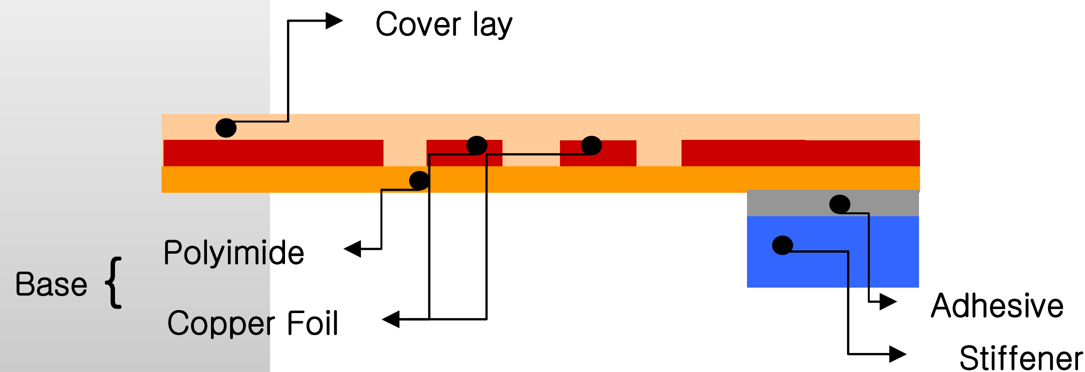
Flexible Circuit Simple Design Guide

INDEX

- Flexible Circuit Board Types and Definitions
- Design Guides and Rules
- Process Flow
- Raw Material

Single Side Flexible PCB

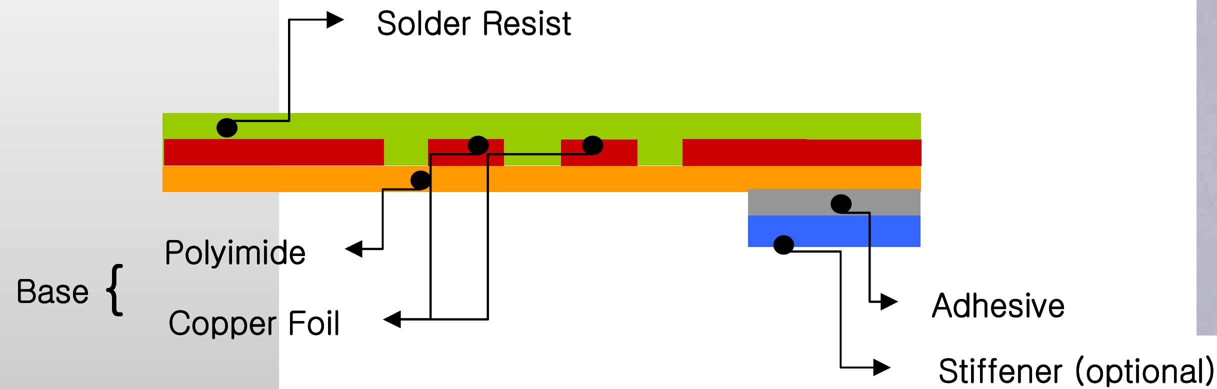
Single Side Flexible PCB (Cover layer type)

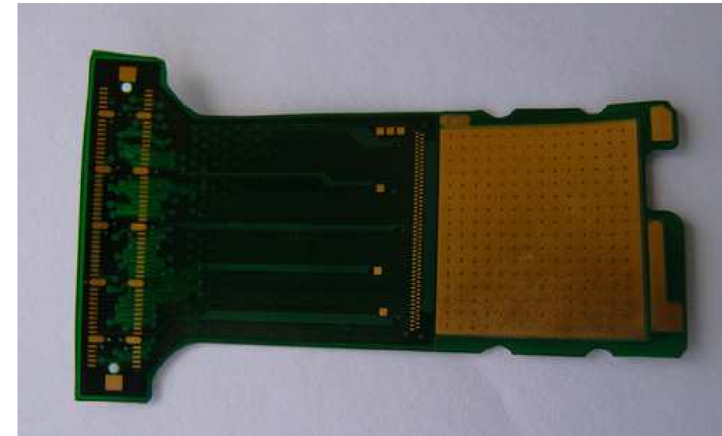


Layer Structure				SPEC (Unit:㎛)	Remark
1 Layer	FPC	Cover Lay	Polyimide	12.5	1/2Mil (Adh: 22.5~25㎛)
			Adhesive	22.5	
		FCCL	Copper	18	1Mil ½ oz, RA or ED S/S (Adh: 10~15㎛)
			Adhesive	10	
			Polyimide	25	
		Double Side Tape	Adhesive	50	Adh: 50~55㎛
		Stiffener	Epoxy	100	Epoxy, Kapton, Polyester

Single Side Flexible PCB

Single Side Flexible PCB (Solder resist type)



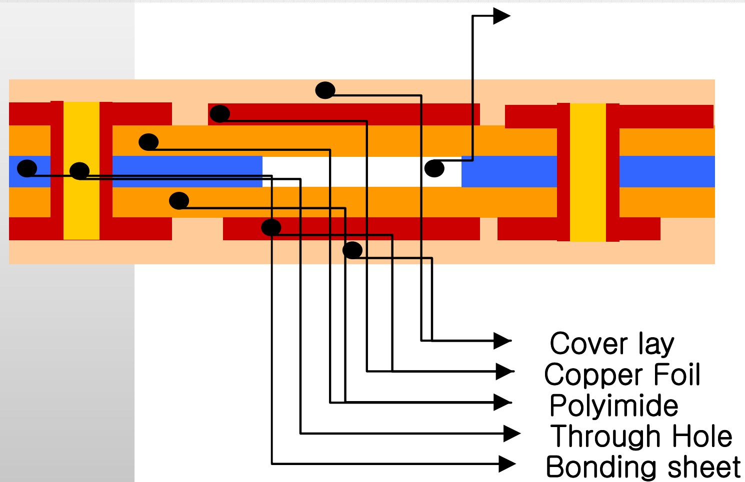


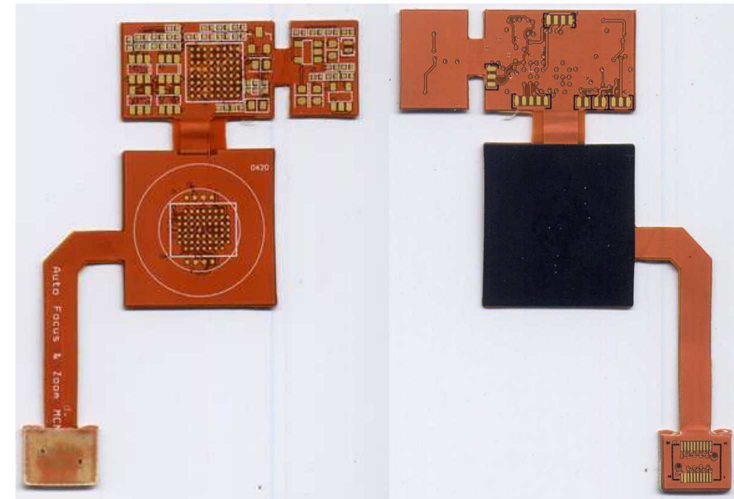
Layer Structure				SPEC (Unit:㎛)	Remark
1 Layer	FPC	Solder Resist	UV curable ink	17	15~20㎛
		FCCL	Copper	18	1Mil ½ oz, ED or RA S/S (Adh: 10~15㎛)
			Adhesive	10	
			Polyimide	25	
		Double Side Tape	Adhesive	50	Adh: 50~55㎛
		Stiffener	Epoxy	100	Epoxy, Kapton, Polyester

