P-Flex®

Flexible Printed Circuit Board (Flex PCB)

P-Flex Design Guideline

Ver. 1.3.0

Last updated on: August 06, 2020

TABLE OF CONTENTS

1. OUTLINE OF MANUFACTURING SPECIFICATIONS

Table 1-1: 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 × 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 area, film stiffeners can be used to adapt to connectors of 200 μm or 300 μm in total thickness. Component mounted areas 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.

2. DATA ENTRY SPECIFICATION

2.1. Data Format

All data entry should generally be in the extended Gerber format (RS-274X), but we also accept other data formats as shown in **Table 2-1**. Please contact us individually if you would wish to use other data formats.

Table 2-1: Accepted data formats

Data format
Extended Gerber (RS-274X)
ODB++
Illustrator
DXF
CADLUS
Others, instructions in PDF and such

2.2. Data List

The types of data that are required for data entry are as listed in **Table 2-2** below.

As round holes are also cut out using laser, Elephantech does not use drill data. Therefore, customers must include data on any holes in the outline data.

Table 2-2: Types of data to be provided

Data type
Pattern data
Soldermask data, coverlay data (if necessary)
Pad data (if soldermask data or coverlay data is unavailable)
Legend data (if necessary)
Outline data (mandatory)
Stiffener data (if necessary)

2.3. Precautions to be followed for data entry

1. Elephantech recommends choosing 0.1 mm as the width of the outline. If the width of the outline is 0 mm, it may be omitted during the data output, depending on the CAD software specifications, thereby preventing the creation of proper cut data in some cases.
2. For the soldermask and coverlay data, please send us the negative pattern data (data showing only areas without soldermask print).
3. If soldermask data or coverlay data is unavailable, the pad areas for performing open/short tests by a flying probe tester cannot be identified. Therefore, even if a

customer does not require any soldermask, data equivalent to soldermask data specifying the pad areas is required.

3. LIST OF CLEARANCE

The clearance between each element is as shown in Table 3-1. The design must comply with these specifications. In parentheses are figures for high accuracy specifications.

Table 3-1 Clearance list

Unit : mm	Copper pattern	Soldermask/Coverlay aperture	Cut line	Legend
Copper pattern	0.2 (0.15)			
Soldermask/Coverlay aperture	0.2	0.2		
Cut line	0.3	-	1.0	
Legend	-	0.5	-	-

4. COPPER PATTERN

4.1. When to round the corners and avoid right angles

When bent, Flexible PCBs are prone to problems such as the tearing and peeling of the pattern in angular areas of the wiring. In order to avoid such problems, round the corners and avoid right angles in the corner areas of the wiring, the corners of pads, and when changes are made to the land (circular or rectangular portions of the pattern used for component mounting areas) or the width of the wiring. These measures must be applied to strengthen the flex resistance, especially where bending is repetitive.

