Flexible Printed Wiring Board (PWB)
P-Flex Design Guideline

Ver. 1.0.0

Last updated on: October 29th, 2018

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 / interval	200/200 μm min., 200/150 μm min. (option)
Hole diameter	1.0 mm min.
Outline-pattern interval	Standard Precision: 0.5 mm min. High Precision: 0.3 mm min.
Copper foil thickness	3 μm, (6 μm-thick copper foil may be also applied on a case-by-case basis. Please consult us to see if this option is available.)
Panel size	180 × 270 mm max.
Wiring layer	Single-sided
Solder mask application	UV inkjet printing (green)
Legend printing	UV inkjet printing (black)
Surface treatment	Oxidation prevention treatment, Electroless nickel gold plating (option)
Outline trimming	Laser cutting
Hole processing	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. More solutions upon consultation.
Inspection	Optical 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)
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 circular holes are also cut out using laser, Elephantech does not accept orders with drill data. Therefore, customers must include data on any holes that must be created on the product surface, in the outline data.

Table 2-2: Types of data to be provided

Data type
Pattern data
Soldermask data (if necessary)
Pad data (if soldermask data is not available)
Legend data (if necessary)
Outline data
Stiffener data (if necessary)

2.3. Precautions to be followed for data entry

1. Elephantech strongly 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 data, please send us the negative pattern data (data showing only parts without soldermask print).
3. If soldermask data is not available, the pad areas for performing open/short tests using a flying probe tester cannot be identified. Therefore, even if a customer does not require any soldermask, data similar to soldermask data specifying the pad areas must be sent to us.

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 opening	Cut line	Symbol
Copper pattern	0.2(0.15)			
Soldermask opening	0.2	0.2		
Cut line	0.5(0.3)	–	1.0	
Symbol	–	0.5	–	–

4. COPPER PATTERN

4.1. Use gentle curves and apply corner roundness when there are changes to the wiring width such as in the corner parts of the wiring or on the edges of the pads

When bent, Flexible PWBs are prone to problems such as the tearing and peeling of the pattern in angular parts of the wiring. In order to avoid such problems, corner roundness must be applied to the corner parts of the wiring, and the edges must be rounded off and the wiring must be changed gradually in areas where there are changes to the land (circular or rectangular portions of the pattern used for connecting parts) or wiring width. These measures must be applied to strengthen the flex resistance, especially where bending is repetitive.

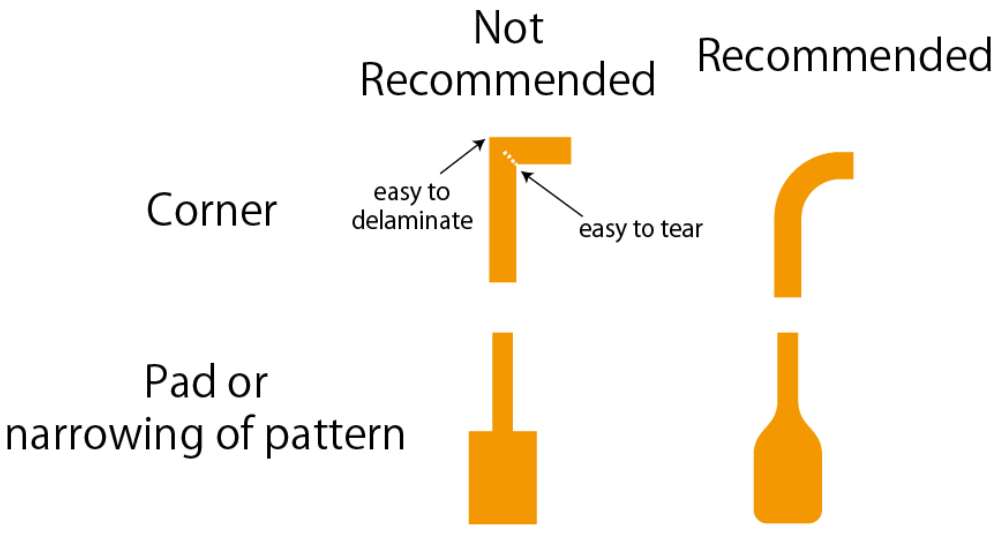


Figure 4-1 Good and bad examples of the shape for corners in wiring and parts of variable wiring width and land

4.2. Do not use in ways that require high stringency from the resistance values of the wiring

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 disapprove 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 solid surface but there are no issues with normal usage as it is guaranteed to exceed the specified copper thickness.

