Multek RFPC (Rigid Flex Printed Circuits) Design Rules Metrics

B2F/B1
Rigid Flex Design Guideline

Revision D

Note: The intent of this design guide line is to assist you, our customer, to capture the initial design in an efficient & straight forward flow. It is highly recommended to keep close communication with the RFPC manufacturer on important design details to ensure that your particular design will be both cost-effective and producible. Release of these Design Guidelines outside the company in parts or in whole, is subject to approval and is for reference only.

Table of Contents:

7 Revision History:

Effective Date	Version	Description	Revised Page	Revised By
Sep.11.2013	Rev D	Add stack up for full Coverlay and update some material	18~20,25,30	Lifa Yang

Approved: Victor Llanes 09132013

1 General Information

Application Guide:

This guide is intended to help you determine when it is appropriate to use Rigid-Flexible printed circuitry and which technology is best suited to your needs.

Consider Flexible Circuitry if your product requires:

- Three dimensional packaging
- Flex-to-install
- Dynamic flexibility
- Heat dissipation
- Compatibility with thermal cycling
- Mechanical Shock & Vibration reliability is characterized by end-product
- Minimized weight and space

Benefits & Advantages of Rigid-Flexible Circuitry includes:

- A single substrate for Electronic System Packaging. Repeatability – Reliability – High Density
- System cost savings resulting from component integration
- Elimination of wiring errors during installation and servicing, reduced assembly labor
- Three dimensional packaging - Flexible circuitry can be bent to fit individual products needing to be installed in non-planar space.
- Minimized weight and space, 75% less weight of conventional wiring
- Improved reliability through elimination of connectors and reduced thermal stress on solder joints
- Thin, flat conductors and thin insulation resulting in improved thermal dissipation
- Electrical performance as Key attributes. More consistent Electrical Impedance Performance with integral ground planes.
- Uniform electrical characteristics with consistent conductor spacing and insulation parameters

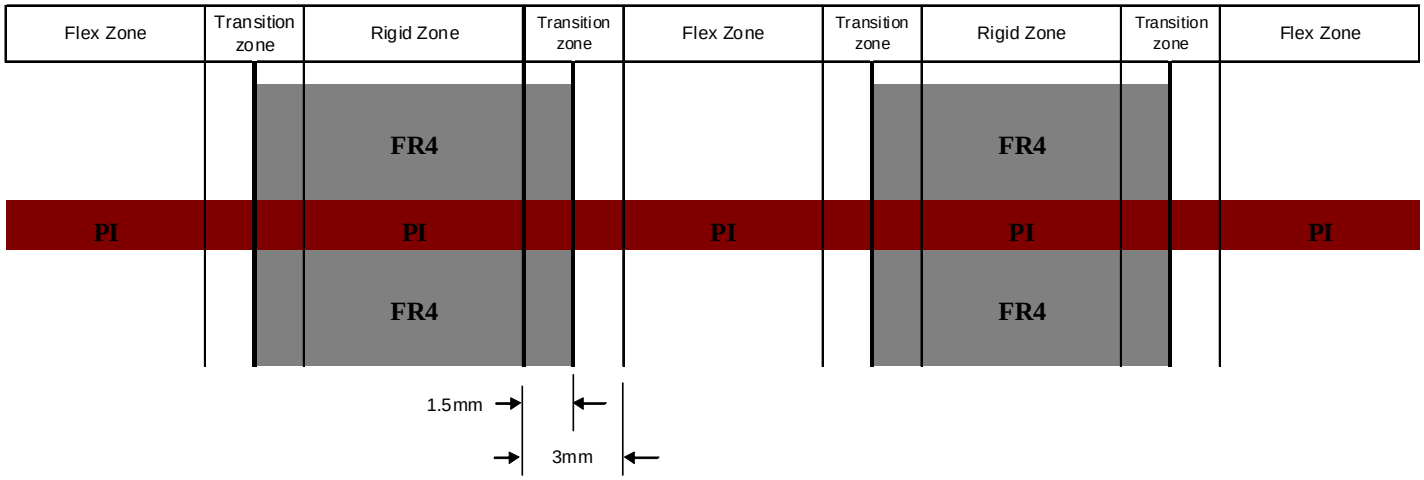
Classes:

- **Use A:** Capable of withstanding flexing during installation.
- **Use B:** Capable of withstanding continuous flexing for a number of cycles as specified on the procurement documentation. (Generally not used for more than two conductor layers.)
- **Use C:** High Temperature Environment (over 105 C [221 F]).
- **Use D:** UL Recognition.

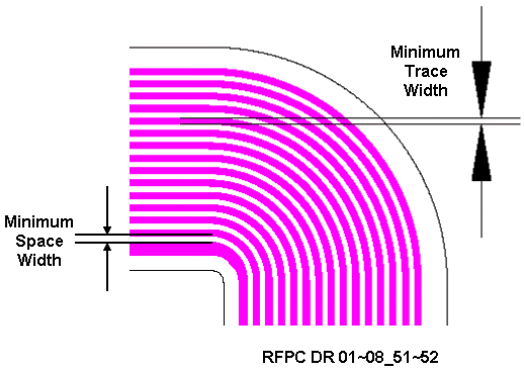
Remark: The recommendation contained in this document is general information.
For specific concerned projects, please feel free to discuss with engineering team for solutions.

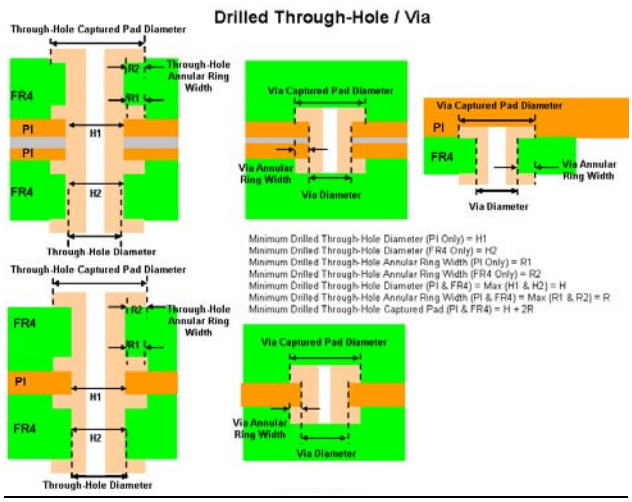
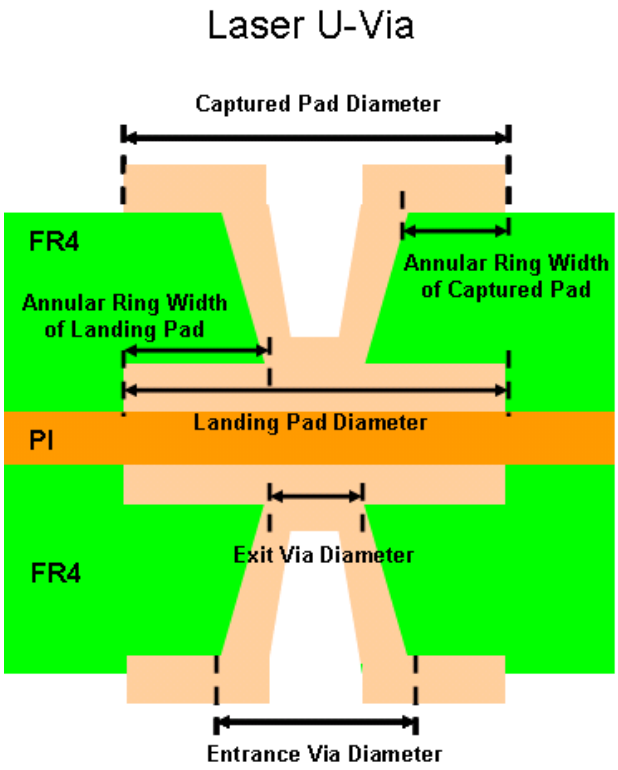
2 Multek RFPC (Rigid Flex) Design Rules Metrics

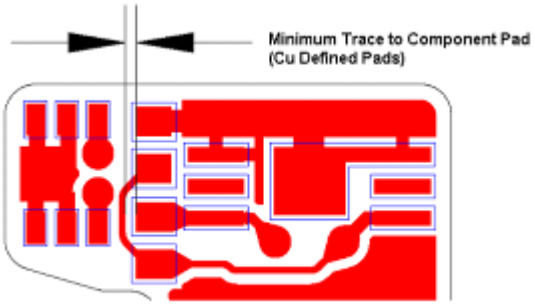
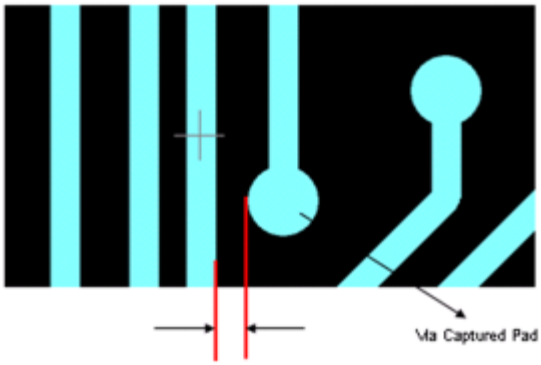
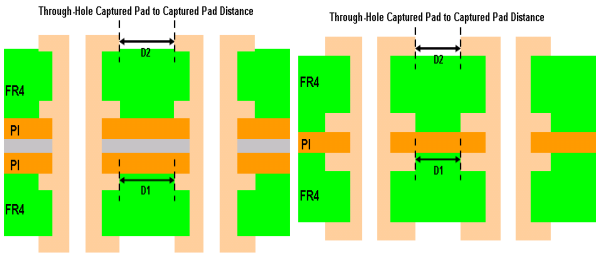
Rigid Flex Illustration

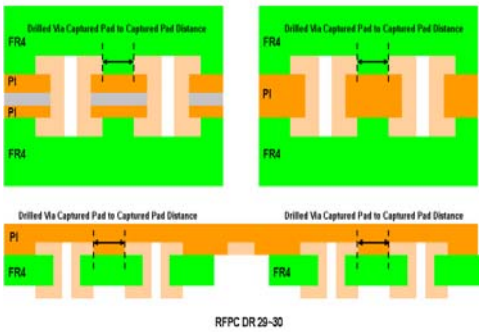
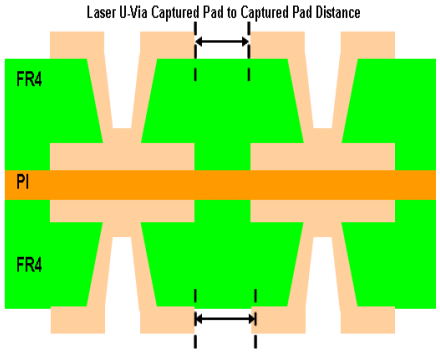
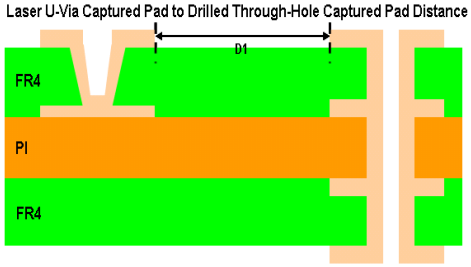
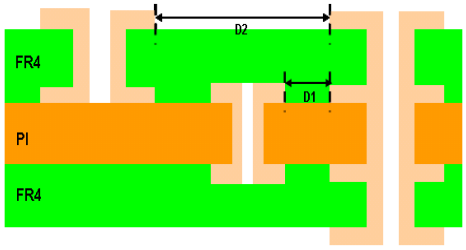