Advanced Electronic Technology
Printed Circuit Boards

Double Side Flexible PCB

Double (Folder) side with Air Gap

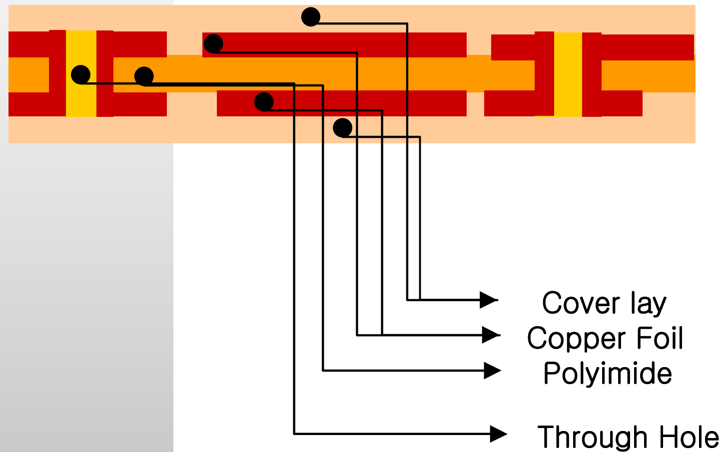


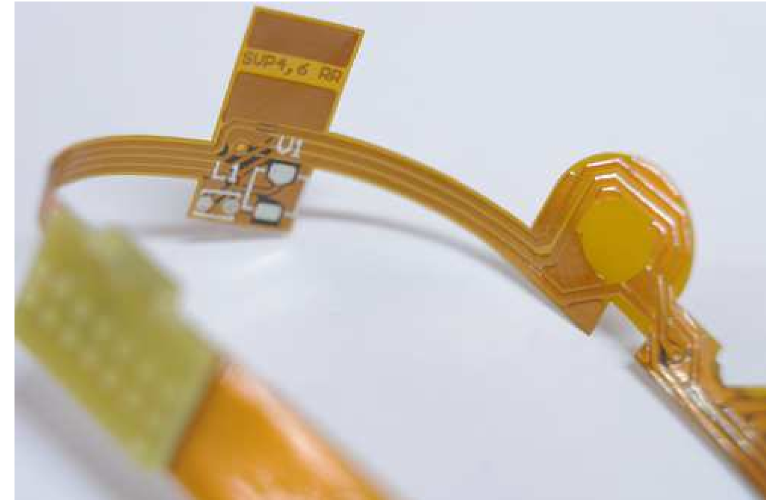


Layer Structure				SPEC (Unit:㎛)	Remark
2 Layer	FPC	Cover Lay	Polyimide	12.5	1/2Mil (Adh: 22.5~25㎛)
			Adhesive	22.5	
		FCCL	Copper	18	1Mil 1/2 oz
			Adhesive	10	RA, S/S
			Polyimide	25	(Adh: 10~15㎛)
		ADHESIVE	Epoxy	25	Epoxy/Acrylic
		FCCL	Polyimide	25	1Mil 1/2 oz
			Adhesive	10	RA, S/S
			Copper	18	(Adh: 10~15㎛)
		Cover Lay	Adhesive	22.5	1/2Mil (Adh: 22.5~25㎛)
			Polyimide	12.5	
		Total Thickness		201	

Double Side Flexible PCB

Double sided flexible PCB



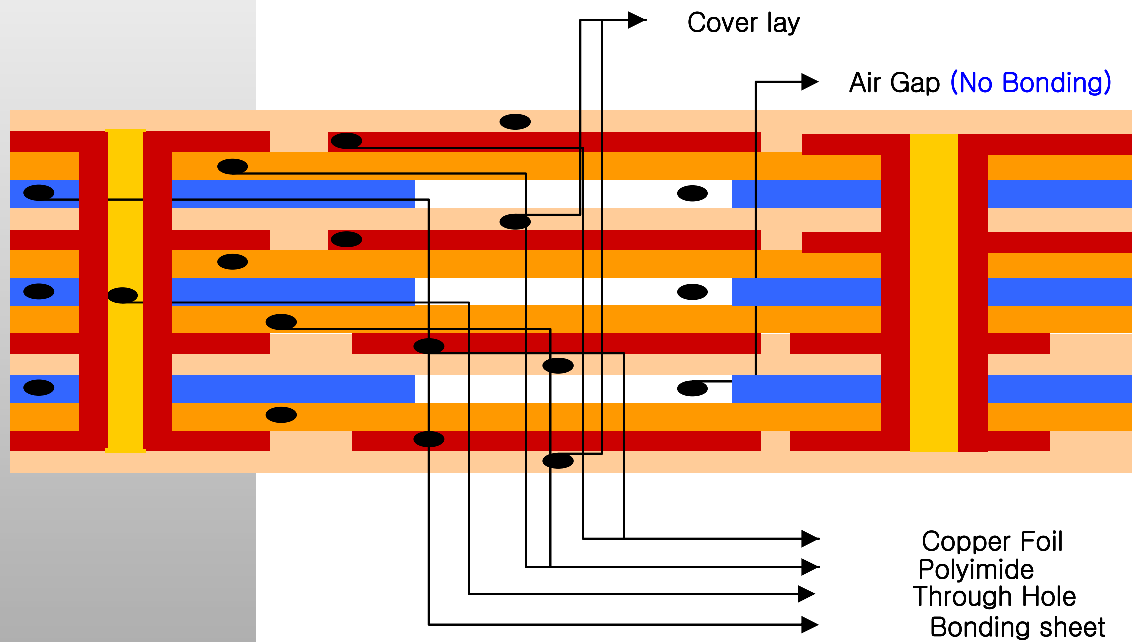


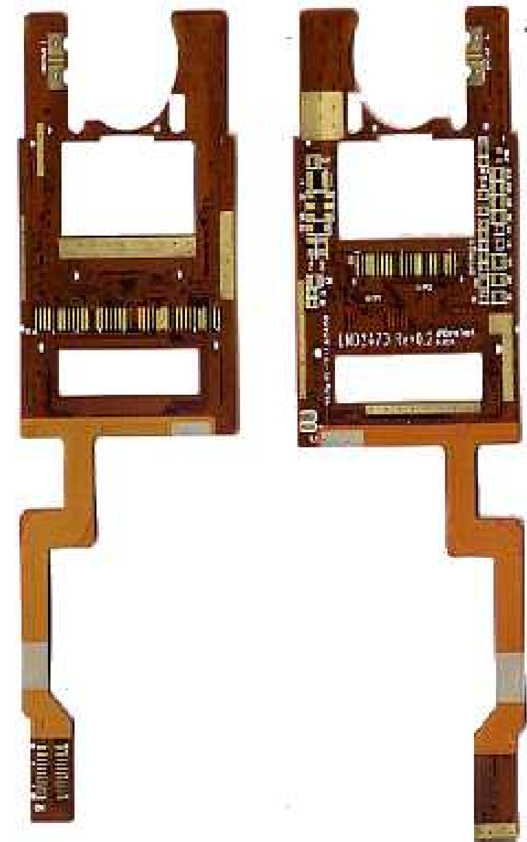
Layer Structure				SPEC (Unit:㎛)	Remark
2 Layer	FPC	Cover Lay	Polyimide	12.5	1/2Mil (Adh: 22.5~25㎛)
			Adhesive	22.5	
		FCCL	Copper	18	1Mil 1/2 oz ED or RA, D/S (Adh: 10~15㎛)
			Adhesive	10	
			Polyimide	25	
			Adhesive	10	
			Copper	18	
		Cover Lay	Adhesive	22.5	1/2Mil (Adh: 22.5~25㎛)
			Polyimide	12.5	
		Total Thickness		151	