5. SOLDERMASK

When designing soldermask, a ±0.2mm soldermask tolerance must be taken into account and attention must be paid to always separate soldermask end from pattern start 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. Maintain a margin of at least 0.2mm for soldermask openings on the inside of copper patterns

When setting the soldermask opening 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.

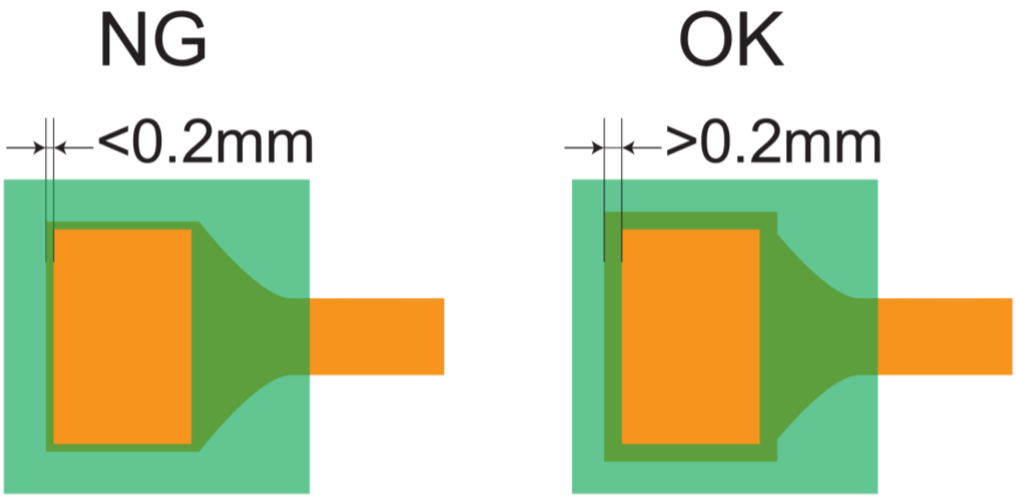