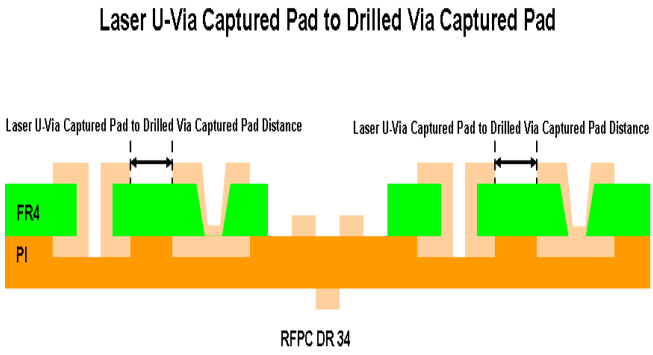
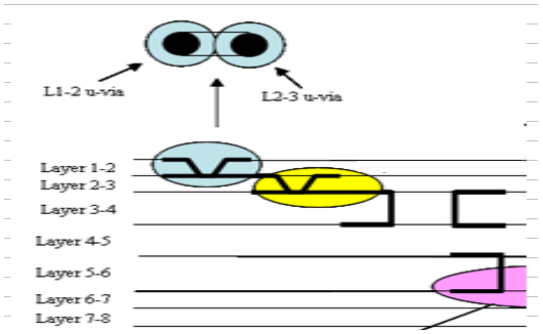
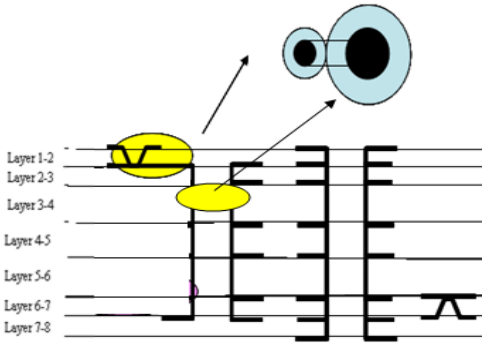
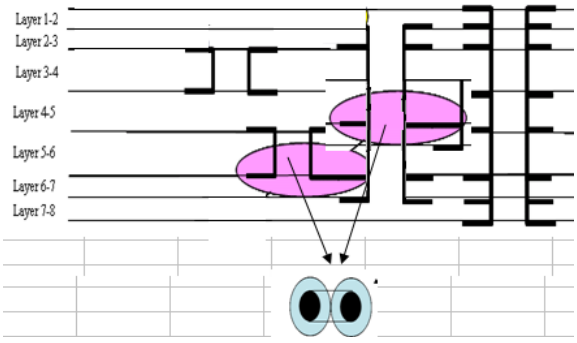
2.1 Rigid Region

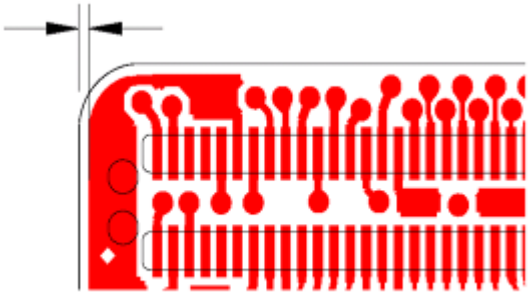
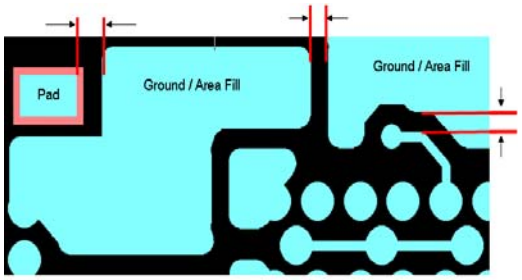
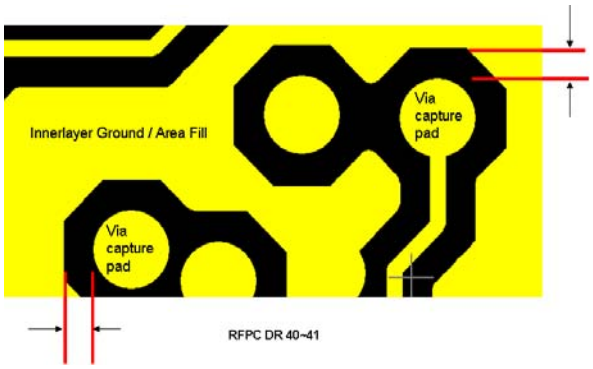
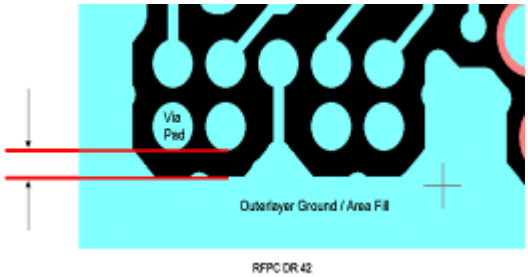
DR#	**TRACE & SPACE** *Utilized standard design rule for Trace & Space for cost reduction and higher yield.*	Photos/Drawings	**Standard (µm)**
1	Minimum Trace Width (FR4 Outer Layer: Cu thickness < 1 oz)	 RFPC DR 01~08_51~52	100
2	Minimum Trace Width (FR4 IL using ½ oz Cu Foil Only)		75
3	Minimum Trace Width (FR4 IL using ½ oz Cu Foil & Plating) Cu thickness < 40um		100
4	Minimum Trace Width (PI IL using ½ oz Cu Foil)		75
5	Minimum Space Width Btw Traces (FR4 Outer Layer) Cu thickness < 1 oz		100
6	Minimum Space Width Btw Traces (FR4 IL using 1/2 oz Cu Foil Only)		75
7	Minimum Space Width Btw Traces (FR4 IL using 1/2 oz Cu Foil & Plating) Cu thickness < 40um		100
8	Minimum Space Width Btw Traces (PI IL using 1/2 oz Cu Foil)		75

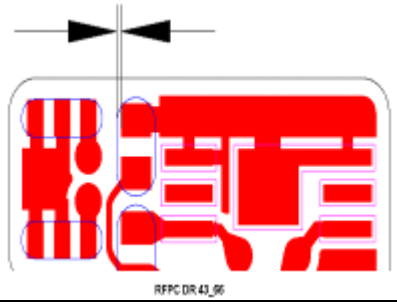
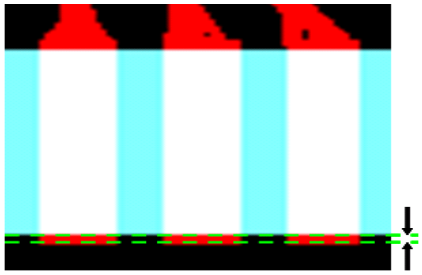
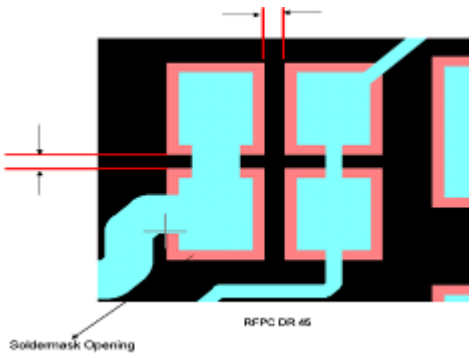
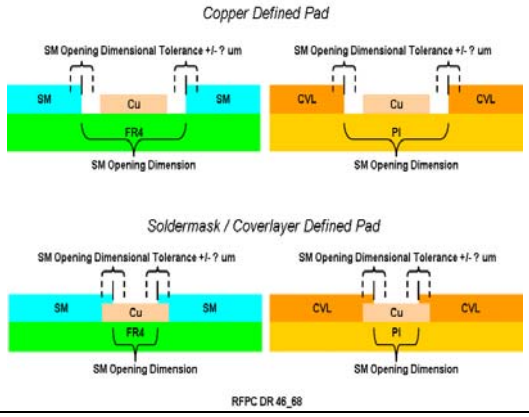
DR#	DRILLED THROUGH HOLE / DRILLED VIA / LASER U-VIA	Photos/Drawings	Standard (µm)
9	Minimum Drilled Through-Hole Diameter (PI + FR4) - Thk < 0.6mm		200
10	Minimum Drilled Through-Hole Captured Pad (PI + FR4)		450
11	Minimum Drilled Through-Hole Annular Ring Width (PI + FR4)		125
12	Minimum Drilled Via Diameter (PI (PI Only)		75
13	Minimum Drilled Via Captured Pad (PI Only)		375
14	Minimum Drilled Via Annular Ring Width (PI Only)		150
15	Minimum Laser U-Via Top Diameter (FR4 Only, dielectric<50um)		125
16	Minimum Laser U-Via Landing Pad Diameter (FR4 Only)		325
17	Minimum Laser U-Via Captured Pad Diameter (FR4 Only)		325
18	Minimum Laser U-Via Annular Ring Width on Landing Pad (FR4 Only)		100
19	Minimum Laser U-Via Annular Ring Width on Captured Pad (FR4 Only)		100