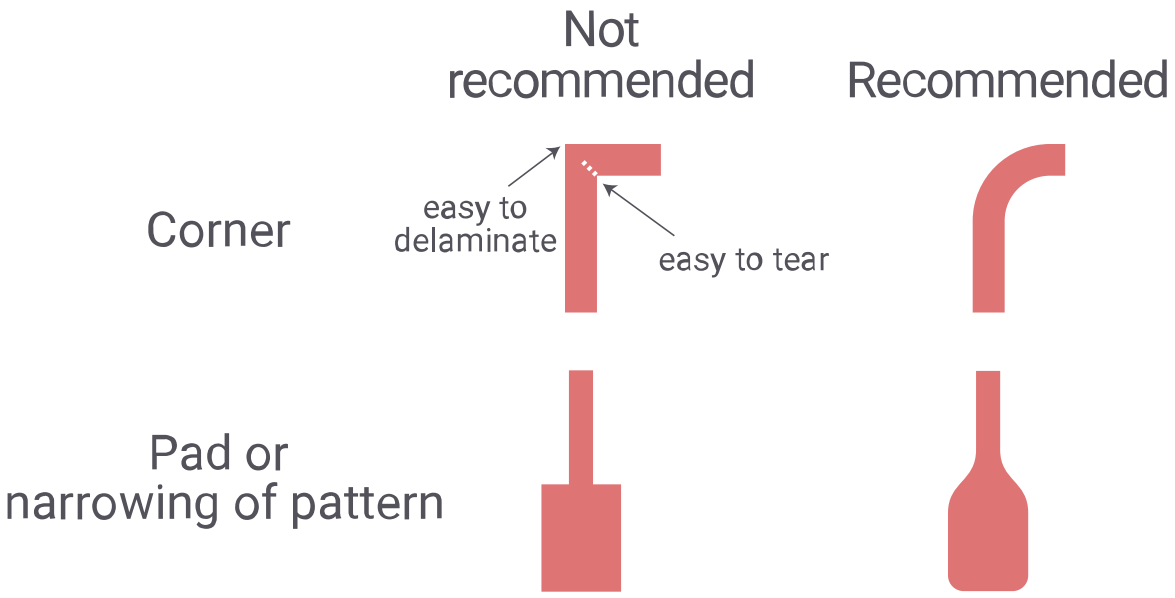

Figure 4-1 Good and bad examples of the shape for corners in wiring and areas of changing wiring width and land

4.2. No use that is highly dependent on the wiring resistance value

With the Pure Additive™ manufacturing method which we have developed, there may be some unevenness in the copper thickness due to the wiring being formed by electroless copper plating. We guarantee that, having passed various inspections, the copper thickness exceeds the specified values, but we do not recommend usage that demands high stringency from the resistance values of the wiring. With P-Flex, the copper thickness generally tends to be thinner the wider the wiring pattern is.

Also, for the same reasons, there can be irregularities in the appearance of the copper pour but there are no issues with normal usage as it is guaranteed to exceed the specified copper thickness.

5. SOLDERMASK/COVERLAY

Soldermasks and polyimide coverlays are generally used for flexible PCBs. At Elephantech, we use soldermasks for PET substrate P-Flex. Specifically, our method prints soldermask ink by UV inkjet printer and enables maskless manufacturing.

In addition, although PI coverlay is used for PI substrate P-Flex, the same soldermask rules apply for precision and the following design guidelines.

The role of the soldermask or coverlay is 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 manual or reflow soldering. 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.

When designing soldermask, a ±0.2mm soldermask tolerance must be considered and attention must be paid to always separate soldermask end from pattern edge by at least 0.2mm and the opening must be wide open when there is less than 0.6mm between pads.

Specifically, attention must be paid to the following 4 points.

5.1. 0.2mm minimum margin for soldermask apertures on the inside of copper patterns

When setting the soldermask aperture on the inside of the copper pattern, the design must maintain a margin of at least 0.2mm from the edge of the copper pattern. This is due to there being a ±0.2mm tolerance in soldermask.

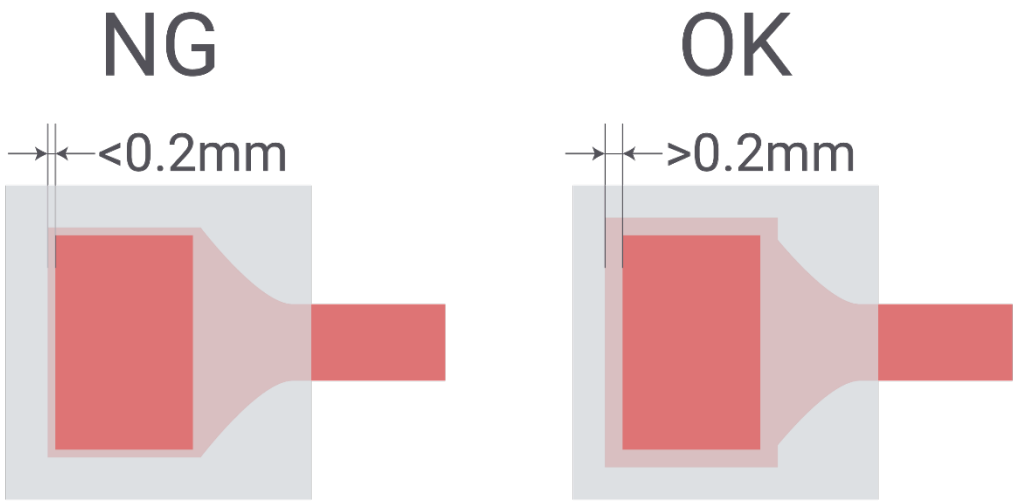

Figure 5-1 Soldermask aperture on the inside of copper patterns

5.2. 0.2mm minimum margin for soldermask apertures on copper pattern exteriors

As with when on the inside, a margin of at least 0.2mm is also necessary when the soldermask aperture is to be set on the outside of the copper pattern.