Figure 5-1 Soldermask opening on the inside of copper patterns

5.2. Maintain a margin of at least 0.2mm for soldermask openings on the outside of copper patterns

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

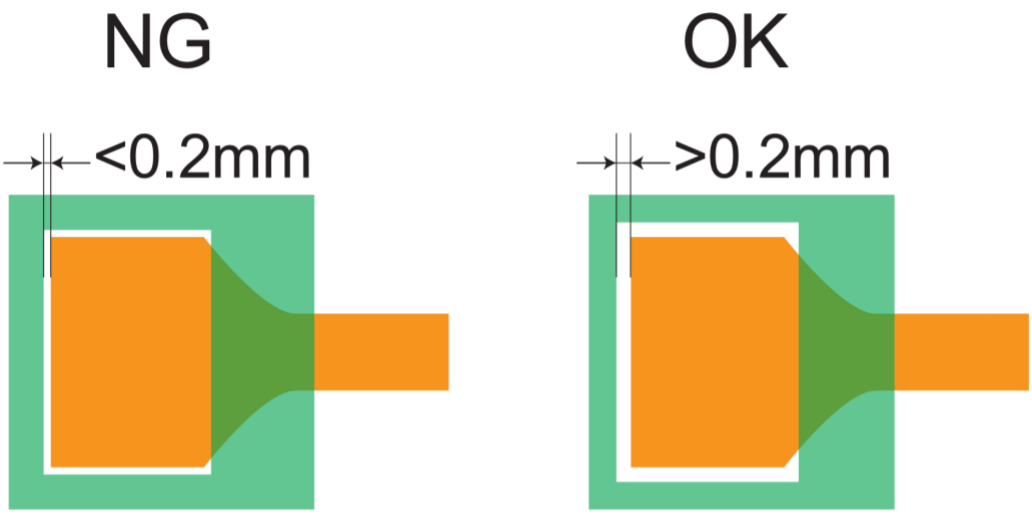


Figure 5-2 Soldermask openings on the outside of copper patterns

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

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

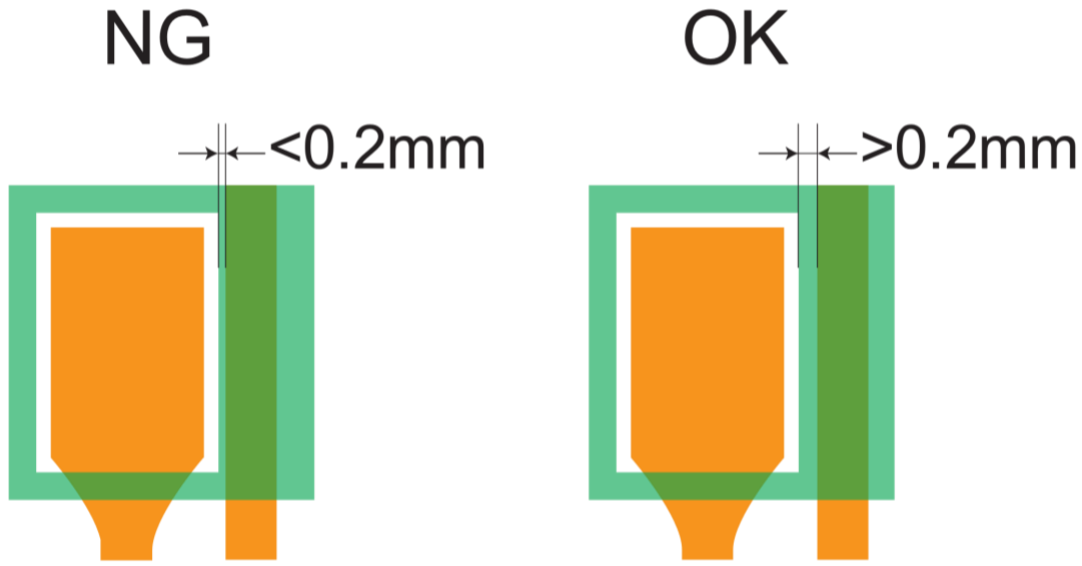
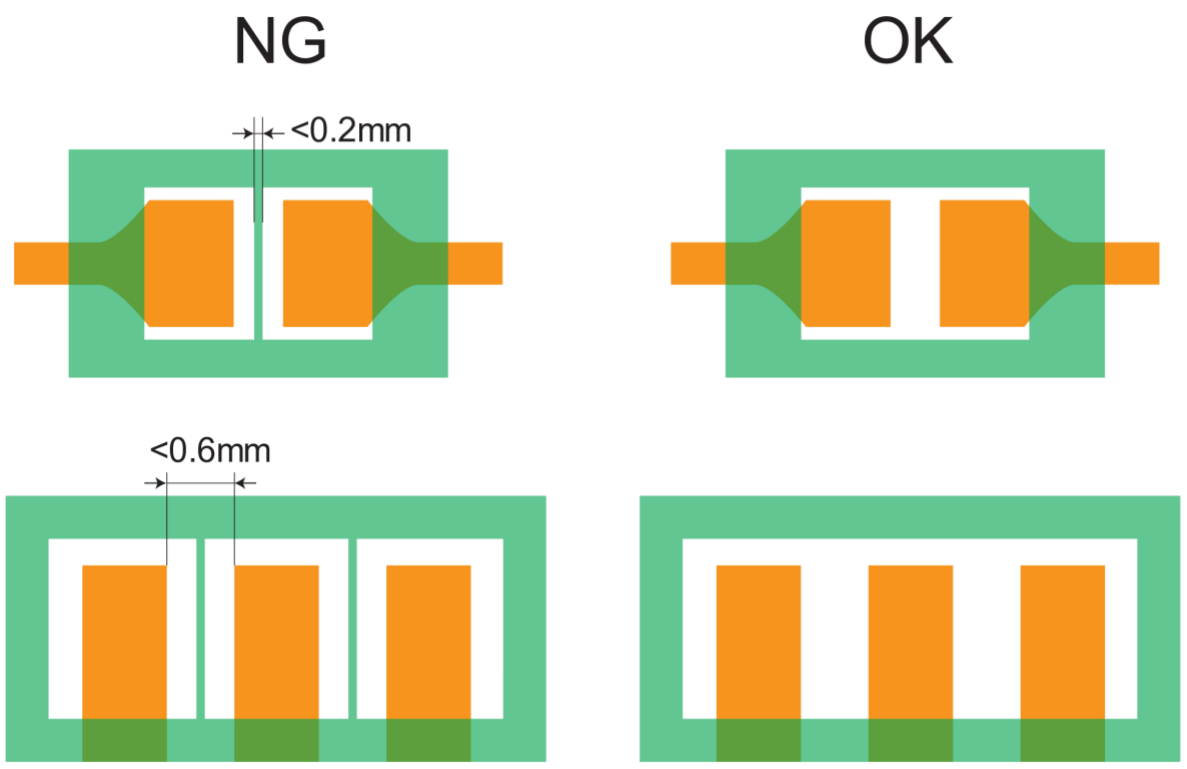