Advanced Electronic Technology
Printed Circuit Boards

Multilayer Flexible PCB with Air Gap

See next page for details



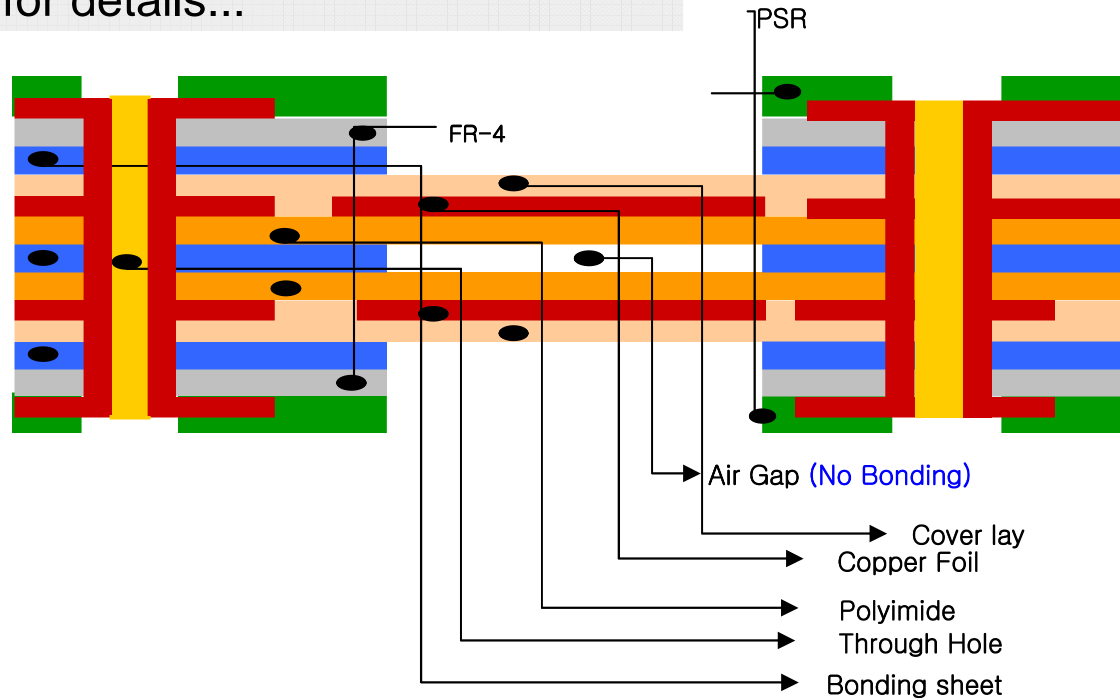


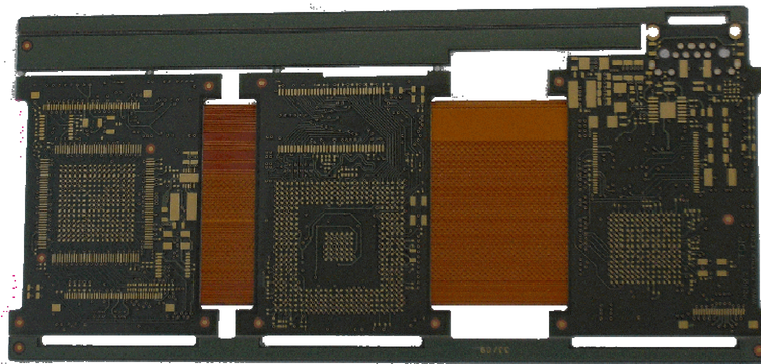
Multilayer Flexible PCB with Air Gap

Layer Structure				SPEC (Unit:㎛)	Remark
4 Layer	FPC	Cover Lay	Polyimide	12.5	1/2Mil (Adh: 22.5~25㎛)
			Adhesive	22.5	
		FCCL	Copper	18	1/2Mil 1/2 oz RA, S/S (Adh: 10~15㎛)
			Adhesive	10	
			Polyimide	12.5	
		Bonding Sheet	Epoxy	25	Epoxy/Acrylic
		Cover Lay	Polyimide	12.5	1/2Mil (Adh: 22.5~25㎛)
			Adhesive	22.5	
		FCCL	Copper	18	1/2Mil 1/2 oz RA, S/S (Adh: 10~15㎛)
			Adhesive	10	
			Polyimide	12.5	
		Bonding Sheet	Epoxy	25	Epoxy/Acrylic
		FCCL	Polyimide	12.5	1/2Mil 1/2 oz RA, S/S (Adh: 10~15㎛)
			Adhesive	10	
			Copper	18	
		Cover Lay	Adhesive	22.5	1/2Mil (Adh: 22.5~25㎛)
			Polyimide	12.5	
		Bonding Sheet	Epoxy	12.5	Epoxy/Acrylic
		FCCL	Polyimide	25	1/2Mil 1/2 oz RA, S/S (Adh: 10~15㎛)
			Adhesive	10	
			Copper	18	
		Cover Lay	Adhesive	22.5	1/2Mil (Adh: 22.5~25㎛)
			Polyimide	12.5	
		Total Thickness		377~407	Plating (20㎛)

Rigid Flex PCB with Air Gap

See next page for details...