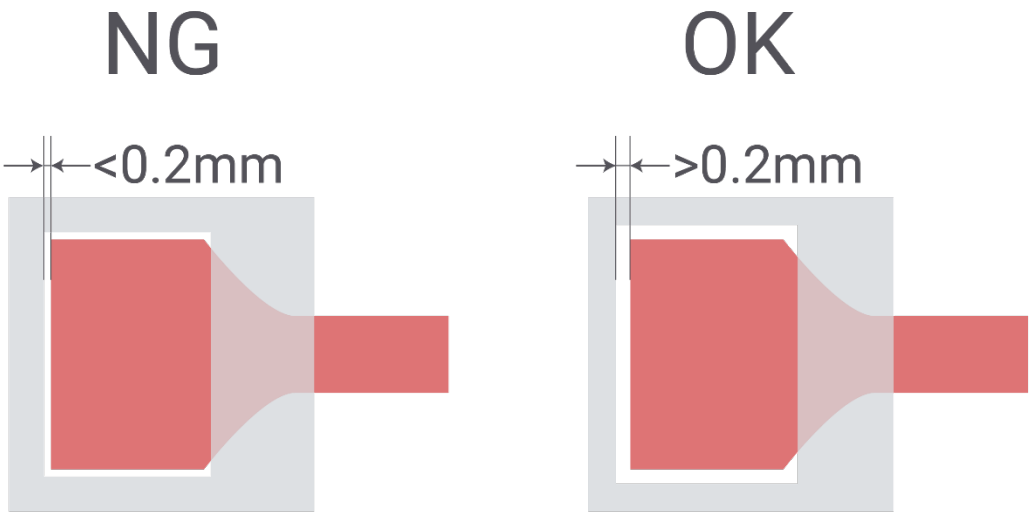

Figure 5-2 Soldermask apertures on the outside of copper patterns

5.3. Leave a margin of at least 0.2mm when in proximity of other copper wiring areas

Apart from the pad, a margin of at least 0.2mm is also necessary when in close proximity to copper wiring areas to avoid unintentional exposure of the copper wiring.

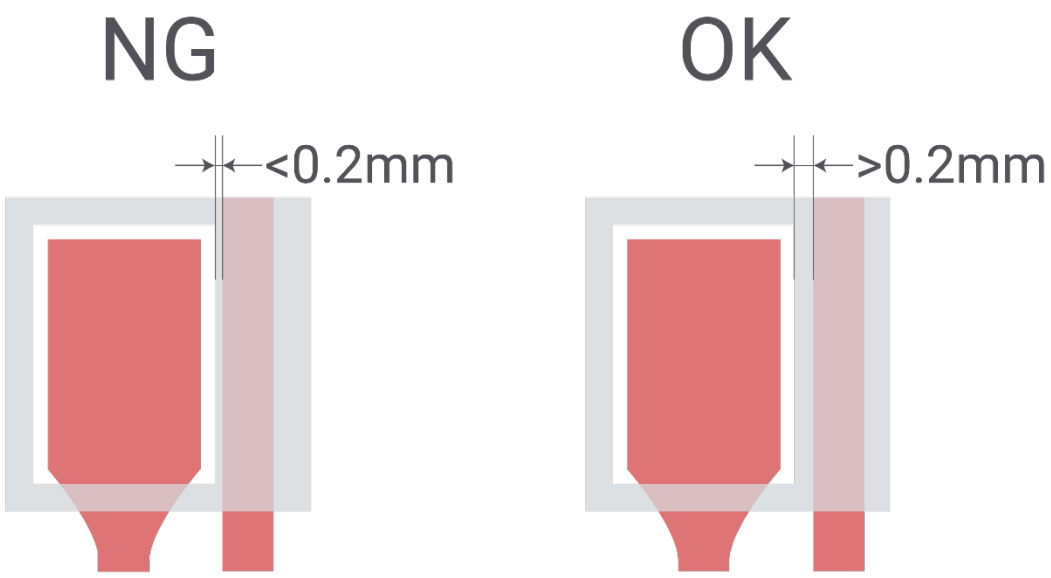

Figure 5-3 When in close proximity of other copper wiring areas

5.4. Wide apertures shall be used when pad spacing is under 0.6mm

Soldermask areas under 0.2mm won't print properly. For this reason, soldermask cannot be placed between pads and wider apertures are necessary when pad spacing is less than 0.2mm.

