FLEX / RIGID FLEX DESIGN GUIDELINES 1.2

pcbx

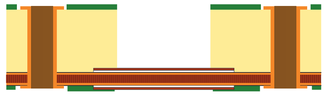
Build capabilities

Single sided flex (IPC-6013 type 1)
Coverlay (polyimide + adhesive) bonded onto an adhesiveless single sided FPC core. With or without stiffeners.

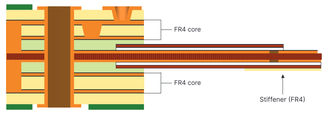
Double sided flex (IPC-6013 type 2)
Coverlay bonded onto both sided of an adhesiveless double sided FPC core (two conductive layers) with plated through holes. With or without stiffeners.

Multilayer flex (IPC-6013 type 3)
Coverlay bonded on both sides of an adhesiveless construction containing three or more conductive layers with plated through holes. With or without stiffener. Capability is 4L.

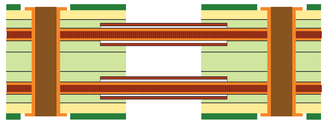
Traditional rigid flex construction (IPC-6013 type 4)
Multilayer rigid and flexible circuit combination containing three or more layers with plated through holes. Capability is 22L with 10L flex layers.



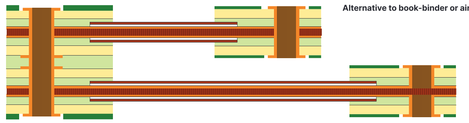
Asymmetrical rigid flex construction, where the FPC is situated on the outer layer of the rigid construction. Containing three or more layers with plated through holes.



Multilayer rigid flex construction with buried / blind via (microvia) as part of the rigid construction. 2 layers of microvia are achievable. Construction may also include two rigid structures as part of a homogeneous build. Capability is 2+n+2 HDI structure.



Book-binder and air-gap build
With spaces between layers of flex to allow for improved flexing of the FPC. Flex layers may be of different lengths on the book-binder constructions to minimise compression of inner flex layers within bend radius.



Alternative to book-binder or air-gap

Rigid - flex interface

Flex penetration region (IPC-2223 5.2.2.2)
To minimize z-axis expansion, and risks to PTH, the levels of adhesive should be kept to a minimum within the rigid construction. To achieve this IPC recommends the partial coverlays of the flexible layers should be overlapped by the rigid sections by 1.27 to 2.54mm / 0.05 to 0.10inch.

PCBX advanced capability is minimum 0.5mm / 0.02 inch depending upon design and volumes. Noting that we have to consider the potential transition zone implications. Please consult with PCBX Technicians when working to advanced capabilities.

PTH to edge of rigid section (IPC-2223 5.2.2.3)
IPC recommends PTH's in the rigid section should not be less than 3.18mm / 0.125in plus ½ of the PTH pad diameter from the rigid to flex interface when measured from the PTH center to the edge of rigid material.

PCBX advanced capability is minimum 1.3mm plus ½ PTH diameter depending upon design and volumes.

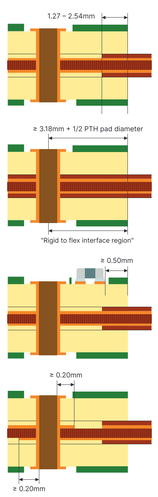
SMD keep out
– SMD components should be kept away from the edge at the flex interface to avoid any flatness concerns in this area. Preferable to avoid such areas (coverlay overlap / flex penetration) if space permits.

PCBX advanced capability is 0.5mm / 0.02 inch depending upon design and volumes, PCBX general capability is 0.8mm / 0.03 inch

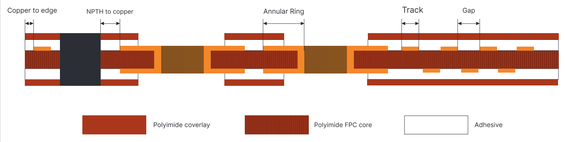
Annular rings on inner layers
For the rigid part of the construction IPC-2221 can be followed.

PCBX advanced capability is ≥0.13mm / 0.005 inch for 18um base copper and ≥ 0.15mm / 0.006 inch for 35um base copper.

PCBX general capability is ≥ 0.20mm / 0.008 inch is recommended for flex inner layer where possible, allowing for the less stable material



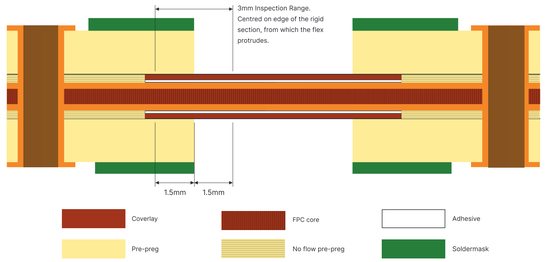
Circuitry design



Feature	RECOMMENDED (BASE 18UM / 35UM)	ADVANCED (BASE 18UM / 35UM)
Copper to edge (flex)	250um	152um
NPTH to copper	250um	152um
Annular ring	150um	100um
Track width	100um	64 / 76um
Gap	100um	64 / 76um

For questions relating to finer track width and gap, please contact PCBX Technicians regarding use of thinner copper foils.