Advanced Electronic Technology
Printed Circuit Boards

Rigid Flex PCB with Air Gap

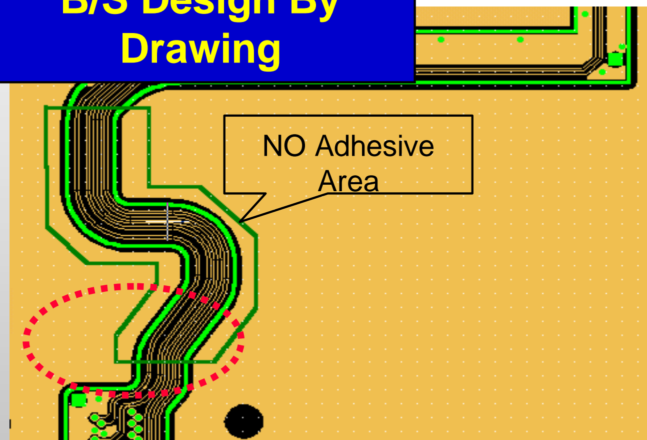
Layer Structure				SPEC (Unit:㎛)	Remark
4 Layer		Solder Resist ink	Green	20	Green/Black (UV)
		FR-4	Copper	18	0.1mm 1/2oz S/S
			Epoxy	100	
		Bonding Sheet	Epoxy	25	Epoxy/Acrylic
		Cover Lay	Polyimide	12.5	1/2Mil (Adh: 22.5~25㎛)
			Adhesive	22.5	
		FCCL	Copper	18	1/2Mil 1/2 oz RA, S/S (Adh: 10~15㎛)
			Adhesive	10	
			Polyimide	12.5	
		Bonding Sheet	Epoxy	25	Epoxy/Acrylic
		FCCL	Polyimide	12.5	1/2Mil 1/2 oz RA, S/S (Adh: 10~15㎛)
			Adhesive	10	
			Copper	18	
		Cover Lay	Adhesive	22.5	1/2Mil (Adh: 22.5~25㎛)
			Polyimide	12.5	
		Bonding Sheet	Epoxy	12.5	Epoxy/Acrylic
		FR-4	Epoxy	100	0.1mm 1/2oz S/S
			Copper	18	
		Solder Resist ink	Green	20	Green/Black (UV cured)
		Thickness		>= 490	Plating (25㎛)

Advanced Electronic Technology
Printed Circuit Boards

Design Guides and Rules

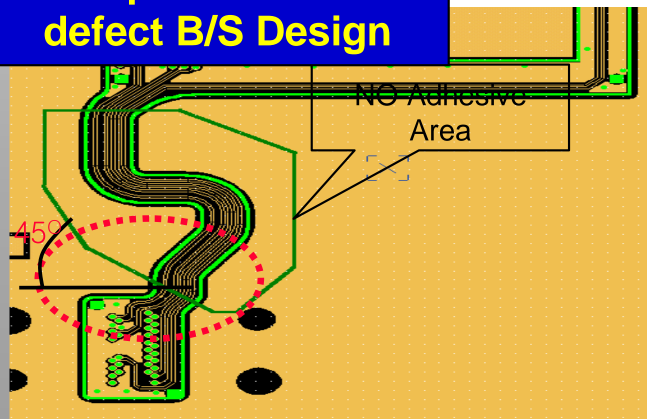
Bonding Sheet Design (Folder area)

B/S Design By Drawing



● If the border line type of NO ADHESIVE AREA part is vertical, it can cause SHORT or OPEN circuit problem.

For prevention of defect B/S Design

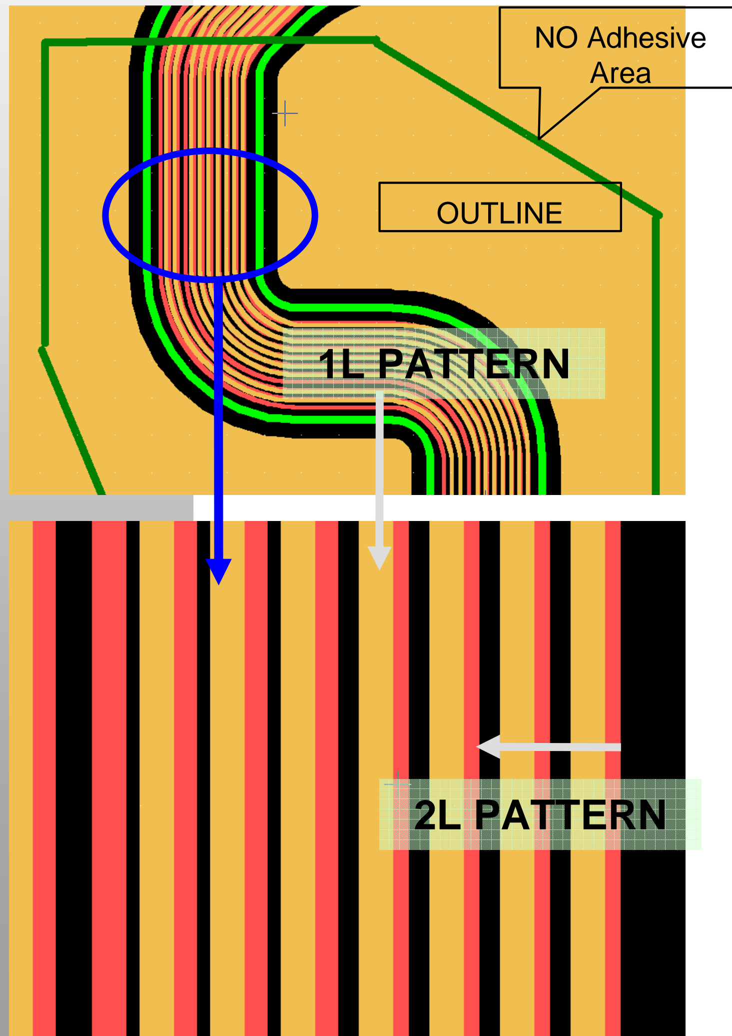


● **We prefer to design the Bonding Sheet in the manner that gives a slope of 45 degrees to one of the border lines of LCD mounting area or TAIL part.**

(Exception: Single sided type may not be affected)

Design Guides and Rules

Pattern design of Folder Area



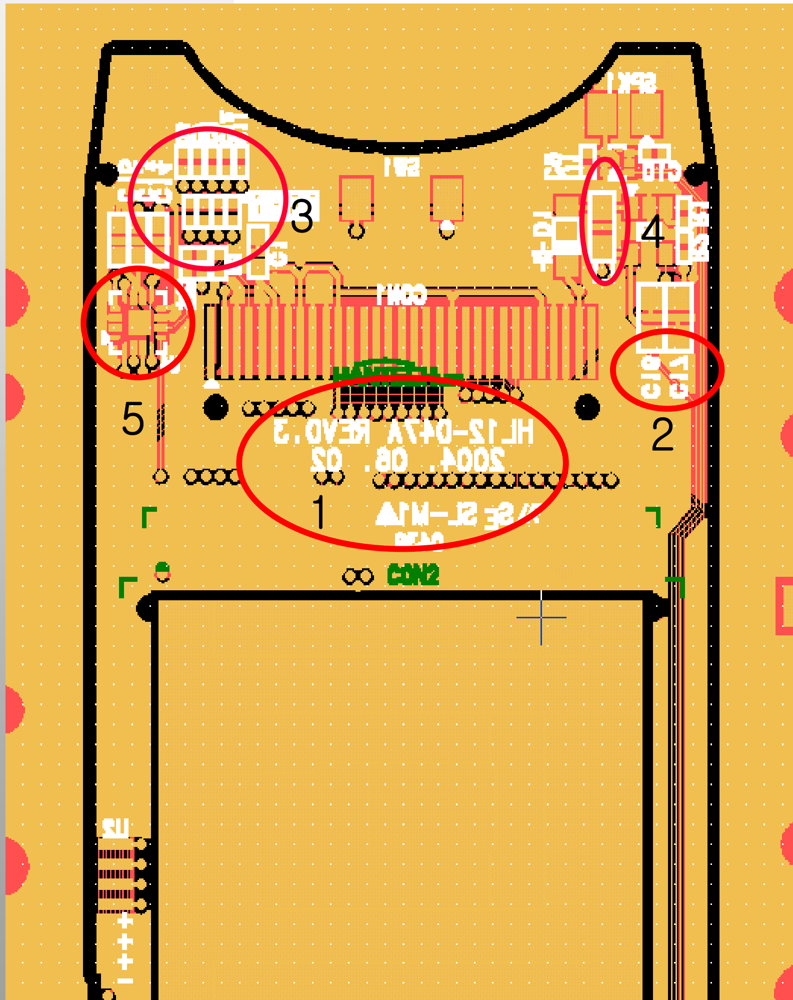
1) Purpose : To retain the maximum flexibility by staggering the pattern lines.