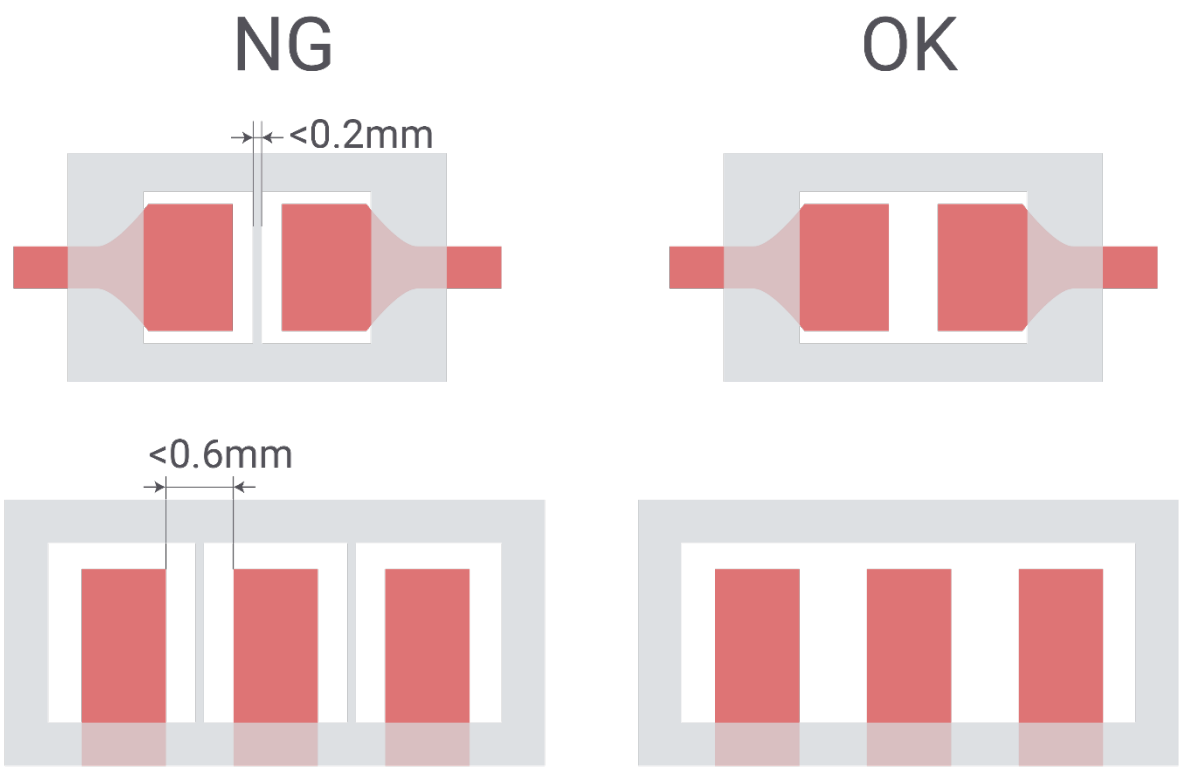

Figure 5-4 Eliminating soldermask under 0.2mm

6. OUTLINE

6.1. Outline specifications

The minimum line width/minimum line spacing for copper patterns is 0.2mm/0.2mm (0.2mm/0.15mm for high precision specification), the minimum spacing between copper patterns and outlines or holes is 0.3mm, the minimum outline width is 1.0mm, and the minimum hole diameter is 0.5mm. Please consult us if you require cuts to copper foil areas.