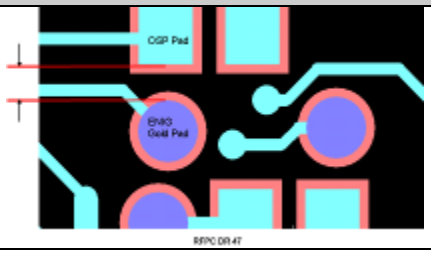
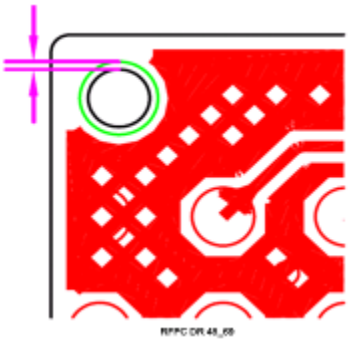
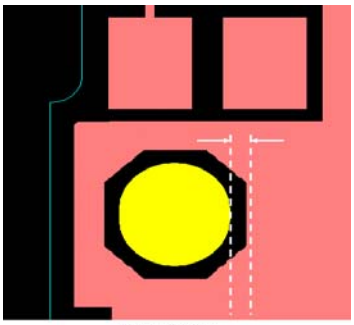
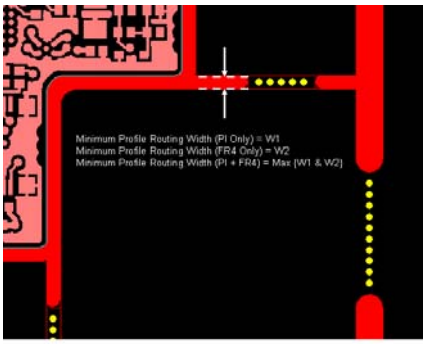
DR#	COPPER TO COPPER CLEARANCE	Photos/Drawings	Standard (µm)
20	Minimum Trace to Component Pad (LPI SM Defined Pads) (FR4 - 1oz)	 RFPC DR 23	100
21	Minimum Trace To Component Pad (Copper Defined Pads) (FR4 - 1oz)	 RFPC DR 24	150
22	Minimum Trace to Through-Hole / All Via Captured Pads (FR4 Outer Layer 1oz)	 Minimum Trace to Through-Hole / Via Captured Pad RFPC DR 25~27_59	100
23	Minimum Trace to Through-Hole / All Via Captured Pads (FR4 Inner Layer 0.5oz)		75
24	Minimum Trace to Through-Hole / All Via Captured Pads (PI Inner Layer 0.5oz)		75
25	Minimum Through-Hole Captured Pad to Through-Hole Captured Pad (PI + FR4 1 oz)	Through-Hole Captured Pad to Through-Hole Captured Pad  Minimum Through-Hole Captured Pad to Captured Pad Distance (PI Only) = D1 Minimum Through-Hole Captured Pad to Captured Pad Distance (FR4 Only) = D2 Minimum Through-Hole Captured Pad to Captured Pad Distance (PI & FR4) = Max {D1 & D2} = D RFPC DR 28	100

DR#	COPPER TO COPPER CLEARANCE	Photos/Drawings	Standard (µm)
26	Minimum Drilled Via Captured Pad to Drilled Via Captured Pad (FR4 - 1oz)	Drilled Via Captured Pad to Drilled Via Captured Pad  RFPC DR 29~30	100
27	Minimum Drilled Via Captured Pad to Drilled Via Captured Pad (PI - 0.5oz)		75
28	Minimum Laser U-Via Captured Pad to Laser U-Via Captured Pad (FR4 - 1oz)	Laser U-Via Captured Pad to Laser U-Via Captured Pad  RFPC DR 31	100
29	Minimum Laser U-Via Captured Pad to Through-Hole Captured Pad (FR4 - 1oz)	Laser U-Via Captured Pad to Drilled Through-Hole Captured Pad Distance  Minimum Laser U-Via Captured Pad to Drilled Through-Hole Captured Pad Distance (FR4 Only) = D1	100
30	Minimum Drilled Via Captured Pad to Through-Hole Captured Pad	Drilled Via Captured Pad to Drilled Through-Hole Captured Pad Distance  Minimum Drilled Via Captured Pad to Drilled Through-Hole Captured Pad Distance (PI Only) = D1 Minimum Drilled Via Captured Pad to Drilled Through-Hole Captured Pad Distance (FR4 Only) = D2 Minimum Drilled Via Captured Pad to Drilled Through-Hole Captured Pad Distance (PI & FR4) = Max {D1 & D2} = D RFPC DR 32~33	100

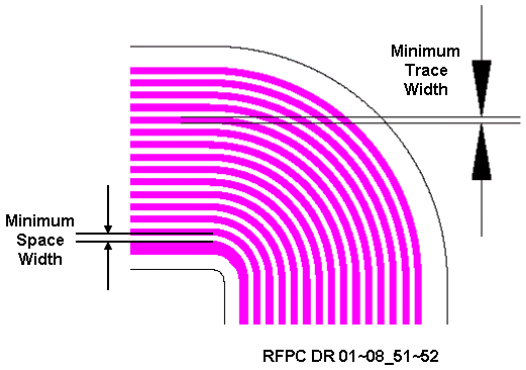
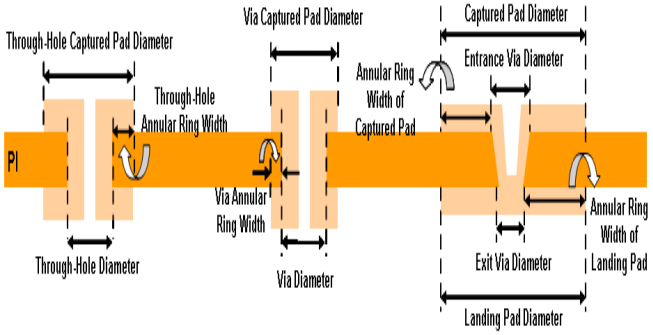
DR#	COPPER TO COPPER CLEARANCE	Photos/Drawings	Standard (µm)
31	Minimum Laser U-Via Captured Pad to Drilled Via Captured Pad (FR4 - 1oz)		100
32	Spacing of laser via to laser via in the same net of adjacent layers(hole edge to hole edge)		125
33	Spacing of laser Via to Drilled Via in the same net (hole edge to hole edge)		200
34	Spacing of Drilled Via to Drilled Via in the same net (hole edge to hole edge)		250

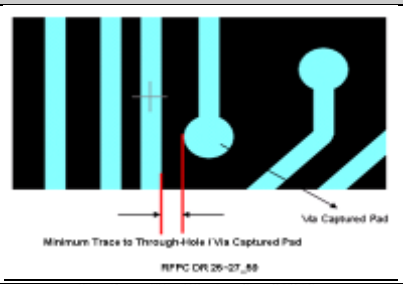
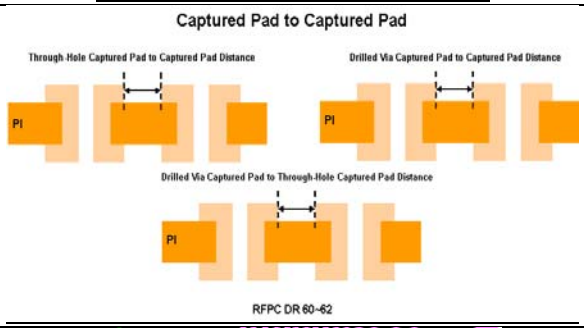
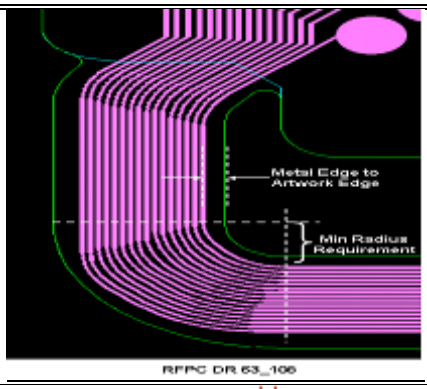
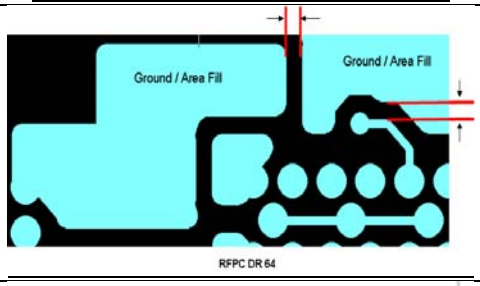
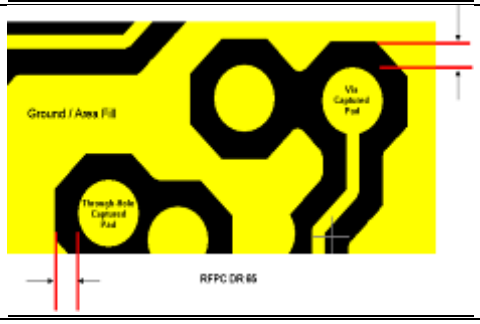
DR#	COPPER TO COPPER CLEARANCE	Photos/Drawings	Standard (µm)
35	Minimum Metal Edge to Board's Edge Distance(Exception: Edge Plating or Assembly Rule Requirements) (FR4 Only)	 RFPC DR 36~37	250
36	Minimum Metal Edge to FPC's Edge Distance(Exception: Assembly Rule Requirements) (PI Only)		250
37	Minimum Space from Pad / Trace / Ground to Ground Fill for all Layers (FR4 - 1oz)	 RFPC DR 38~39	100
38	Minimum Space from Pad / Trace / Ground to Ground Fill for all Layers (PI - 0.5oz)		75
39	Minimum Space from Drilled Via Capture Pads or Laser U-Via Landing Pads to Ground Fill for all Inner Layers (FR4 – 1oz)	 RFPC DR 40~41	100
40	Minimum Space from Drilled Via Capture Pads or Laser U-Via Landing Pads to Ground Fill for all Inner Layers (PI - 0.5oz)		75
41	Minimum Space from All Types of Via Pads to Ground Fill for Outer Layer (FR4 - 1oz)	 RFPC DR 42	100

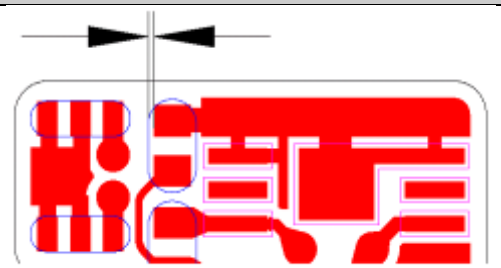
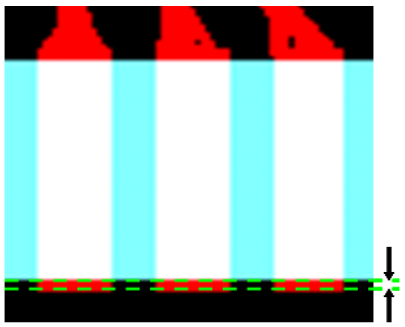
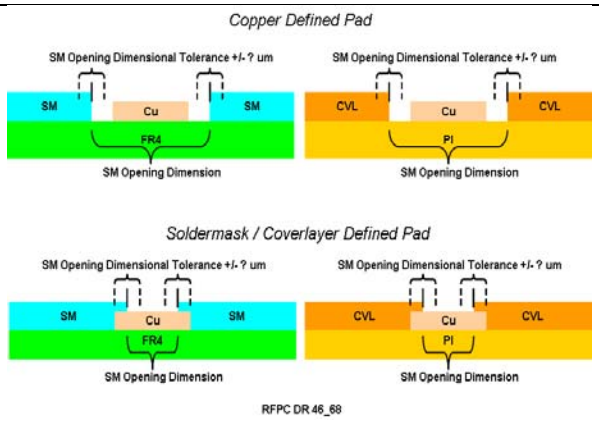
DR#	**SOLDER MASK:** Solder Mask can be used in the Rigid areas of Rigid-Flex circuits using LPI (Liquid Photo Image able).	Photos / Drawings	Standard (µm)
42	Minimum Space Between Solder mask Edge to Metal Edge (FR4 Only)	 RFPC DR 43_66	60
43	Minimum Solder mask Overlap on Solder Pad (Not Applicable to 0402 and smaller components)	 RFPC DR 44_67	75
44	Minimum Solder Mask Width (FR4 Only)		80
45	Solder Mask Opening Dimensional Tolerance(Cu / SM Defined Pads) (FR4 Only)		+/- 25