2) Method : ① Pattern lines on each layer will be staggered (as much as possible).
② Pattern lines on 1st and 2nd layer are staggered (Refer to diagram beside).
③ Pattern lines on 3rd and 4th layer are staggered (Refer to diagram beside).
④ As result, pattern lines on each layer can be staggered with each other.
⑤ This is to be considered for signal pattern line.

3) Reason : If the pattern of every Layer is located above the same line, it causes decrease of flexibility.

Design Guides and Rules

Silk Screen Specification



1) Purpose : To avoid possible default by understanding silk screen production condition.

2) Method :

-1 Text Mark : Customer's mark, Symbol, Date code;
=> SIZE 2mm

-2 Component Text Mark : Min. 0.7mm, Max. 1.5mm
→ We may shift the marking position depending on the situation (After discussion with the customer)

-3 Insulation Line : ① Line for preventing short circuit between the lands.
② Line thickness : 0.15mm (Standard)
③ Distance between line & land : 0.2mm

-4 Land Out-Line : ① Silk screening on the outer line of the land is of no use.
② Prefer to remove unless it is an insulation line (After discussion with the customer).

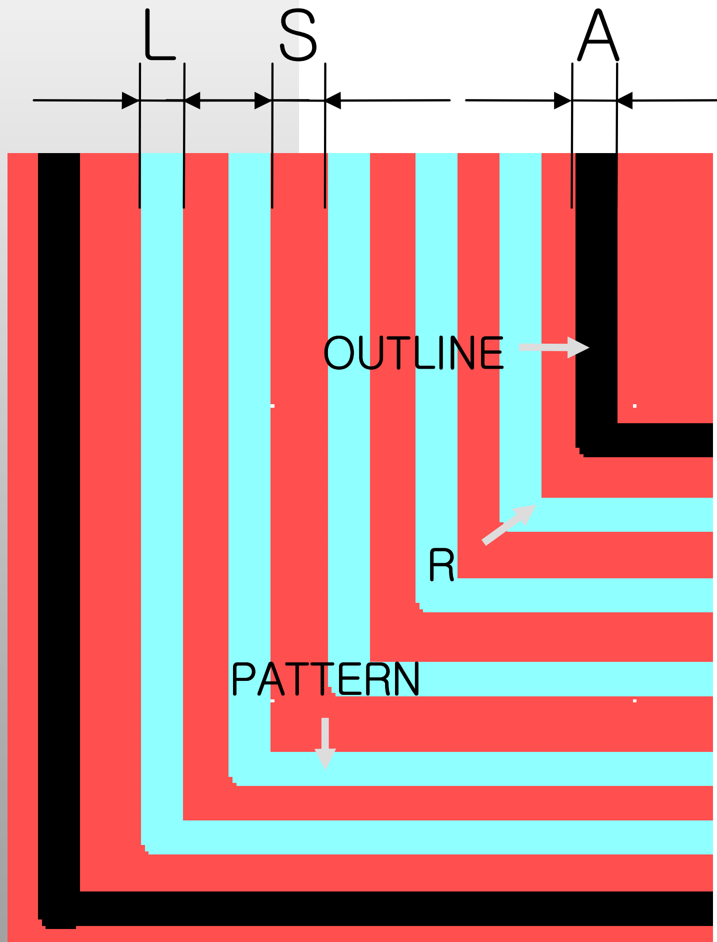
-5 Alignment Line : Follow the customer's request.

-6 Space : Min. 0.2mm between the lines.
If it is beyond specification, it will be shifted after approval from customer.

Advanced Electronic Technology
Printed Circuit Boards

Design Guides and Rules

Pattern line thickness & tolerance of raw material

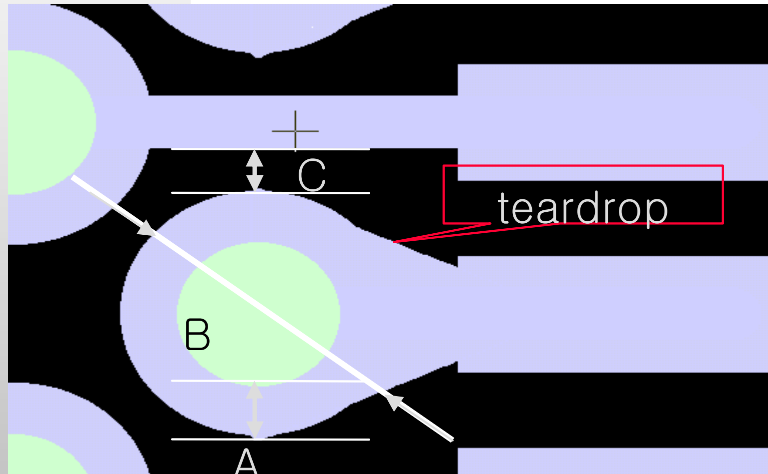


(Unit : mm)

Two layer type	L (Min. Line)	S (Space-Pattern/Pattern)	A (Space-Pattern/Border)	R (Min. Radius Value)
1/2 oz	0.075 (±10%)	0.075	0.3	0.2
1 oz	0.095 (±10%)	0.09	0.3	0.2

Design Guides and Rules

Through hole / Pad (Inside)



(Unit : mm)

	Mechanical CNC	Laser N.C
A	0.15	0.10
B	0.45	0.30
C	0.10	0.10

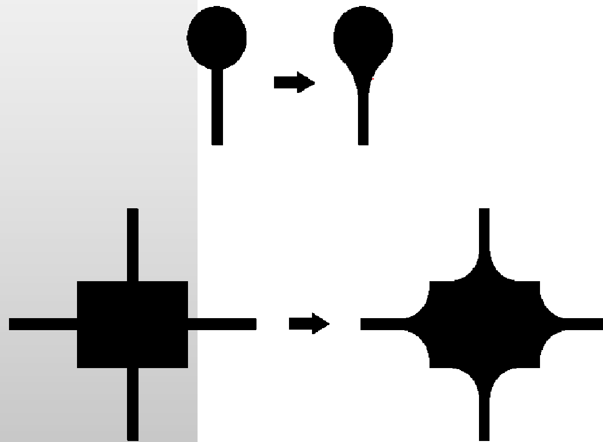
Stiffener / Tape Area Tolerance

(Unit : mm)

Type	Adhesive	Stiffener	
		Universal (Manually)	JIG type
Tolerance	±0.3	±0.3	±0.15

Design Guides and Rules

Tear-drop design



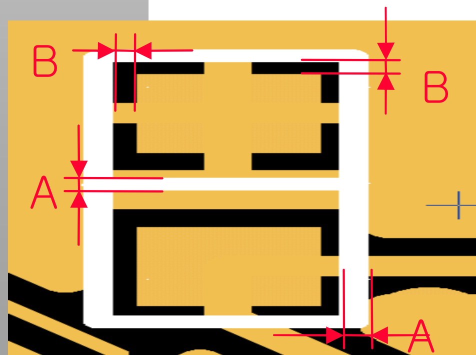
1) Purpose : Prevention of open circuits caused by cracks or disconnections between PAD and pattern.