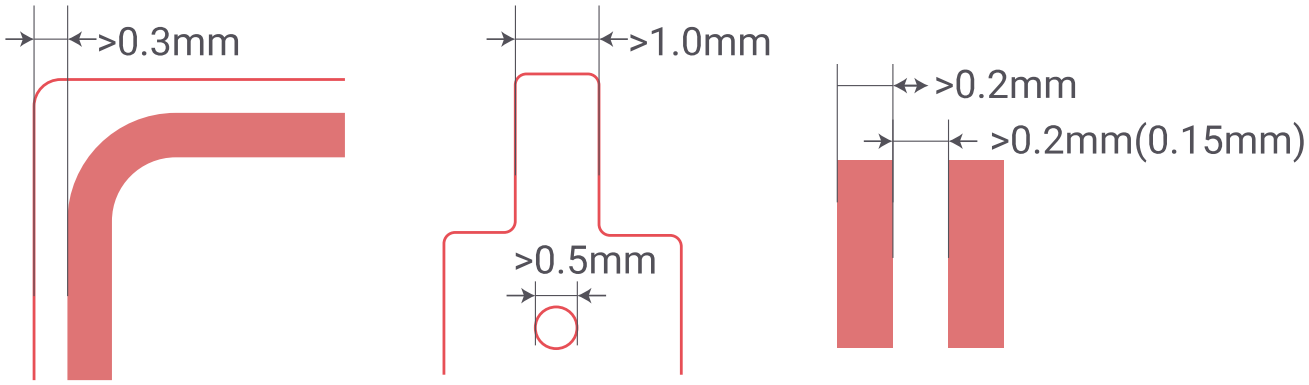

Figure 6-1 Copper patterns and outlines

6.2. Recommendations concerning the form of the outline

Due to the nature of flexible PCB, the substrate is susceptible to rips and tears from the bending and stretching of the outer shape of the film. In order to avoid such problems, hard angles must be avoided in the outer shape and the shape must be rounded or changed gradually. Also, to strengthen the substrate's resistance to tears, adding a copper pattern as a reinforcing band is effective.

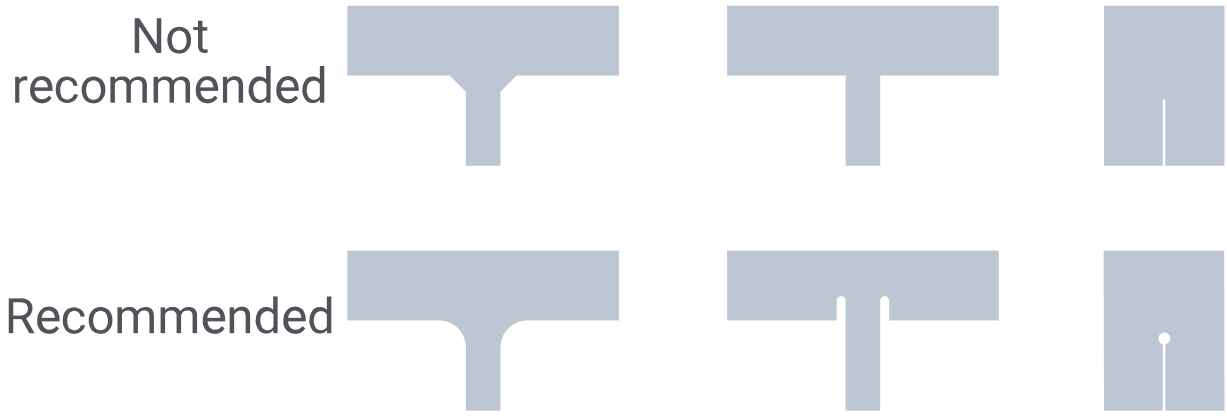

Figure 6-2 Examples of recommended and discouraged outline shapes

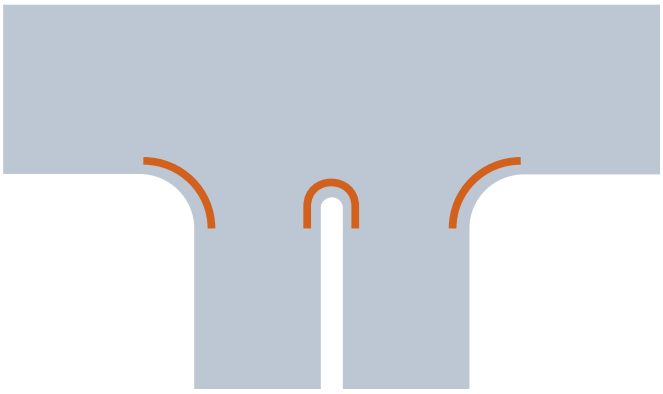

Figure 6-3 Example of reinforcement by copper pattern

7. LEGEND

Also known as “silk-screen printing”, identifying legends are printed to aid assembly work. In principle, Elephantech only uses white for the color of its legends.