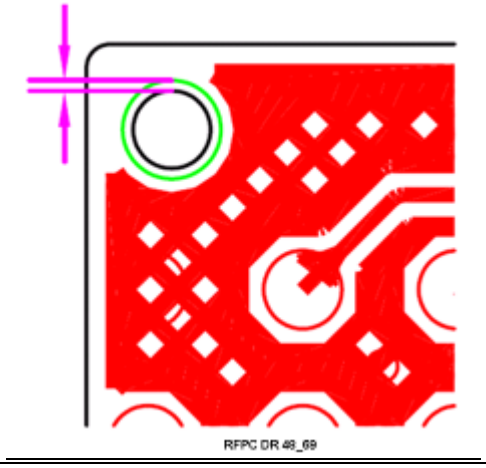
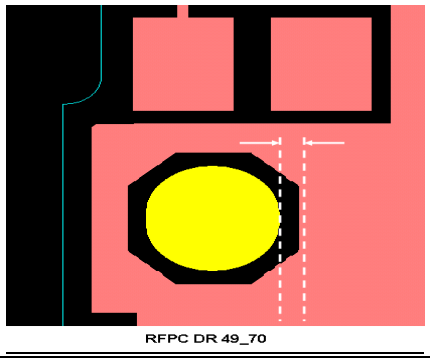
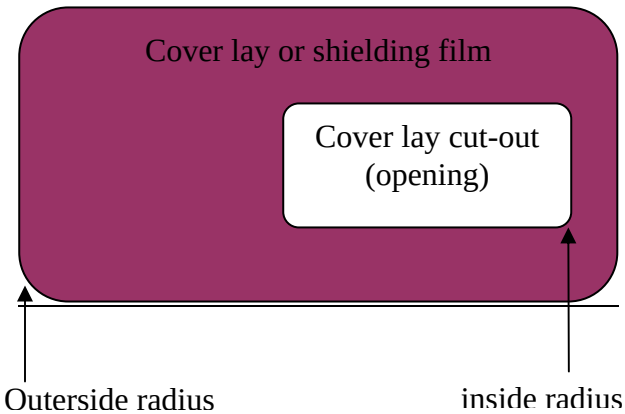
DR#	SOLDER MASK (cont'd)	Photos / Drawings	Standard (µm)
46	Minimum OSP Pads to ENIG Pads Spacing(FR4 Only)		350
47	Minimum Space Solder Mask to Non-Plated Through Hole (FR4 Only)		125
48	Minimum Non-Plated Slot or Hole Edge to All Metalized Edge (FR4 With SM Only)		200
49	Minimum routing path		1000

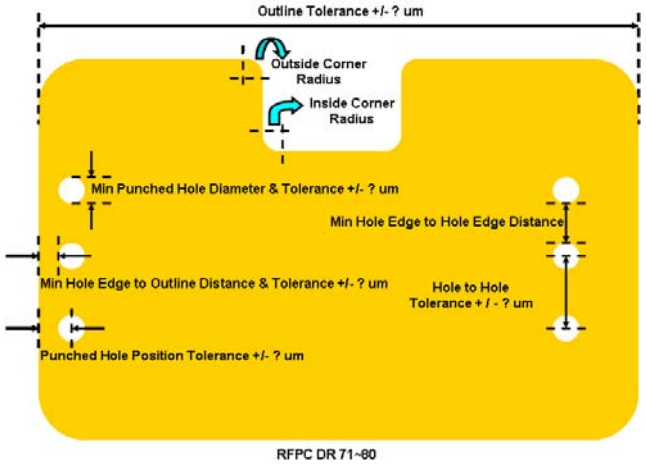
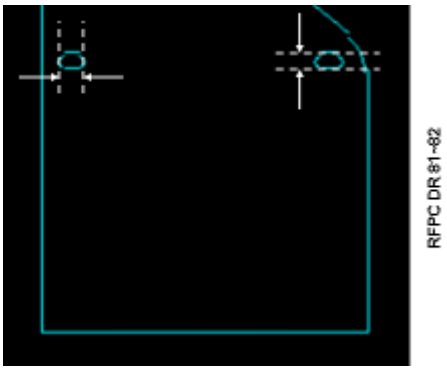
2.2 Flex Region

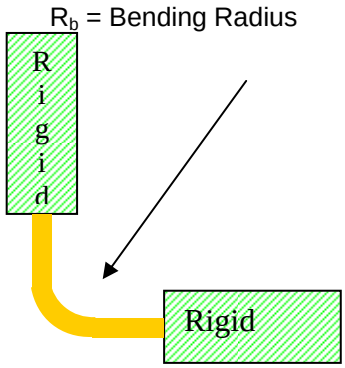
DR#	TRACE & SPACE	Photos / Drawings	Standard (μm)
50	Minimum Trace Width (1/2 oz Cu Foil)	 RFPC DR 01~08_51~52	63
51	Minimum Space Width Btw Traces (1/2 oz Cu Foil)		63
DR#	**DRILLED THROUGH HOLE / DRILLED VIA / LASER U-VIA**	**Photos / Drawings**	**Standard (μm)**
52	Minimum Drilled Through-Hole Diameter	Drilled Through-Hole / Drilled Via / Laser U-Via  RFPC DR 53~58	75
53	Minimum Drilled Through-Hole Captured Pad		375
54	Minimum Drilled Through-Hole Annular Ring Width		150

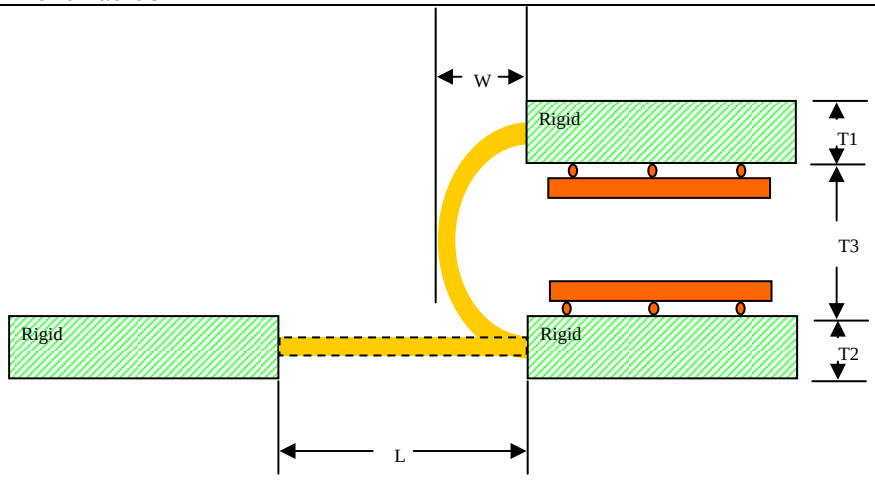
DR#	COPPER TO COPPER CLEARANCE	Photos/Drawings	Standard (µm)
55	Minimum Trace to Through-Hole /Via Captured Pad (1/2 oz Cu Foil)		63
56	Minimum Through-Hole Captured Pad to Through-Hole Captured Pad		125
57	Minimum Metal Edge to FPC's Edge Distance		250
58	Minimum Space from Trace / Ground to Ground Fill for all Layers (1/2 oz Cu Foil)		63
59	Minimum Space from Through-Hole / Drilled Via Capture Pads or Laser U-Via Landing Pads to Ground Fill for all layers (1/2 oz Cu Foil)		63

DR#	**COVER LAYER** Film bonded with thermally cured adhesive. Good abrasion resistance. Generally used in dynamic flexing applications. Hard tooling, N/C cutting, punching or drilling required	**Photos / Drawings**	**Standard (µm)**
60	Minimum Space Between Cover layer Edge to Metal Edge		150
61	Minimum Cover layer Overlap on Cu (CVL Defined Pads)		100
62	Cover layer Opening Dimensional Tolerance (Cu / CVL Defined Pads)		+/-100

DR#	COVER LAYER (cont.)	Photos / Drawings	Standard (µm)
63	Minimum Space Cover layer to Non-Plated Through Hole		175
64	Minimum Non-Plated Slot or Hole Edge to All Metalized Edge (PI With CVL Only)		200
65	Minimum cover layer outside and inside radius		200
66	Minimum shielding film outside radius		500

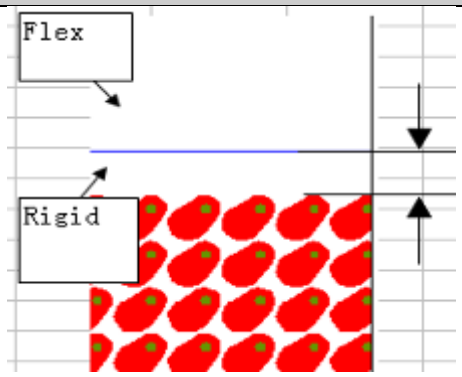
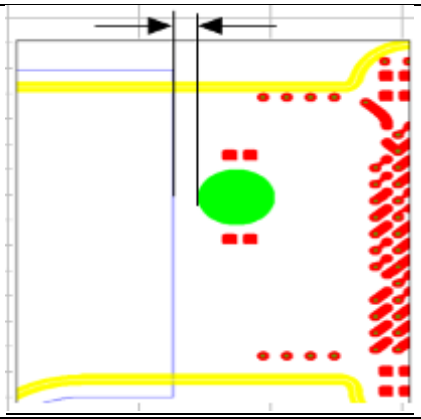
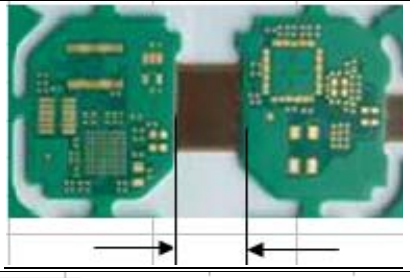
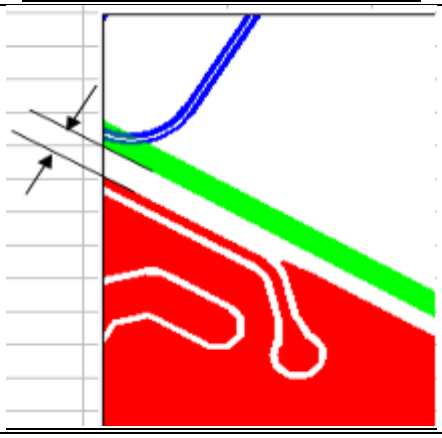
DR#	**PRECISION MALE / FEMALE DIE** (Applicable for 100% Punch in All Flexible Regions W/O Any Routing Involved)	**Photos / Drawings**	**Standard (µm)**
67	Outline Tolerance		+/- 75
68	Min. Outside Corner radius		200
69	Min. Inside Corner Radius		200
70	Minimum Punched Hole Diameter		500
71	Punched Hole diameter Tolerance		+/- 50
72	Punched Hole Position Tolerance		+/- 75
73	Minimum Hole Edge to Hole Edge Distance		300
74	Hole to Hole Tolerance		+-50
75	Hole Edge to Outline Distance		300
76	Hole Edge to Outline Tolerance		+-50
77	Minimum Slot Width		500