Figure 5-3 When in close proximity of other copper wiring parts

5.4. Wide openings shall be used when pad spacing is under 0.6mm, eliminating resist under 0.2mm

Resist parts under 0.2mm won't print properly. For this reason, resist cannot be placed between pads and wider openings are necessary when pad spacing is less than 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 0.5mm (0.3mm for high precision specification), the minimum outline width is 1.0mm, and the minimum hole diameter is 0.5mm.

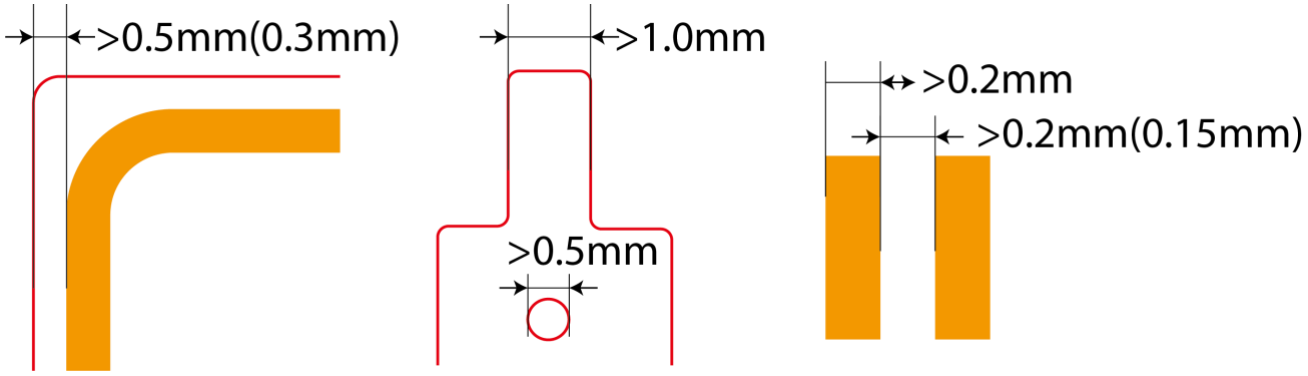
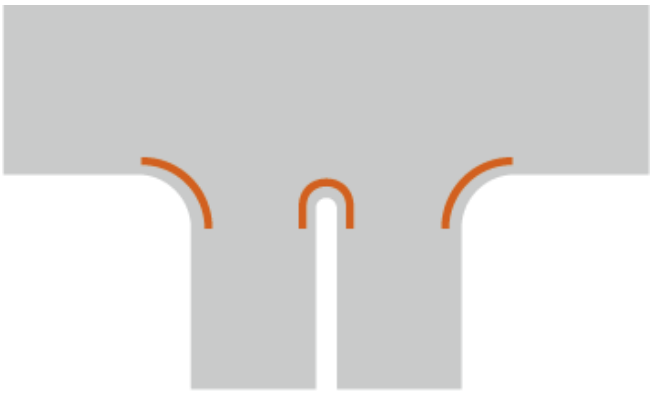


Figure 6-1 Copper patterns and outlines

6.2. Recommendations concerning the form of the outline

Due to the nature of flexible PWB, 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.



Copper pattern for support

7. SYMBOL

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

Also, as there is a strong possibility that mounting failure will arise during parts assembly if the symbol overlaps with the pad (area where the copper pattern and the soldermask opening overlap), symbols 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 symbols and pads.