Also, as there is a strong possibility that mounting failure will arise during parts assembly if the legend overlaps with the pad (area where the copper pattern and the soldermask aperture overlap), legends cannot be placed within a 0.5mm perimeter of the pad as shown in Figure7-1. However, this does not limit specific instructions such as wanting to intentionally overlap legends and pads.
Although the ink used to print the legends are flex resistant, we do not recommend placing legends on areas that are particularly prone to repetitive bending.

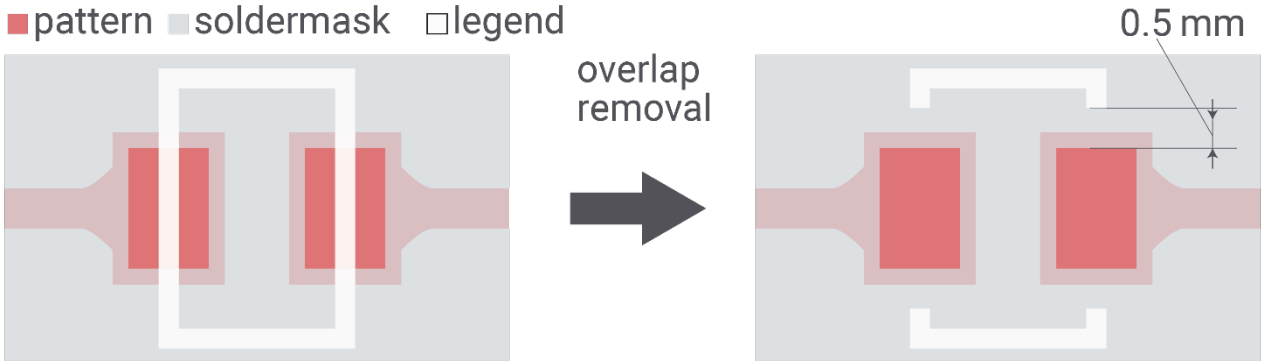

Figure 7-1 Overlapping legend and pad

8. CONNECTOR SECTIONS

8.1. Regarding FPC connectors

Concerning FPC connectors, connectors up to a minimum of 0.4mm pitch are supported.

Essentially, designing in keeping with the recommended dimensions of the FPC connector shall be sufficient, but modifications must be made so as to maintain more than 0.3mm of separation between the outer shape and the pattern. Also, the soldermask must not be placed between pads and the aperture must be wide open.

As for the FPC connector to be used, one with a locking mechanism must be used. Breakage on the film side is more likely when using FPC connectors without a locking mechanism due to repeated insertion and removal.

Regarding the thickness of the FPC connector section, the thickness dimensions are specified for each connector, typically 200μm or 300μm with a tolerance of ±30μm. For thickness adjustment of the connector section, we can manufacture within the tolerance of the connector's thickness dimensions if so specified on the stiffener.