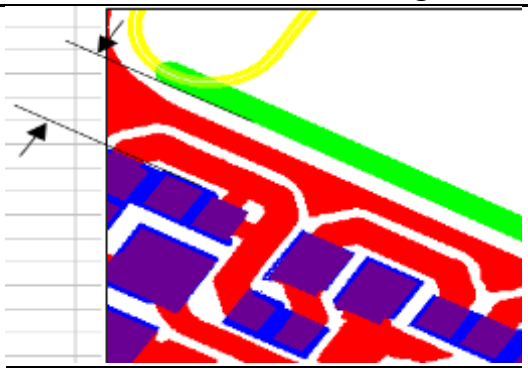
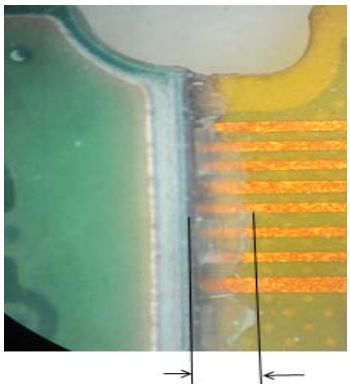
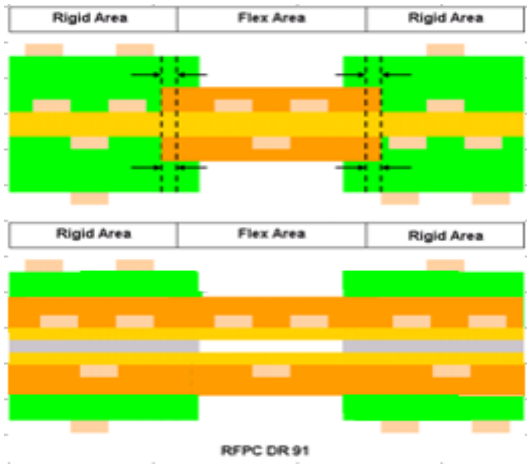
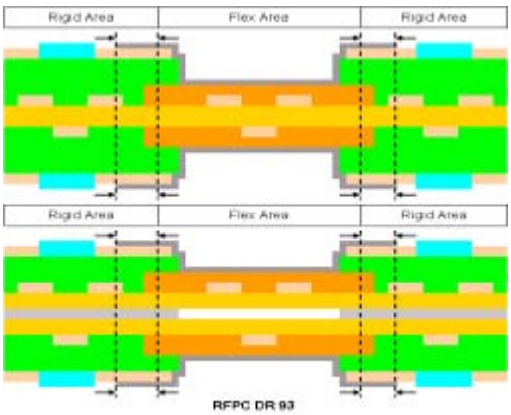
<table>
<tr><th>DR#</th><th>RADIUS OF FLEX</th><th>Photos / Drawings</th><th>Standard (µm)</th></tr>
<tr><td>78</td><td>Minimum bend radius</td><td>


Bend Radius calculations:
<table>
<tr><td>Single Sided – FPC ≤ 0.1 mm Overall thk.</td><td>Standard - 6 times of FPC thickness</td><td>Static</td></tr>
<tr><td>Single Sided – FPC (Dynamic Application)</td><td>20–40 times of FPC thickness (Increases in bend radius normally increases dynamic life cycle)</td><td>Dynamic</td></tr>
<tr><td>Double Sided – FPC ≤ 0.2 mm Overall thk.</td><td>Standard - >10 times of FPC thickness</td><td>Static</td></tr>
<tr><td>FPC with Cover Film + Silver film shield</td><td>Bend Cycle Test Required</td><td>Dynamic / Static</td></tr>
<tr><td>Multi-Layer – FPC ≥ 0.3 mm Overall thk.</td><td>Standard - ≥15 times of FPC thickness</td><td>Static</td></tr>
</table>
Factors that affect “FLEXIBILITY”
-Type of material used – Copper, Base type and Thickness
-Button plating or panel plating---General speaking, panel plating is preferred for static bending product with dielectric thickness no more than 2mil.
-Proper balance in construction
-Placement of metal in neutral axis
-Bend radius
</td><td>See Bend Radius calculations</td></tr>
<tr><td>79</td><td>Min. flex length</td><td rowspan="2">


</td><td>L=0.5 x 3.14 x(T1+T2+T3)</td></tr>
<tr><td>80</td><td>Min. installation flex width</td><td>W= 0.5 x(T1+T2+T3)</td></tr>
</table>

2.3 Transition Region

DR#	TRANSITION REGION Distance between Rigid & Flexible interface	Photos / Drawings	Standard (µm)
81	Minimum plated hole to R-F transition		800 (For partial coverlay) 500 (for full coverlay)
82	Minimum NPTH to R-F transition		500
83	Min. flex width between rigid to rigid section		5000
84	Min. spacing from trace to R-F transition		500

85	Min. spacing from component pads to R-F transition		800
86	Maximum Prepreg Squeeze out		1000
87	Minimum Coverlay Overlap Distance		For static bending, use partial coverlay , 500um overlap. For dynamic bending: full coverlay
88	Minimum EMI Shielding Film Overlap Distance		500

DR#	TRANSITION REGION	Photos / Drawings	Standard (µm

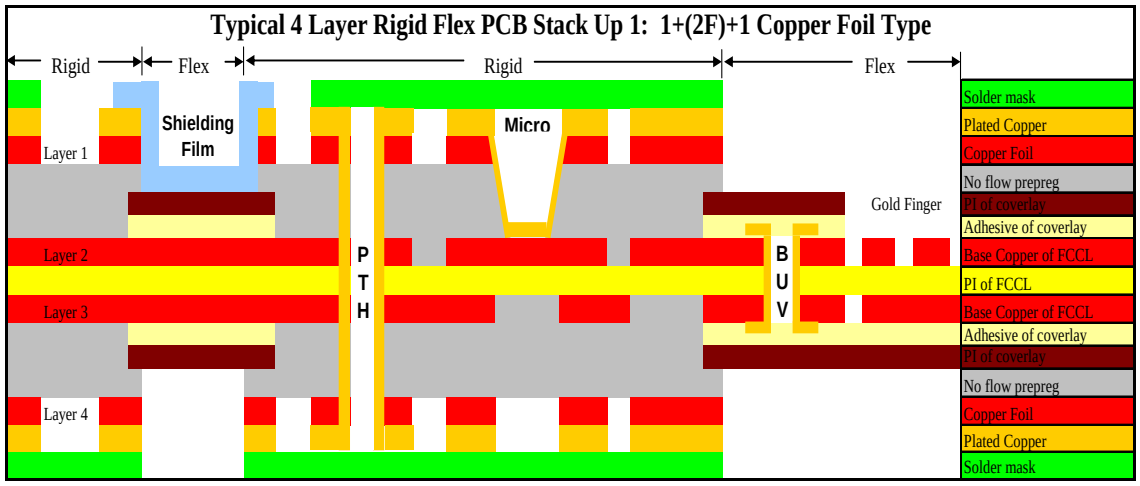
3 RIGID-FLEX STACK-UP SAMPLES

3.1 Partial coverlay stack up

3.1.1 Rigid Flex 4 Layers – Copper Foil Type

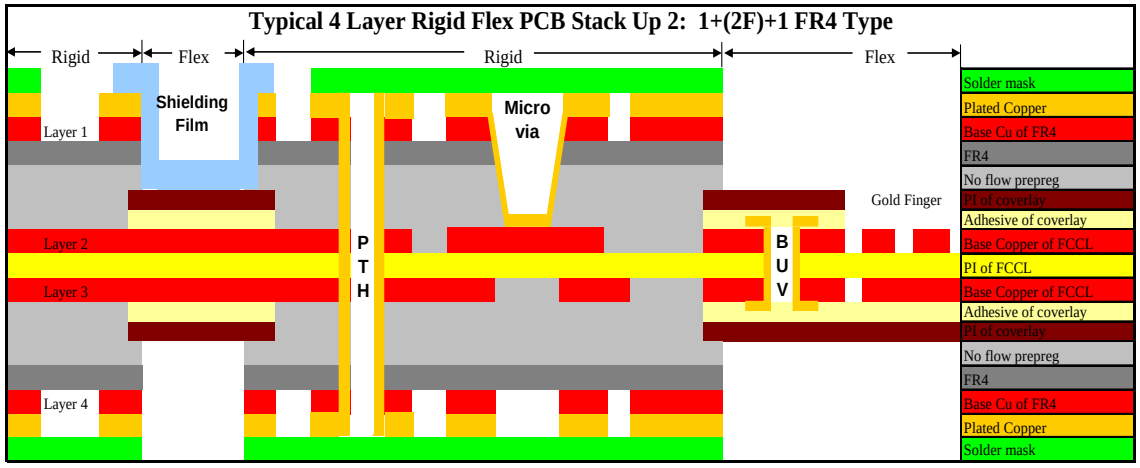
Key Features:

- Micro-via and through hole located in a combination of FR4/PI (Adhesiveless)
- BUV on "Flex areas" covered by Coverlay
- Gold finger (internal layer) can be exposed during Outer layer process
- Micro-via at outer layer only through FR4 material and aspect ratio is 0.5 max. (micro-via size = dielectric thickness x 2, maximum is 250um)
- EMI Shielding film at flex area can be offered



3.1.2 Rigid Flex 4 Layers – FR4 Type

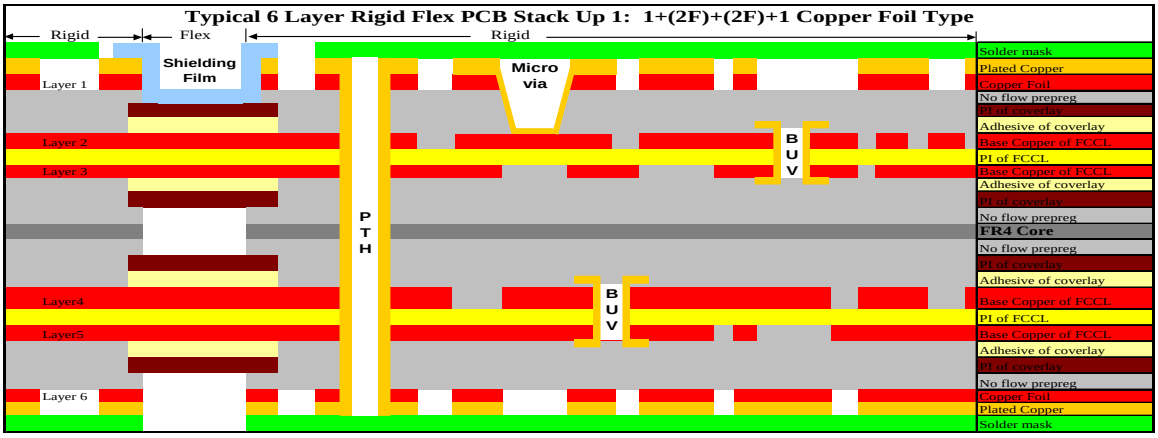
- FR4 Core thickness can be adjusted to meet desired overall thickness on the Rigid areas