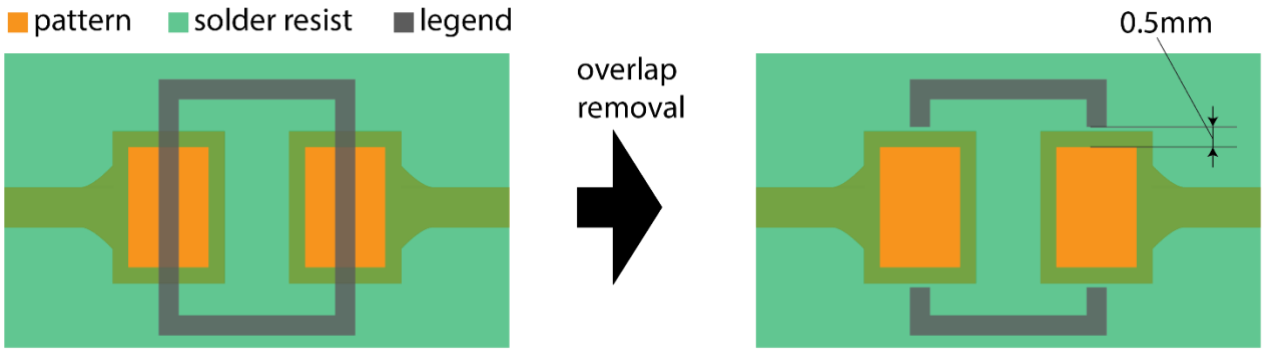


Figure 7-1 Overlapping symbol and pad

8. CONNECTOR PARTS

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

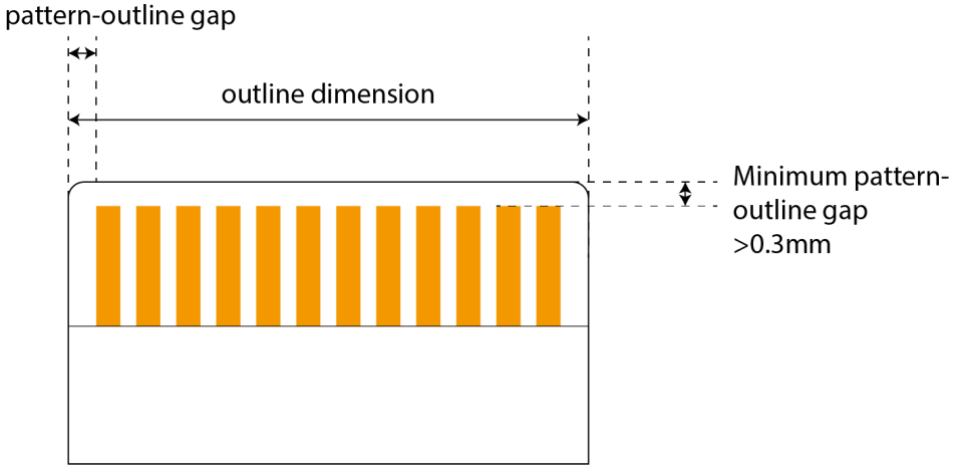


Figure 8-1 Connector part dimensions

8.2. Regarding board-to-board connectors

The board-to-board connector assembles the parts to be paired with the connector and the 2 connecting boards

and allows for an electronical connection between boards by fitting them in such a way that they are 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 PWB side's connector part. 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 parts

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

9.2. Regarding stiffeners for parts other than connector parts

Other than on FPC connector parts, stiffeners are used to reinforce the parts used for parts assembly. On flexible PWBs, all parts used for assembly must be reinforced with stiffeners. Although it is possible to create stiffeners for assembly parts 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 assembly parts isn't sufficiently rigid against bending, parts will peel off of the board as the stiffener bends along with the board during post-assembly handling.

The minimum stiffener width shall be 5mm for stiffeners other than for FPC connector parts. 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.