2) Type : 1. Between Through-Hole PAD and pattern
2. Between lands and pattern

3) Method : Use tear-drop tool editor, input R (size).
In the case of impossibility of inputting R,
it should be input made manually

4) Exception : Area that can not be input as R will be worked manually (R value disregarded).

Silk Screen Tolerance

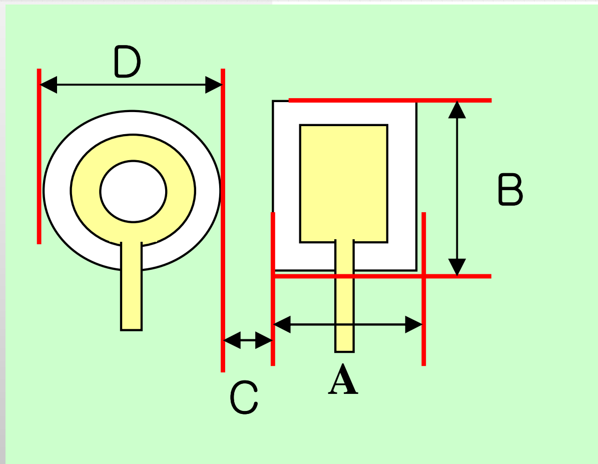


Item	Dimension
A (Min. Width Of Marking)	Min. 0.15 mm
B (Min. Distance from Land)	Min. 0.2 mm

Design Guides and Rules

Cover lay & Solder Resist Formative Dimension

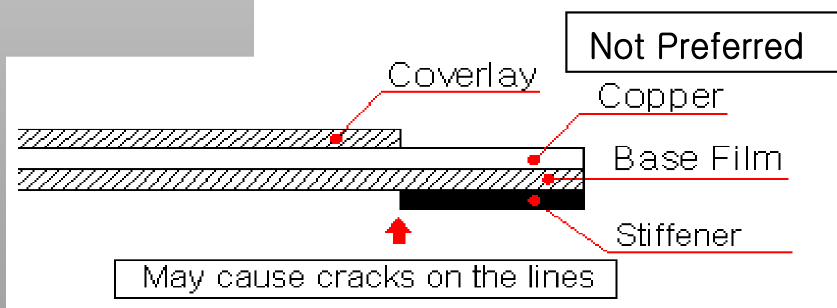
(Unit : mm)



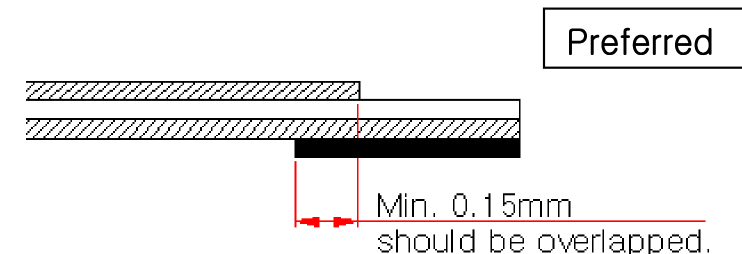
Item	Cover lay	Solder Resist
A (Min.Size of Rectangle)	0.5	0.2
B (Min.Size of Rectangle)	0.5	0.2
C (Min.Space between Openings)	0.5 (within1/3 of 'B')	0.1
D (Min. Dia. of Circle)	0.5	0.2

Cover lay Open Area Stiffener Specification

Not Preferred



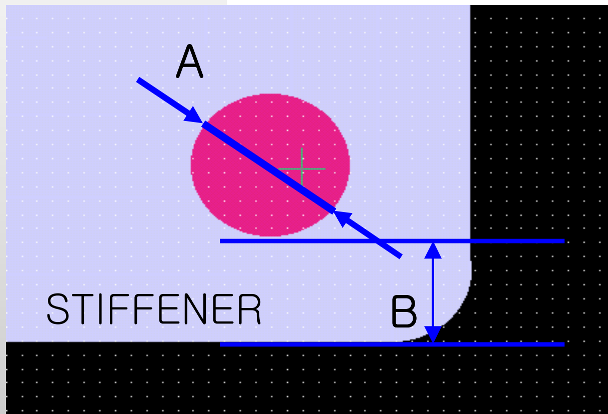
Preferred



Overlap Distance (Minimum 0.15mm)

Design Guides and Rules

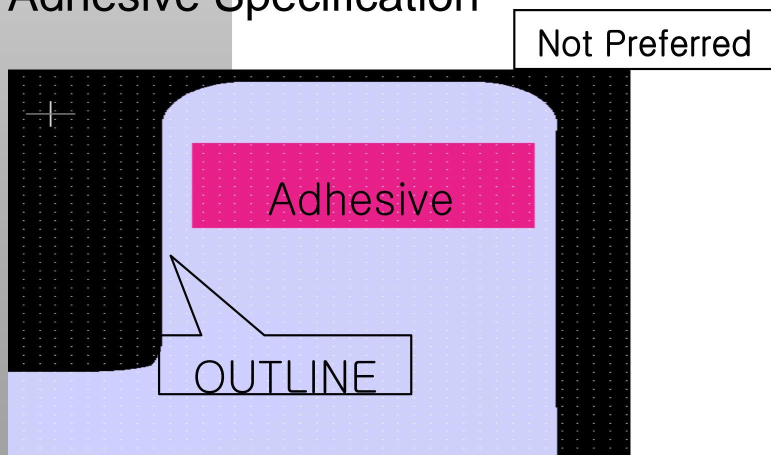
Gap from stiffener edge to Hole

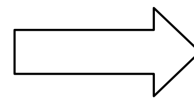


Item	Min. Size
A (Hole Size)	Φ A ≥ Stiffener Thickness
B (Distance between hole and edge)	B ≥ Stiffener Thickness (For prevention of cracks)