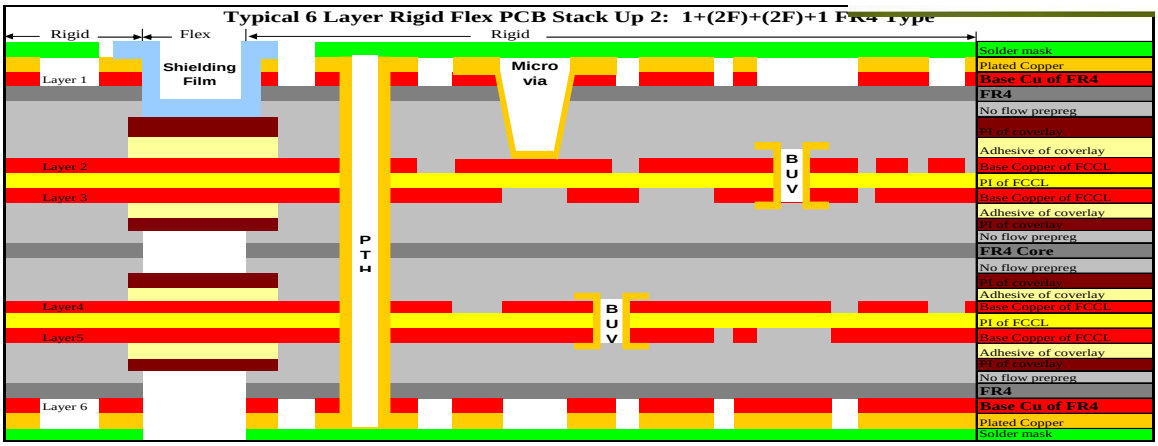
3.1.3 Rigid Flex 6 Layers – Copper Foil Type with "Air Gap" on Flex region

Key Features:

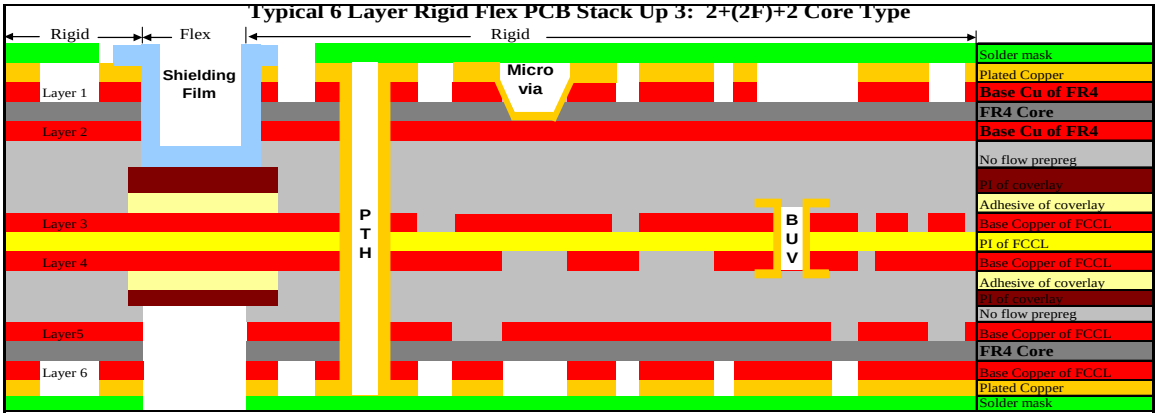
- Center FR4 core can only be removed if boards are singulated (for boards in delivery panel format, the FR4 can only be removed after assembly and singulation)
- BUV on Flex cores
- Micro-via at outer layer only through FR4 material and aspect ratio is 0.5 max. (micro-via size = dielectric thickness x 2, maximum is 250um)
- EMI Shielding film at flex area can be offered
-



3.1.4 Rigid Flex 6 Layers – FR4-Cap Type with "Air Gap" on Flex region



3.1.5 Rigid Flex 6 Layers – FR4-Core Type

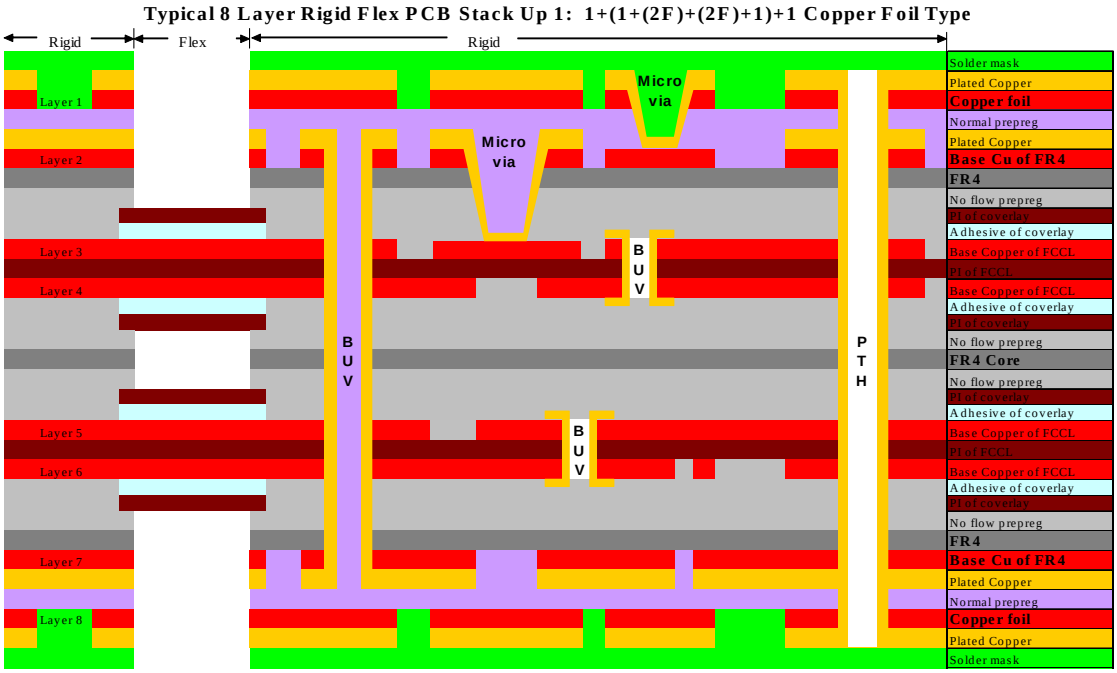


3.1.6 Rigid Flex 8 Layers – Copper Foil Type "Air Gap"

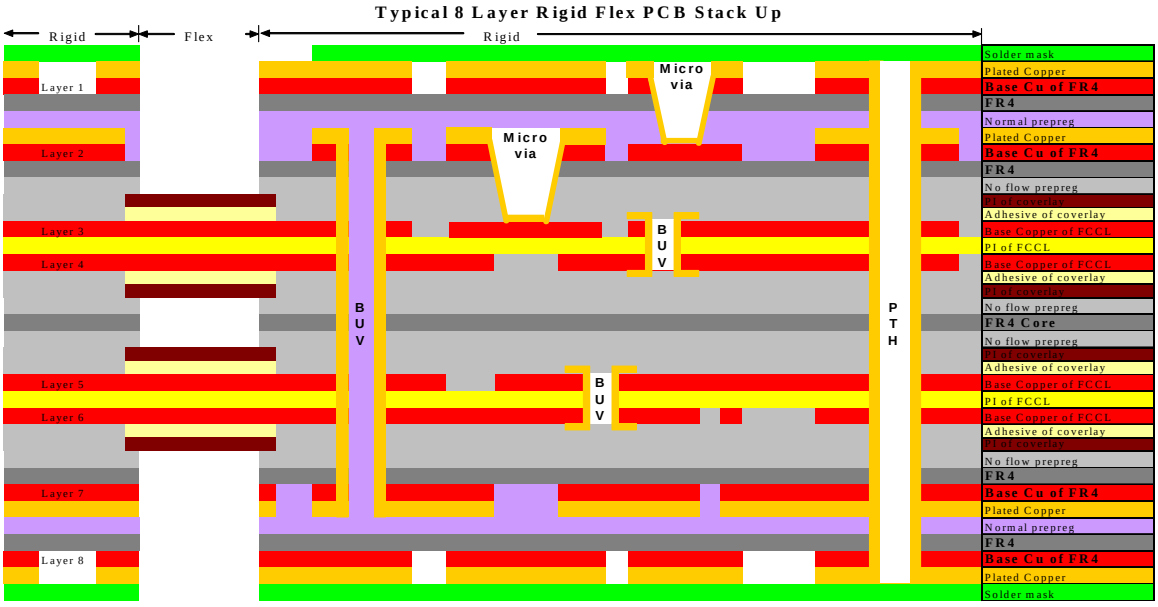
Key Features:

- Buried via on Flex & Rigid areas
- Buried via L2 to L7 are filled by standard pre-preg. L2 & L7 need to have FR4 material to facilitate this type of build-up, and by using standard pre-preg to flow into the buried vias when L1 & L8 is laminated.
- Center FR4 core can only be removed if boards are singulated (for boards in panel array format, the FR4 can only be removed after assembly and singulation)
- Micro-via on outer layer and inner-layer FR4 material areas.