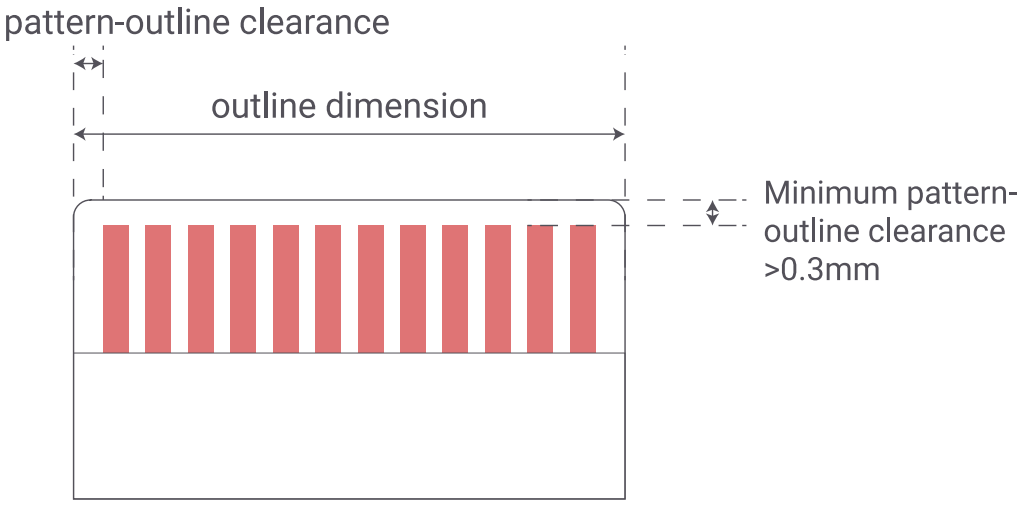

Figure 8-1 Connector section dimensions

8.2. Regarding board-to-board connectors

The board-to-board connector is mounted on the both side of PCBs and allows for an electronical connection between boards by fitting them parallel to each other. Attention must be paid to the following as well as the standard parts assembly precautions.

When using board-to-board connectors, hard material stiffeners such as FR-4 must always be used on the back side of the flexible PCB side's connector section. As a guideline, we strongly recommend FR-4 boards of at least 0.5mm in thickness. The board will bend and break when removing the connector if the stiffener is weak as the pattern will peel off from the pad. Always maintain the 2 boards at parallel when attaching or detaching as, even if the stiffener is sufficiently strong, the connector will fall off from the board if the boards are twisted or broken apart during detachment.

9. STIFFENERS

9.1. Regarding stiffeners of connector sections

Regarding the stiffeners of connector sections, please refer to the explanations in 8. Connector sections.

9.2. Regarding stiffeners for areas other than connector sections

Other than on FPC connector sections, stiffeners are used to reinforce the sections used for parts mounting. On flexible PCBs, all sections used for mounting must be reinforced with stiffeners. Although it is possible to use stiffeners for mounting sections with PI film or PET film, we strongly recommend FR-4 boards of at least 0.5mm in thickness as a guideline. If the stiffener for the mounting sections isn't sufficiently rigid against bending, parts will peel off of the board as the stiffener bends along with the board during post-mounting handling.

The minimum stiffener width shall be 5mm for stiffeners other than for FPC connector sections. Also, it must be created in a shape that touches at least 2 sides, and more than 3 sides if possible of the outline. Unexpected breakage of the film may occur on the extremities of the stiffener if the stiffener is designed in a shape that doesn't touch the outer shape.