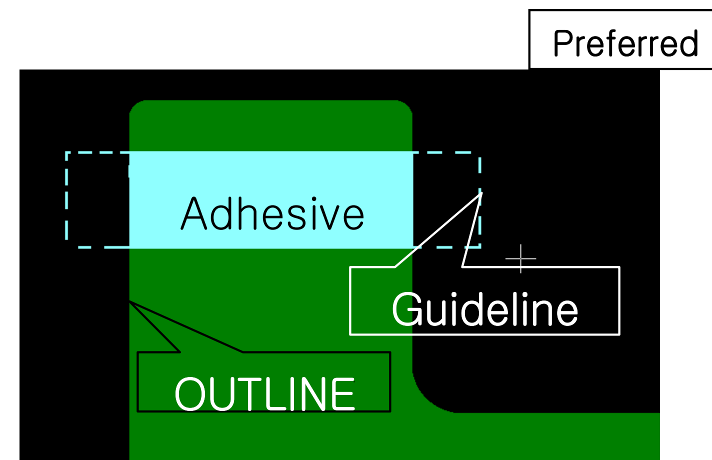
Adhesive Specification

Not Preferred





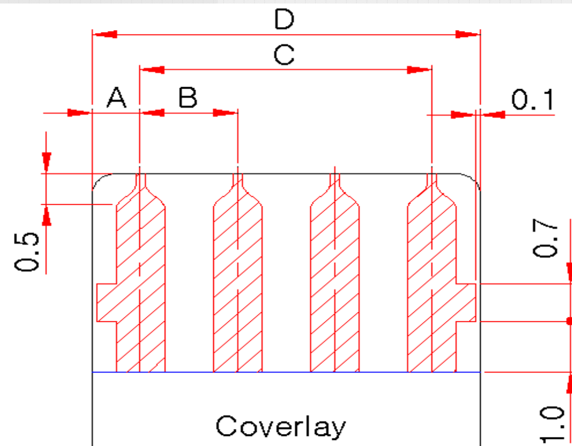
Preferred



Design Guides and Rules

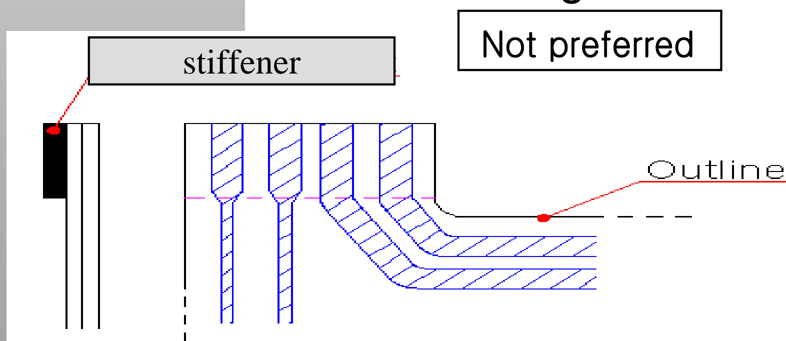
Goldfingers design

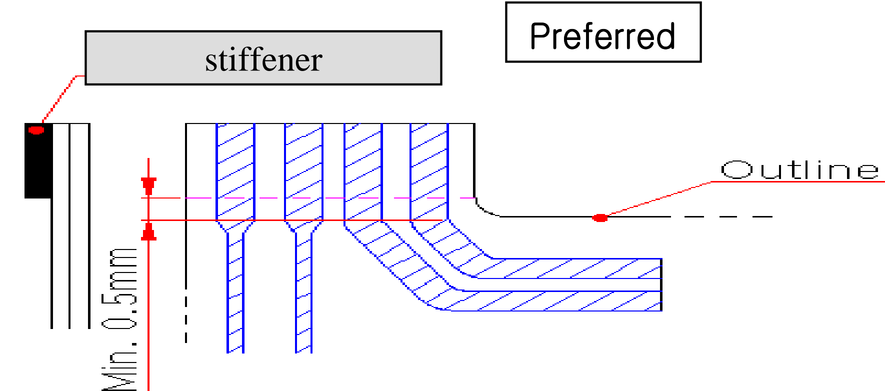
(Unit : mm)



Item	Tolerance
" A " Dimension Tolerance (Misalignment of Punching to Pattern)	± 0.1
" B " Dimension Tolerance (Tolerance of Pattern Pitch)	± 0.02
" C " Dimension Tolerance (Tolerance of Accumulated Pattern Pitch)	± 0.03
" D " Dimension Tolerance (Tolerance of Connector Width)	± 0.05

Stiffener and Pattern Design

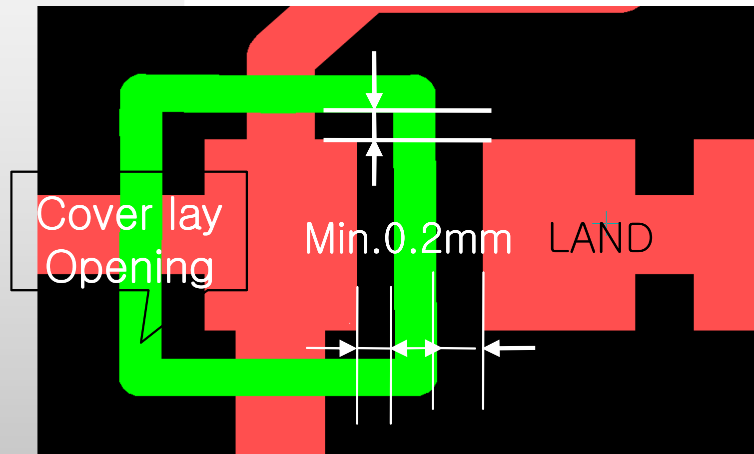


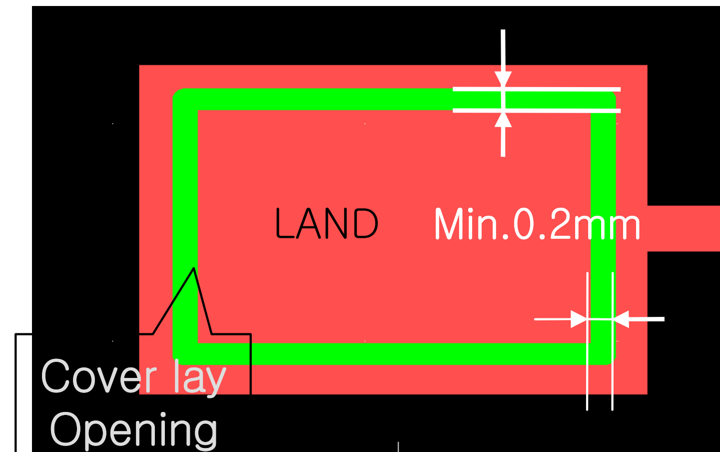


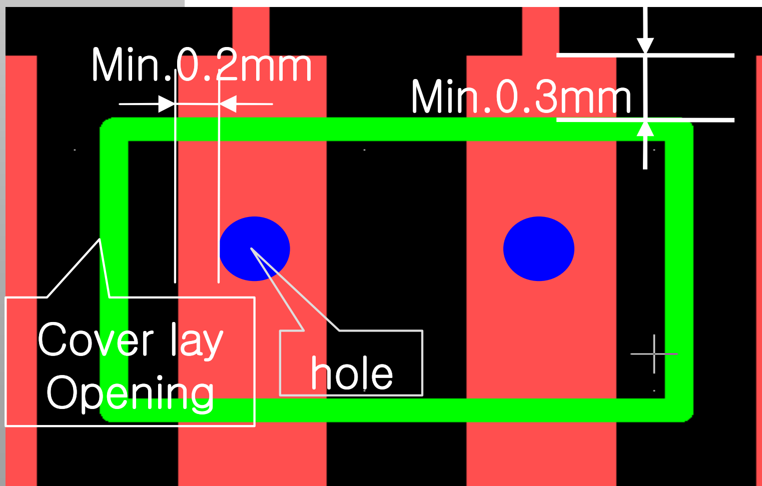
Pattern line should be 0.5mm longer than the stiffener's size to prevent cracks.

