantech to report the defect using the contact information that is included in the product package, within 30 days of the product delivery date.

6. DISCLAIMERS

- These Specifications have been prepared by Elephantech with utmost care. However, Elephantech does not provide any guarantee that the Specifications are completely free of error. Elephantech shall not be held responsible for any damage that is suffered by a customer as a result of any incorrect information that may be included in these Specifications.

- Any part of these Specifications is subject to change without prior notice for technical or quality improvement purposes, etc. Such being the case, please note that the information provided in these Specifications may not be perfectly consistent with the specifications of the particular product being used by each customer.

7. REVISION HISTORY

Ver.	Revision date	Description of revision
1.0	January 6th, 2017	Newly drafted and issued.
1.1	January 10, 2017	The data entry specifications, inspection items in the manufacturing specifications, comments on ink splash, soldermask printing specifications, and bending test data (for reference purposes only) were added.
1.2	January 26th, 2017	Supplementary information on the outline cutting and stiffeners was added.
1.3	January 30th, 2017	Information on the maximum instantaneous heat-resistant characteristic of the product was added. Other minor expressions, etc. were also edited.
1.4	February 19th, 2017	Data on the minimum hole diameter and outline-minimum pattern interval was added.
1.5	March 23rd, 2017	The maximum film thickness was changed from 20μm to 10μm.
1.6	April 4th, 2017	The soldermask material was changed to one with higher solvent resistance. Information on the copper foil thickness options was added.
1.7	April 24th, 2017	The copper foil thickness options were modified. The color of the soldermask was changed from green to transparent.
1.8	May 11th, 2017	The result of the product test conducted under conditions of 85°C and 85% was added to the Q&A section. The precautions on stiffener application were added to the data entry procedure.
1.9	May 17th, 2017	With regard to the specification of the copper foil thickness tolerance, change was made to guarantee the minimum foil thickness.
1.10	May 30th, 2017	The precautions to be followed at the time of data entry were added. The word “silk” was changed to “legend.”
1.11	June 23rd, 2017	The material of the soldermask was changed, and its color was also changed from transparent to green. Restrictions on the use of the product for medical devices, etc. were deleted from the disclaimers. Reference data on the anti-ionic migration characteristic of the product were added.

2.0	July 28th, 2017	The brand name was changed from AP-2 to P-Flex™. A note on burn traces that could be left after the laser cutting process was added to the precautions. Electroless Ni-Au plating was added to the surface treatment.
3.0	September 5th, 2017	The company name was changed from AgIC Inc. to Elephantech Inc.
3.1	September 6th, 2017	Typographical errors were corrected.
4.0	December 11th, 2017	As the standard substrate thickness was changed to 50μm, the following points have been revised. ● The standard substrate in the manufacturing specifications has been changed. ● Stiffeners have been changed. ● Explanation on the layer composition has been added. ● Reference data regarding flex resistance have been changed. ● Reference data regarding the minimum bending radius has been added. ● Reference data regarding bendability at 125μm has been deleted. ● Q&A on the flex resistance and bending resistance characteristics have been modifies. ● The minimum pattern interval of 150μm has been made available through consultation on a case-by-case basis. ● A copper foil thickness of 6μm has been made available through consultation on a case-by-case basis. ● The maximum allowable temperature for continuous product usage has been changed to 100℃. ● Reference data regarding ampacity have been added. ● An explanation on stiffeners has been added to the data entry specifications. ● An explanation on the outline data output has been added to the precautions for data entry. ● An explanation on the legend printing for using stiffeners has been added to the precautions for data entry. ● An illustration for explaining the minimum distance between the outline and circuit patterns has been added. ● The general dimensional tolerances have been modified.

		• The tolerance for out-of-bounds circuit patterns has been modified. • The tolerance for missing circuit patterns has been modified to include any pits and pattern floatage. • The tolerances for missing circuit patterns, pits, and pattern floatage have been modified. • The tolerance for misalignment between circuit patterns and matching legends has been newly specified. • The dimensional tolerance for the outline size has been newly specified.
4.0.1	February 8th, 2018	The temperature during continuous use is corrected.
4.2.0	June 25th, 2018	• Mention of RoHS and REACH accordance regarding standard specifications • DXF, PDF, and CADLUS added to accepted file formats • Added explanation for recommended design method of soldermask opening • Updated number of bends (R=5) for which operation has been tested to 20 million bends • Mention of reference values for the adhesiveness of copper foil • Modified thickness of gold plating to 0.03 μm • Modified the gold plating process regarding soldermask opening • Modified the gap between the outline component and the pattern to 0.3 mm • Modified the pattern legend for parts without soldermask to “Do not print” • Modified general dimensional tolerance • Modified outline dimensional tolerance • FAQ was deleted and unified to FAQ from the WEB
5.0.0	July 23rd, 2018	• Added PI specification • Corresponds to data entry by drawings with instructions • Added FR-4 stiffener types
5.0.1	August 10th, 2018	• Modified the sheet resistance • Added the method for measuring copper foil thickness. • Modified the tolerance between the outline and any circuit pattern • Corrected typographical errors

		• Added that, if the distance between pads are 0.7mm or less, the soldermask apertures must be fully open. • Added that, even if no soldermask is needed by a customer, it must still provide data specifying where the pad areas are so that tests can be performed on them using a flying probe tester. • Added that no drill data is accepted in usual cases, and that data on any holes that must be created should be also included in the outline data. • Updated the illustration explaining the layer structure as the silver layer was previously missing.
5.1.0	October 10th, 2018	Following the establishment of the Design Guideline, the items below were moved to the Design Guideline • The data submission specification needed when placing an order • Clearance information concerning each data • Explanation concerning the broadening of the soldermask • Explanation on overlapping legends and pads • A portion of the explanation on stiffeners • A portion of the explanation on connector parts Other • Modified minimum hole diameter to 0.5mm • Added explanation on waterproof soldermask • Modified thickness of PI stiffeners • Added reference data for flex resistance of PI
5.1.1	February 18th, 2019	• Added description of the copper being etched by approximately 0.5 μm for gold plating • Added wording on the theoretical thickness of waterproof soldermask • Added information on double-sided tape • Added wording on disposal method • Added wording on the temperature and humidity used as the measurement standard • Corrected errors in Tables 3-3 and 3-5
5.2.0	May 14th, 2019	• Deleted wording on the availability of 6μm copper foil (please consult us if you require a thickness of more than 3 μm) • Added wording on the availability of silkscreen printing for

		soldermasks • Changed legend color from black to white • Added electromagnetic wave shield film to available stiffeners • Regarding the outline, the option to select a high precision cut has been removed. The conventional high precision cut now comes as standard.
5.2.1	July 30th, 2019	• Changed the soldermask's typical thickness to 30 μm. • Added wording on the availability of white silkscreen soldermask coating. • Deleted data for the flex resistance following changes to soldermask thickness.
5.3.0	2February 10th, 2020020/02/10	• Explanation was added on L/S=200/150 μm only being available for PET substrate • Changed to the registered trademark symbol following the trademark registration of P-Flex • The color of the soldermask was changed from green to transparent • Explanation was added on the availability of PI coverlay
5.3.1	March 3rd, 2020	• Added an explanation on coverlays in the section covering soldermasks
5.3.2	June 11th,2020	• Figure 3-1 has been changed due to the standardization of the solder resist color
5.4.0	August 5th, 2020	• Expanded panel size to 388 x 226mm • Modified soldermask theoretical thickness • Modified total theoretical thickness in accordance with the above • Modified color of the soldermask in the figure