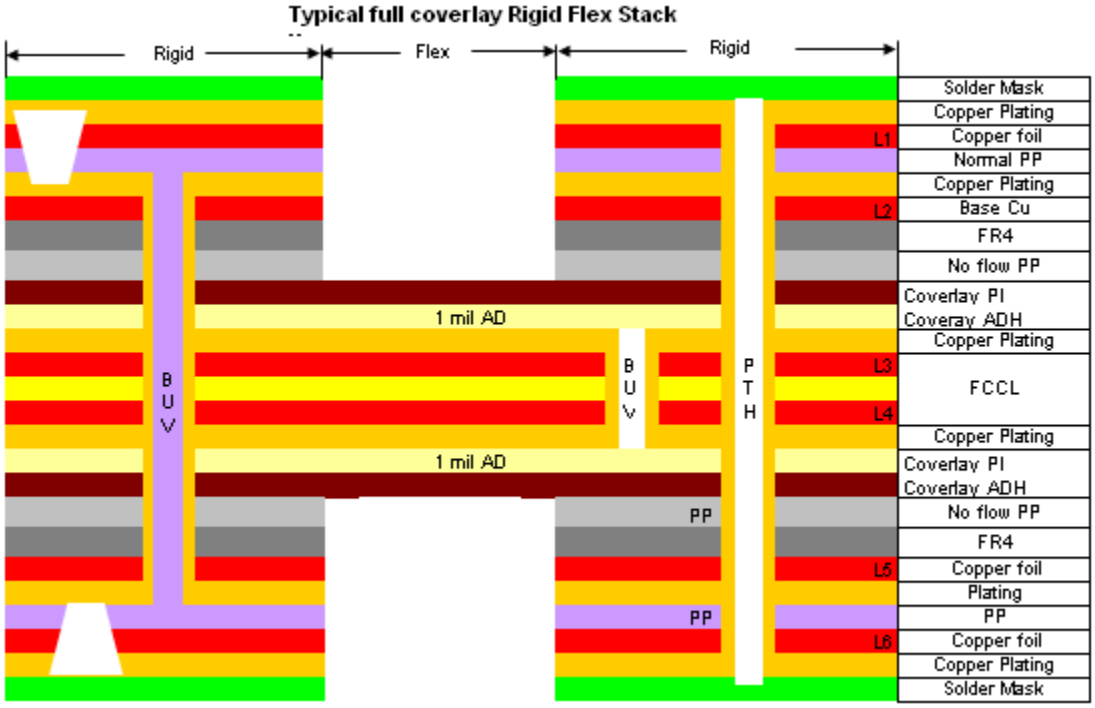
Aspect ratio is 0.5 max. (micro-via size = dielectric thickness x 2, maximum is 250um)



3.1.7 Rigid Flex 8 Layers – FR4 Type "Air Gap"



3.2 Full coverlay stack up

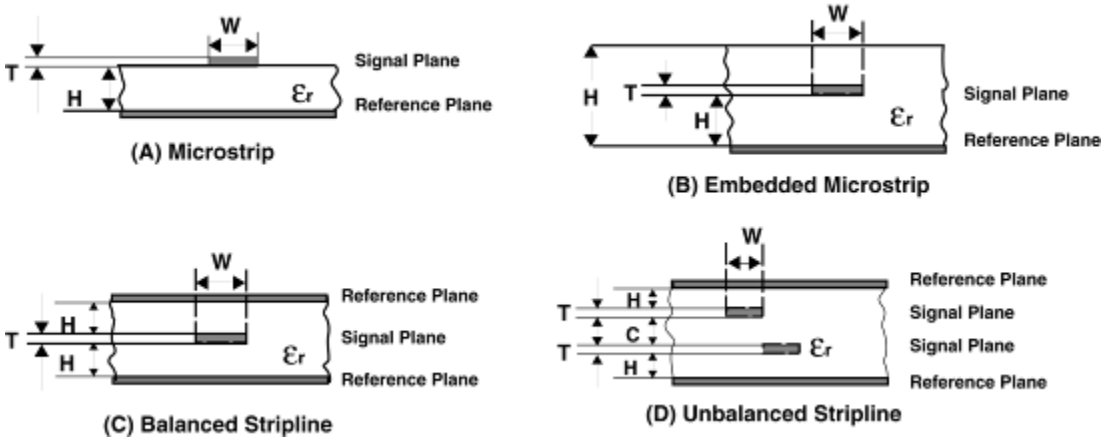


4 ADDITIONAL DESIGN INFORMATION & CONSIDERATIONS

ELECTRICAL DESIGN

-Utilize the 3rd Dimension

-Impedance Control – Single ended and Differential



CIRCUITRY IN FLEX AREA:

-Maintain Straight Conductor Routing

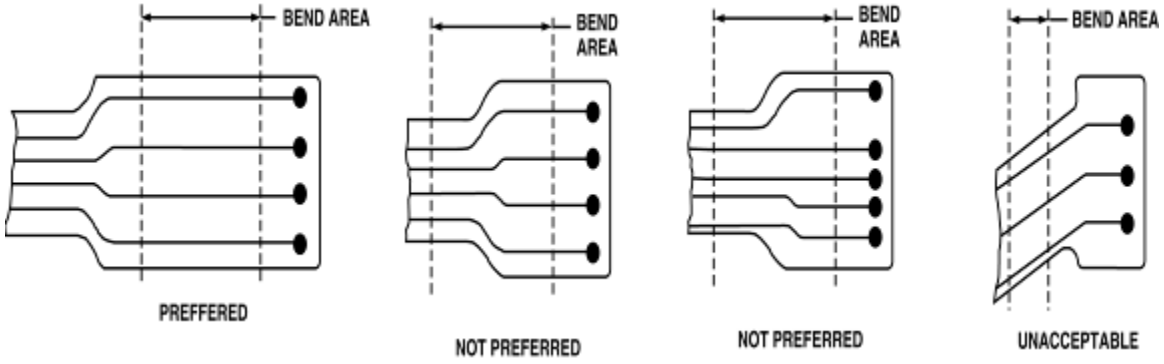
-Avoid Changes of Direction

-Conductors should be uniform in width and should run perpendicular to fold lines.

-Maximize the Conductor Width

-Utilize Radii and Curves

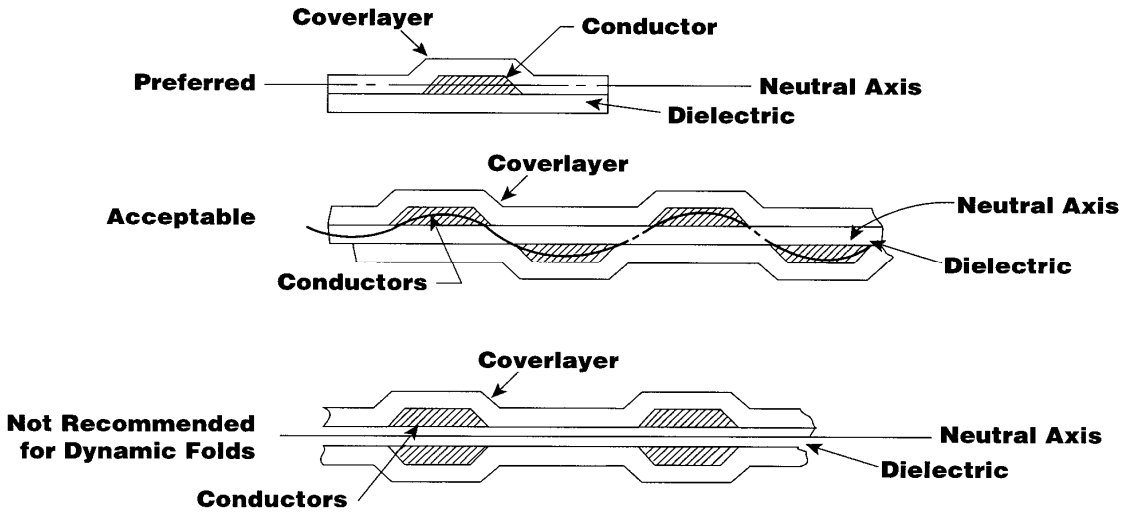
- Always add teardrops on trace/pad interface

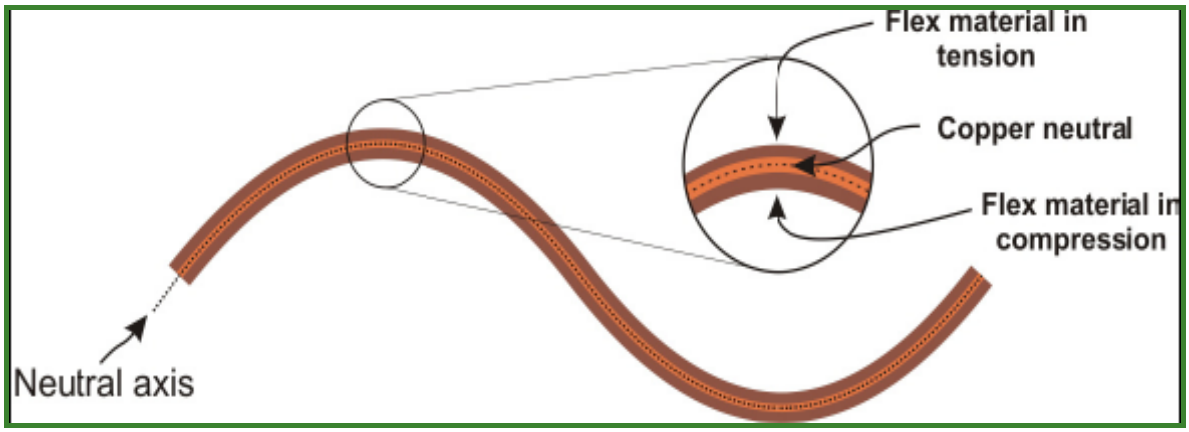


FLEX AREA – NEUTRAL AXIS:

-Staggered or offset traces on both sides, preferred for “Bend areas”.

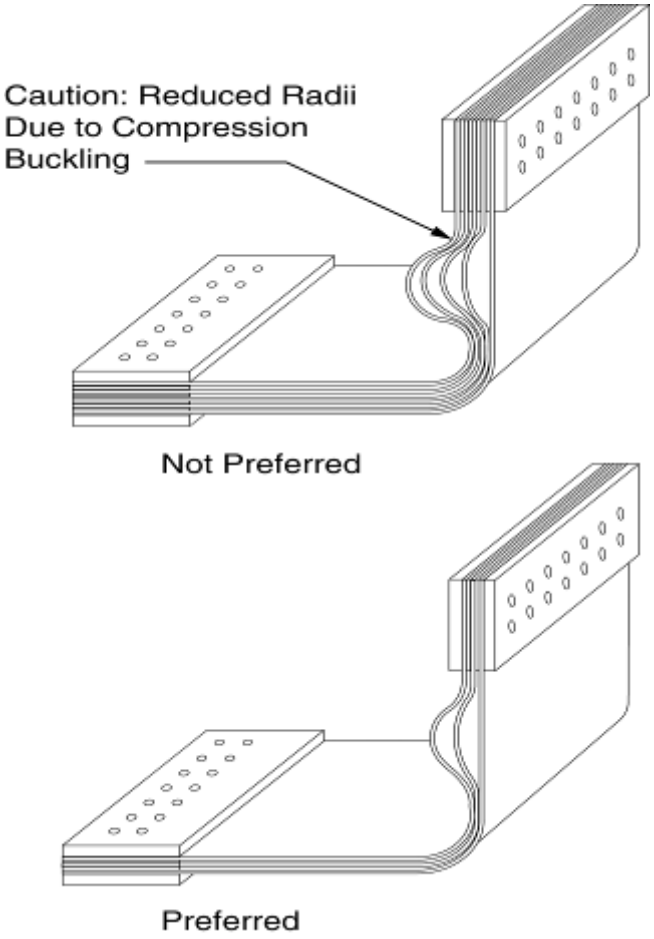
- I-Beam conductor lay-out not-recommended for bend areas.