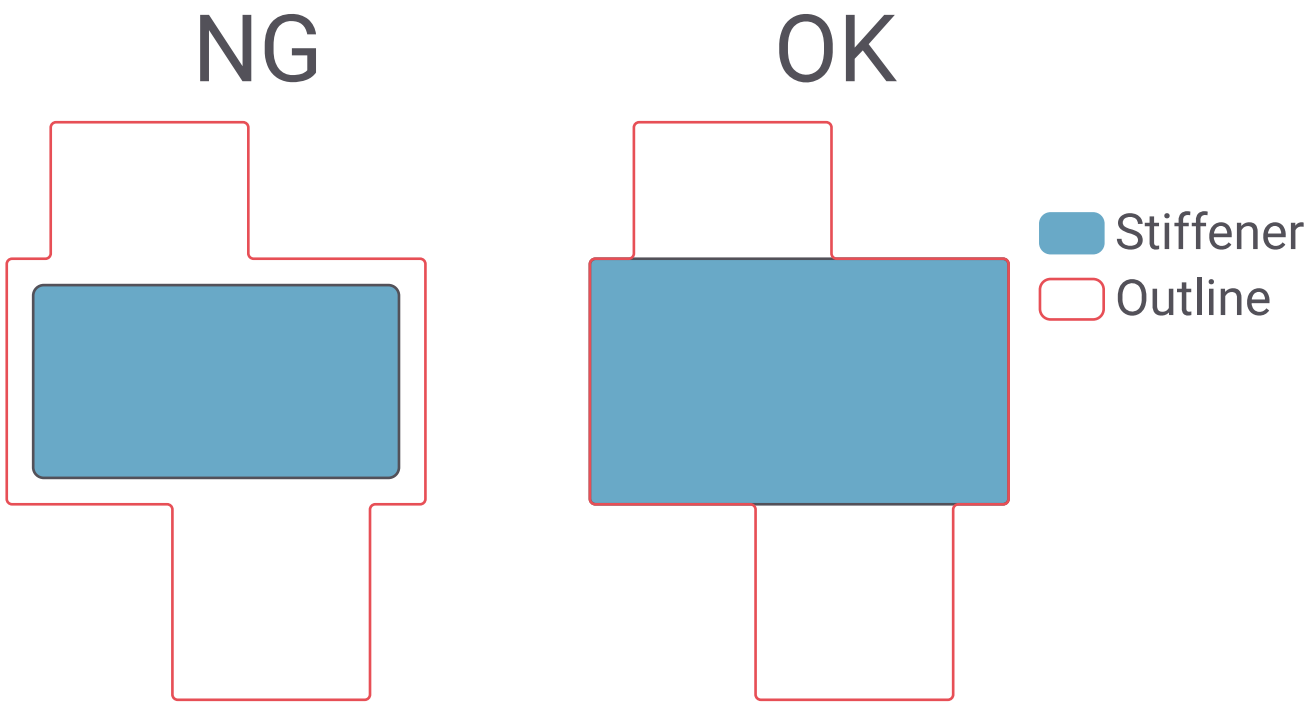

Figure 9-1 Stiffener and outline

10. BENDING SECTION

Below is a summary of the cautionary points regarding bending sections. Bending here is defined as the bending of the board when seen from the cross section of the board as shown in the figure. Flex resistance is an important parameter for flexible PCBs, and flex resistance changes greatly depending on the bending sections and the patterns around it, the outer shape, and the design of the stiffener.

Figure 10-1 Bending as seen from the side

Caution to the patterns of bending section are, as stated in 4.1. Use gentle curves and apply corner roundness when changing the wiring width such as in the corner areas of the wiring or the edges of the pads, and the following must be implemented:

1. Avoid bending the wiring (apply corner roundness when bending)
2. Do not change the wiring width (add a gradual fillet if changing wiring width)

Do not mount any parts on the bending section. Also, if the parts mounted section and the stiffener are close to the bending section, special attention must be paid to the mechanical design so as no “fold” type bends occur on the edges of the stiffener. The wiring will break from extremely few repetitive bends if a “fold” type bend occurs in a frequently bent section.

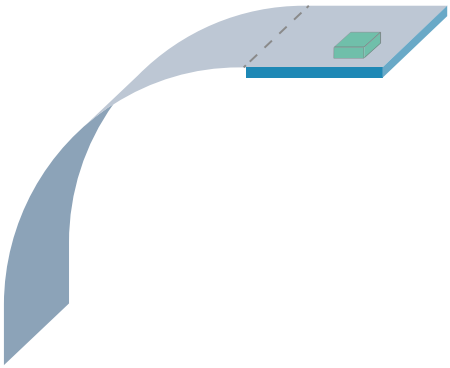

Figure 10-2 Example of a bend with ample margins to prevent folding on the stiffener’s edges

11. 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 and such. 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.

12. REVISION HISTORY

Ver.	Revision date	Description of revision
1.0.0	October 29, 2018	Newly drafted and issued.
1.0.1	January 31, 2019	Replaced image
1.1.0	May 14, 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.
1.1.1	July 30, 2019	• Added explanation on soldermask • Added wording on the availability of white silkscreen soldermask coating. • Rectified the unmodified sections of the explanation for the outline-pattern interval with regards to the unification of the high precision cut.
1.2.0	February 10, 2020	• 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 • Fixed typo
1.2.1	March 3, 2020	• Added wording on ODB++ compatibility • Added an explanation on coverlays in the section covering soldermasks
1.3.0	August 6, 2020	• Expanded panel size to 388 x 226mm • Modified color of the soldermask in the figure • Added explanation on the flex resistance of the ink used for

		legends