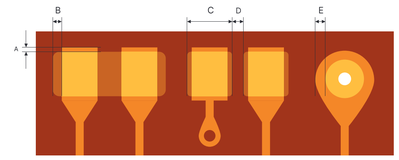
Transition Zone



IPC-6013 3.3.1.3 states "Imperfections such as protruding dielectric, deformation of dielectric, crazing or haloing shall not be a cause for rejection within this zone. A non-laminated gap which is due to material misalignment, may penetrate up to 50% of the distance from the edge to the nearest conductor or the edge of the flexible coverlay, whichever is less".

Adhesive spill out or pre-preg flow onto the flex part of the structure is allowable within this zone.

Coverlay design

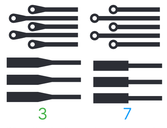


Feature	RECOMMENDED	ADVANCED
A. Coverlay pad capture overlap (min 2 locations)	200um	100um
B. Coverlay opening clearance	200um	76um
C. Coverlay opening width	200um	127um
D. Coverlay opening web	350um	200um
E. Coverlay access holes overlap (min 2 locations)	250um	100um

If dimensions are too large for design, please contact PCBX Technicians regarding use of flexible soldermask as the design rules are close to that of standard soldermask. Machined or cut access in the coverlay leads to risk of squeeze out, whilst photo-imaged coverlay will not.

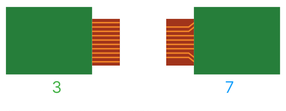
Circuitry design

TEAR DROPPING



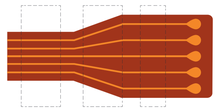
Where the circuitry meets a feature (pad or annular ring) there should be no sudden change in track widths and no sharp corners as features merge. Such sudden changes lead to increased stress and reduced reliability. Add tear drops to such features.

TRACK SPACING / GEOMETRY



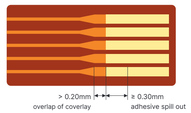
When exiting the rigid part of the PCB the circuitry should be perpendicular to the edge / right angle to the bend. There should be no sudden of abrupt changes in the direction of the tracks within the transition to the flex part. Changes in track width should occur > 2.54mm from the edge. Noting that impedance calculations must consider both track widths.

BEND AREA



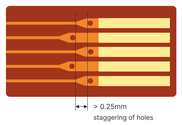
Tracks shall be perpendicular to the bend – do not design for bend areas where the tracks are curved or angled (in this example the middle section). Tracks shall also be evenly spread across the bend area and shall maintain a constant width. There shall be no holes in the bend area and where possible the number of conductive layers shall be kept to a minimum.

COVERLAY OVERLAP ON CONNECTORS / GOLD FINGERS



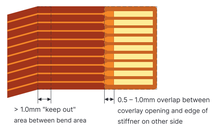
Access openings for gold fingers areas shall cover the 'foot' of the gold finger by more than 0.20mm to prevent any damage from flex stresses – i.e. the coverlay shall extend 'over' the start of the finger. NOTE: IPC-A-600 4.1.2.2 allows ≤ 300um / 0.30mm adhesive squeeze out for class 2 (70um copper foil and below) so we should also take this into consideration where calculating how big the contacts should be.

STAGGERING OF HOLES



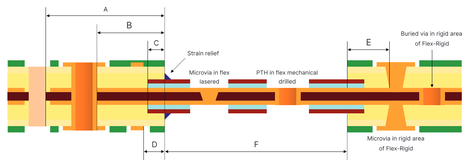
Holes within the gold fingers areas shall not be in a straight line, but rather staggered to avoid generating a weak spot which can cause the circuitry to break. The recommended gap between the holes shall be more than 0.25mm.

STIFFENER OVERLAP ON FINGERS



Without overlap between coverlay / edge of contacts and stiffener then we run the risk of the contacts cracking as it is right behind the stiffener that the board will be begin to flex... also this allows enough coverlay contact to prevent lifting. Combine this with good coverlay overlap as shown previously and we have a good design.