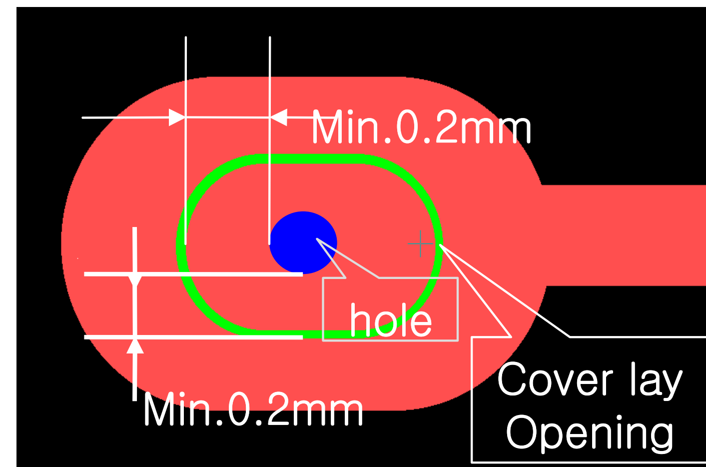
Design Guides and Rules

Pattern Cover lay Open Area Specification









Production Process

Single side

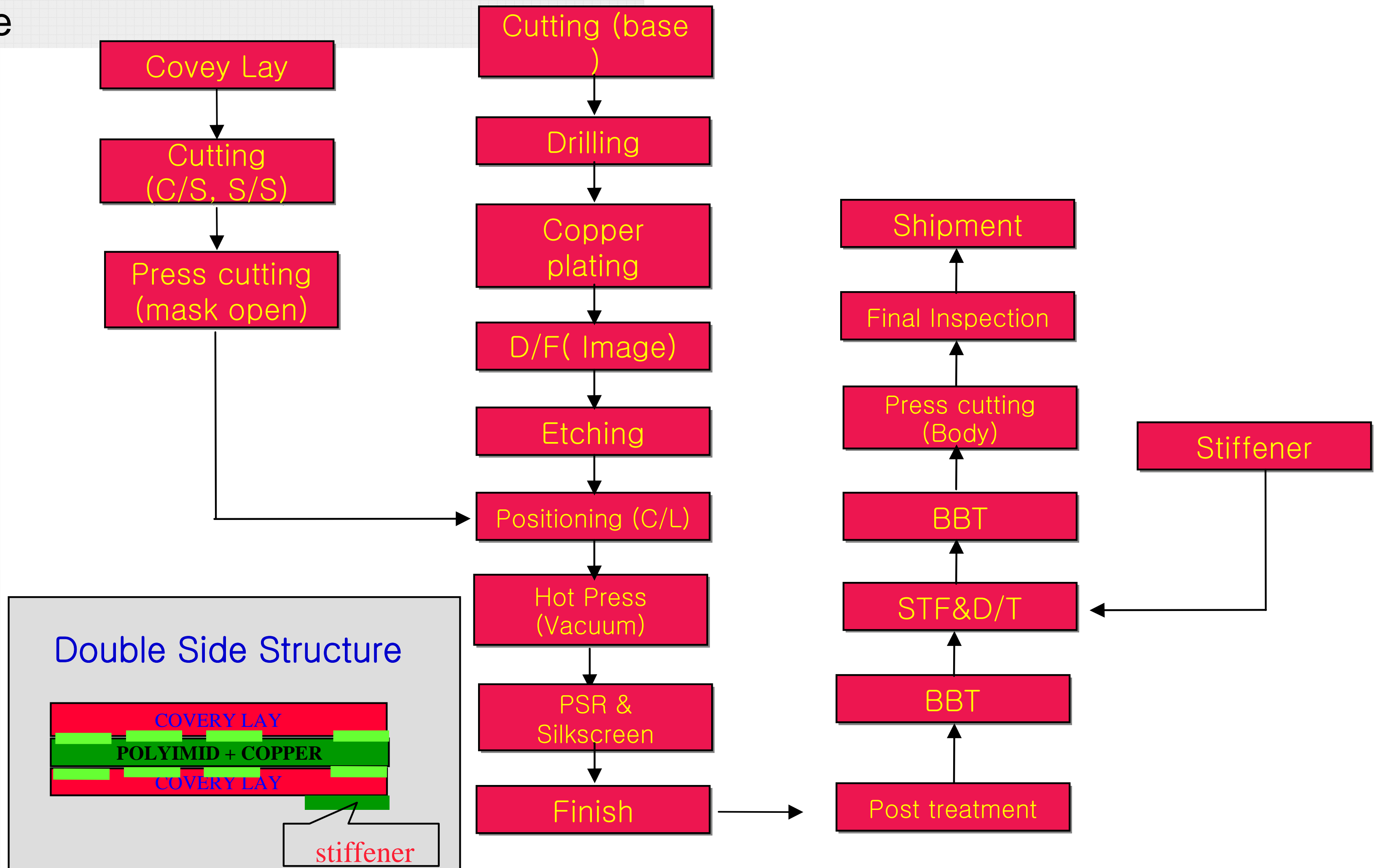
Cutting (base)
Dry film Image
Etching (Outer)
Positioning (C/L)
HOT PRESS (Vacuum)
PSR & Silkscreen
Finish
Post treatment
BBT
Press cutting (Body)
Final Inspection
Shipment

Cover Lay
Cutting
Press cutting (Mask open)

Single Side Structure

Production Process

Double side



Advanced Electronic Technology
Printed Circuit Boards

Raw Material

CCL (Copper Clad Lamination)

Item	Conductor (Copper)	Base Film (Polyimide)
Single Sided CCL	Rolled Annealed(RA) or Electro Deposit(ED) 1/3 oz (12㎛),1/2 oz(18㎛), 1oz(35㎛)	1/2mil (12.5㎛) 1mil (25㎛)
Double Sided CCL	Rolled Annealed(RA) or Electro Deposit(ED) 1/3 oz (12㎛),1/2oz.(18㎛), 1oz. (35㎛)	1/2mil (12.5㎛) 1mil (25㎛)

Cover Lay

Polyimide	Adhesive
1/2mil (12.5㎛)	15㎛, 22.5㎛
1mil (25㎛)	30㎛, 35㎛

Solder Resist

- Photo Solder Resist (PI ink) Thickness 15 ~ 20㎛

Raw Material

Surface Finish

Surface Finish Types	Thickness
HASL	3~10 ㎛
Tin Plating (Pure Sn)	0.6±0.2 ㎛
Electro Gold Plating	Ni 2~6 ㎛, Au .0.5㎛
Electro-less Gold Plating	Ni 1.5~5㎛, Au 0.05㎛ (according to flexibility)
Pre-flux (OSP)	0.3±0.1 ㎛

Stiffener

Stiffener Types	Thickness
Polyimide	25 ~ 225 ㎛
Polyester (Natural, White)	50 ~ 188 ㎛
FR-4 (Glass Epoxy)	0.2 ~ 1.6 mm
XPC-1 (Phenolic Paper)	0.5 ~ 1.6 mm
Etc.	Aluminum, Stainless Steel...