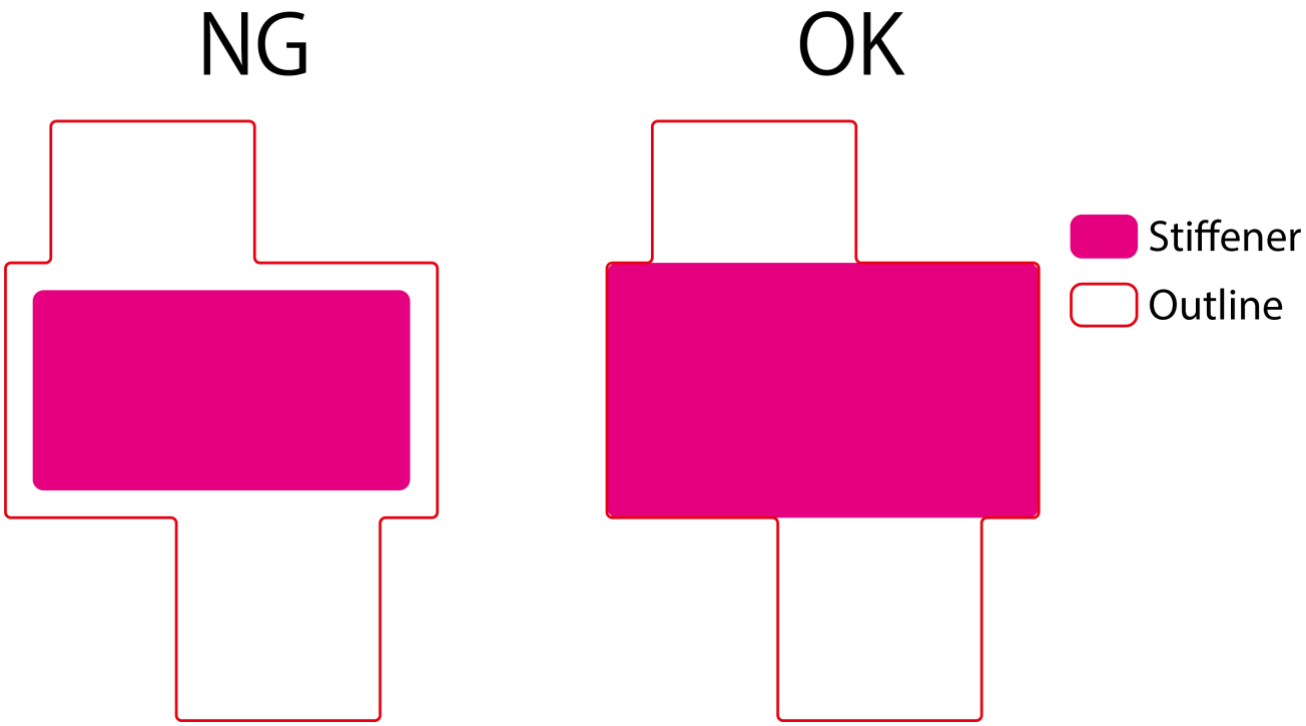


Figure 9-1 Stiffener and outline

10. BENDING PARTS

Below is a summary of the cautionary points regarding bending parts. 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 PWBs, and flex resistance changes greatly depending on the bending parts and the patterns around it, the outer shape, and the design of the stiffener.

Figure 10-1 Bending as seen from the side

Things to pay special attention to with regards to the patterns of bending parts are, as stated in 4.1. Use gentle curves and apply corner roundness when changing the wiring width such as in the corner parts 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)

Also, if the parts assembly section is close to the bending part without there being any parts assembled on the bending part and the stiffener is close, 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 part.

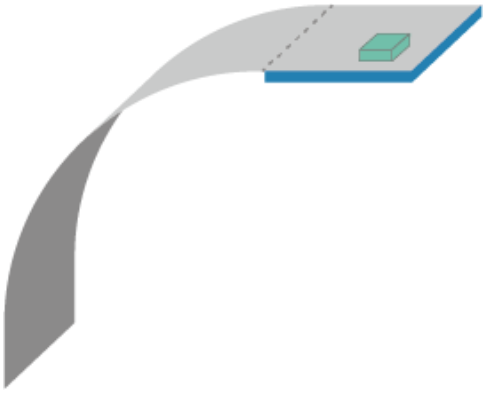


Figure 10-2 Example of a bend with ample margins so as to prevent folding on the stiffener's edges

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 29th, 2018	Newly drafted and issued.