HDI Rigid - flex



Feature	RECOMMENDED (mm)	ADVANCED (mm)
A. Edge NPTH to 'Flex Rigid transition zone'	≥ 1.3	≥ 1.0
B. Edge drill tool PTH to 'Flex Rigid transition zone'	≥ 0.8	≥ 0.5
C. Coverlay penetration in the rigid area	≥ 0.8	≥ 0.5
D. (I/L) Edge SMD pad to 'Flex Rigid transition zone'	≥ 0.5	≥ 0.3
D. (O/L) Edge SMD pad to 'Flex Rigid transition zone'	≥ 0.5	≥ 0.25
E. (I/L) Edge laser micro-via to 'Flex Rigid transition zone'	≥ 0.8	≥ 0.5
E. (O/L) Edge laser micro-via to 'Flex Rigid transition zone'	≥ 0.8	≥ 0.3
F. Length of the flex area	≥ 10	≥ 4

Material capabilities

AVAILABLE SURFACE FINISHES INCLUDE: ENIG / IMMERSION AG / IMMERSION SN / OSP / HARD GOLD / FLASH

Excellent flexibility (all temps)
Good electrical / chemical properties
Good tear resistance
Excellent assembly performance
Higher dielectric strength

Higher cost
Moisture absorption

Feature	MATERIAL	AVAILABLE OPTIONS
Rigid CCL / core	Shengyi, Iteq, Ventec, Rogers, Nelco, Panasonic, Arlon, Isola	FR4 (mid - high Tg), Halogen Free, PI, Low Dk, Df
Pre-preg	Shengyi, Iteq, Ventec, Rogers, Nelco, Panasonic, Arlon	FR4 (mid - high Tg), Halogen Free, PI, Low Dk, Df
No-flow pre-preg	EMC, Doosan, Arlon, Ventec	FR4 (mid - high Tg), Halogen Free, PI
Soldermask	Taiyo, Tamura etc.	
FPC / PI base material	Dupont, Taiflex, Thin flex, Panasonic, Shengyi, Doosan	12um / 25um / 50um / 75um / 100um / 125um / 150um
Coverlay	Dupont, Taiflex, Shengyi, Thin flex	PI = 12um / 25um / 50um / 75um Adhesive = 12um / 25um / 50um
Cu thickness on FCCL		9um / 12um / 18um / 35um / 70um
Adhesive / Bondsheet	Dupont, Taiflex, Shengyi	12um / 15um / 25um / 50um
Flexible soldermask	Taiyo, Sunchemical	
Silver / EMI shielding	Tatsuta, Sanyo, Toyobo	Film, Ink

Design tips

1 **Adhesiveless polyimide systems** should be used due to increased reliability and lower z –axis CTE expansion compared to adhesive systems (acrylic = 2,5 x PI in terms of z-axis CTE).

2 **For dynamic applications** keep layer count as low as possible.

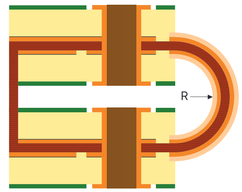
3 **Static / flex to install** applications can support higher layers in the flex construction (XX max).

4 **Semi-flex can be** used for flex to install applications – multiple flexing increases risk of cracks in FR4 / copper.

5 **For dynamic use** materials (FPC + coverlay) with similar properties.

6 **Use IPC-2223** to calculate accurate flex length for minimum bend radius. Incorrect length causes problems.

For basic guides to achieve approximation :
1L flex = 3 – 6 x FPC thickness
2L flex = 7 – 10 x FPC thickness
Dynamic = 20 – 40 x FPC thickness



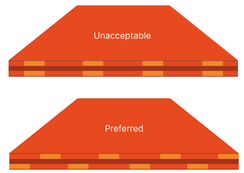
7 **NO sharp edges** or corners on flex outline / circuitry

8 **Solid copper fill** on back of gold finger, if stiffeners cannot be added.

9 **Always keep stiffeners** the same thickness.

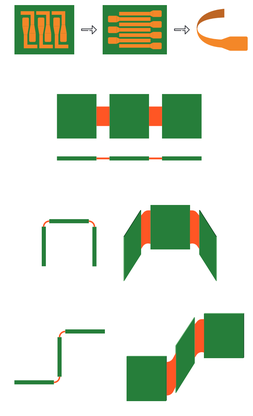
10 **Always design pads** larger than coverlay.

11 **No overlay of** tracks for dynamic applications – they should be offset when comparing L1 to L2.

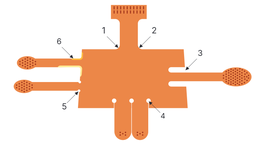


12 **As per IPC** it is allowed for 300um adhesive squeeze out from the edge of the coverlay, so ensure features have sufficient spacing / distance from the termination of coverlay.

13 **Think in flexible** terms – that straight flat flex can flex to fit.



14 **Outline considerations for** robust flexing.



1 Large radius in corners (>1.5mm, large = better)
2 Tangent / same position corners on same feature
3 Recessed slot
4 Hole at slit termination to stop tearing (>1.5mm)
5 Drilled hole at corner (>1.5mm)
6 Extra copper to strengthen corner