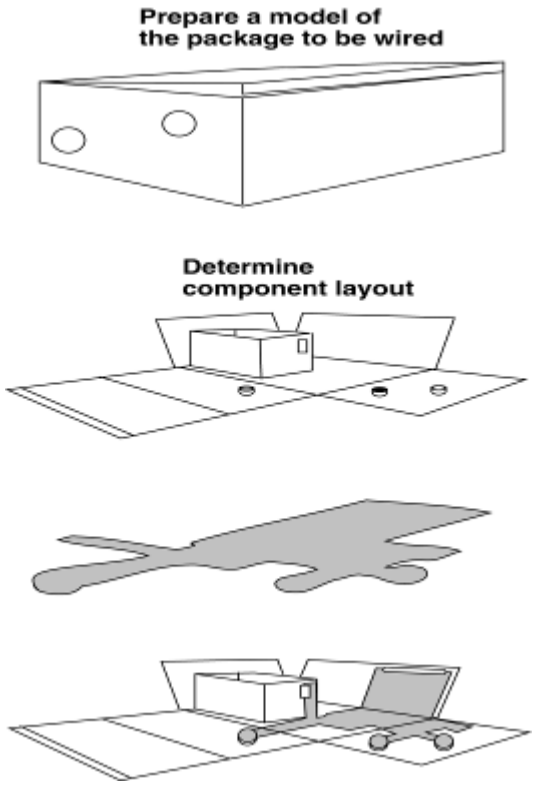
FLEX SECTION WITH MULTIPLE LAYERS:

-Forming of Flex prior to Assembly is very difficult.
-Dielectrics have high memory to stay flat.
-Design to utilize flex capabilities
-No plated through holes in Flex areas.
-Limit plating in Flex areas.



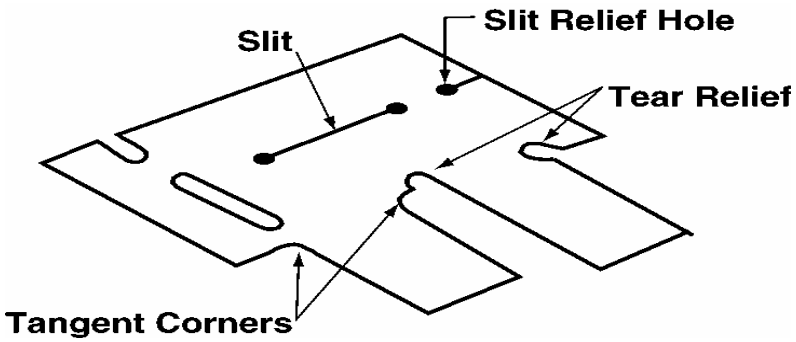
MECHANICAL DESIGN:

*Mechanical design is the most critical aspect for long term efficiency, cost and performance.
- Mock-up or Paper doll cut-out of the planned circuit can clarify the interconnection needs. Mock-up should have all the required hardware installed to make the design complete and prevent interferences.
- Bend radii and service loop needs to be considered when determining the mechanical lay-out to ensure it falls within the acceptable values of the flex circuits overall thickness.



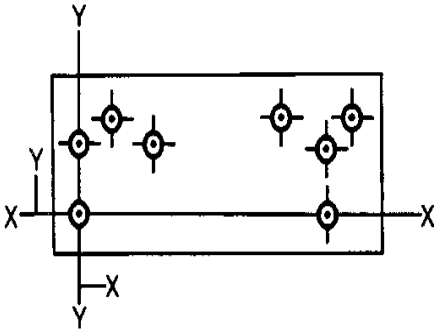
MECHANICAL PROFILE:

- If a slit is required, add a round hole to keep slit from tearing
-Radius all inside corners
-Add Copper tear restrains to inside corners

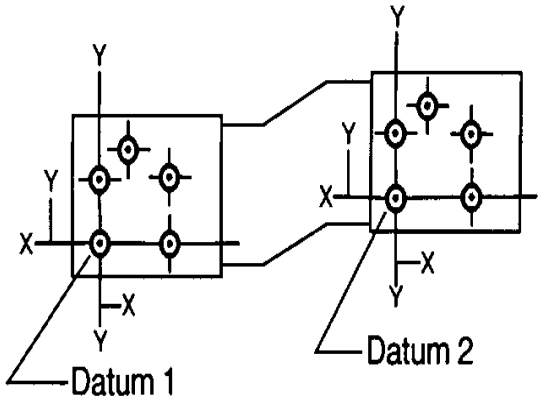


DRAWINGS:

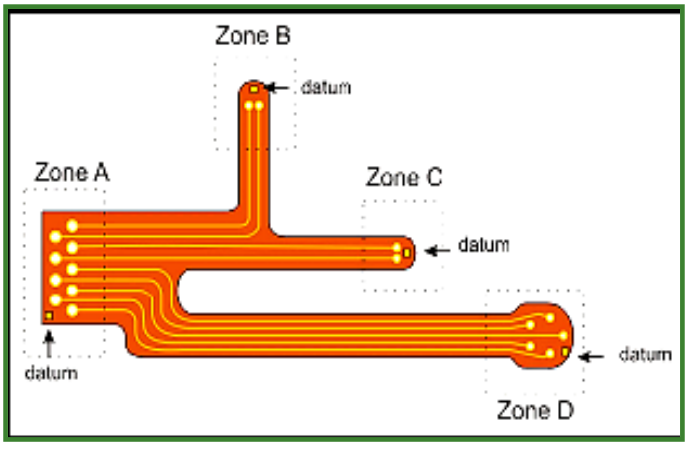
-Dimensions and fabrication prints should be clearly defined.
-Minimize dimensional drawings to only specify the needs to properly convey design intent.
-Focus on the features that matter, the areas that can cause failures.
-Provide consistent documentation to eliminate delays
-Avoid automated tolerance
-Specify generic material description
-Give performance specification. Example: *Built to IPC-6013, Type 4, Class 2, Use A.*
-All Rigid sections should have a local Datum with generous tolerance (Min. of +/- .020)



Datum lines within board
(Non-functional holes may be used)



Additional datums
(Supported or Constrained Condition)



5 Material & Panel Size

5.1 Material and PCB Thickness

Base material	Description	Capability	Recommendation
PI(Polyimide)	Adhesiveless FCCL	PI: 0.5, 1 ~ up copper; 9um, 12um, 18um	1mil PI W/ buried hole 1/3oz Cu W/O buried hole 1/2oz Cu RA copper for bending application
	Coverlay Only covered at flex area	adhesive : 0.5mil,1mil ~ up PI:0.5mil, 1mil ~ up	adhesive : 0.5mil,1mil, PI:0.5mil, 1mil
FR4 – Cores/ Prepreg	Laminate	FR4 (Tg 140 & 170) Halogen free FR4 (Tg140)	FR4 Tg 170
	Minimum core thickness	0.05mm (2 mils)	0.05-0.10mm (2-4 mils)
	No/Low Flow Pre-preg	1037(1.5 mils) 106(2.2 mils) 1080(3.0mil / 3.5mil)	1037(1.5 mils) 106(2.2 mils) 1080(3.0mil / 3.5mil)
Assistant material	Shielding film	Shielding film SF-PC5600 SF-PC5900 SF-PC6000	Shielding film SF-PC5600
	Solder mask	Halogen free PSR4000BL01	Halogen free PSR4000BL01
		Non-Halogen free PSR4000G23K	Non-Halogen free PSR4000G23K
Working panel size	Standard	12"x 18" (305X457mm) 12" X19.7"(305X500mm)	12"x 18" (305X457mm) 12"x19.7"(305X500mm)
	Board edge	14mm	15mm
Min/Max PCB thickness	Min: 0.30mm ;	Max:2.4mm	

5.2 Panel Size / Reel Size

Panel size		Usable area (w/o Board edge [FPC])	Usable area (w/o Board edge [RFPC])	Remark
inch	mm			
12" x 18"	305 x 457mm	280 x 432 mm	275 x 427 mm	minimum of 12.5mm (0.490) space on every side for Flex
12" x 19,7"	305 x 500mm	280 x 475 mm	275 x 470 mm	
18" x 24"	457 x 610mm	432 x 585 mm	427 x 580 mm	minimum of 15mm (0.590) space on every side for Rigid-Flex
19,7" x 24"	500 x 610mm	475 x 585 mm	470 x 580 mm	

Remark: These are just general rules, they will be changed if the projects have something special,

such as the components are out of the outline, stiffener lay up need more room, …

Reel Size		Usable area (w/o Board edge [FPC])	Usable area (w/o Board edge [RFPC])	Remark
inch	mm			
12"	305mm	280 mm	275 mm	single side FCCL
12"	305mm	280 mm	275 mm	double side FCCL w/o reel-to-reel drilling
19.7"	500mm	475 mm	470 mm	double side FCCL w/o reel-to-reel drilling

6 Miscellaneous

6.1 Reference Specifications

IPC - Institute for Interconnecting and Packaging Electronic Circuits supports all facets of the printed wiring board industry and has a large number of specifications and standards covering a variety of topics. The IPC is the principle source for flexible circuit design assistance and documentation. Listed below are key specifications related to flexible printed circuitry.

Number	Title
IPC-T-50	Terms and Definitions
IPC-MF-150	Metal Foil for Printed Wiring Applications
IPC-4202	Flexible Bare Dielectrics for Use in Flexible Printed Wiring
IPC-4203	Specification for Adhesive Coated Dielectric Film for Use as Cover Sheets for Flexible Printed Wiring and Flexible Bonding Films
IPC-4204	Flexible Metal-Clad Dielectrics for Use in Fabrication of Flexible Printed Wiring (Includes Slash Sheet Amendment)
IPC-4101	Specification for Base Materials for Rigid Laminates and Pre-pregs
IPC-2223	Sectional Design Standard for Flexible Printed Boards
IPC-2221A	Generic Standard on Printed Wiring Board Design
IPC-6013A	Qualification and Performance Specification for Flexible Printed Wiring
IPC-A-600	Acceptability of Printed Wiring Boards
IPC-SM-840	Qualification and Performance of Permanent Solder Mask
IPC-TM-650	Test Methods Manual

Information Resources:
IPC-2221A / IPC-2223A
IPC-6012 / IPC-6013A

6.2 Preferred Data Formats

A complete design package should always include the following:

Data Type	Preferred Format
Customer Drawings	RS 274X or DXF
Circuit Data	RS 274X ; ODB++
Drill Data	Excellon 2
Route Data	Excellon 2
Netlist Data	IPC-D-356

Complete data set with layers clearly named
-Provide board CAD rout layer (rout layer should always match drawing)
-Provide complete sub-panel, if required, which would include rout, fudicials, holes, and other features unique to the sub-panel that are listed in the drawing.

Complete and legible drawings in which the intent is clear
-Provide technology requirements for the board to include surface finish, copper thickness (clarify start or finished), coverlay or solder mask and any design requirements and tolerances.
-Provide stack-up specification

Read me file with peculiar or special features of the design clearly described
-Call out any items that are unique or critical to the design
-Special instruction requirements should be documented, list it in the read me file.
-Provide contact telephone numbers and e-mail address for the designer and buyer as applicable

6.3 Technical Contact

To contact us, please log in to: http://www.multek.com

6.4 Revision

Revision 1: May2008
Revision 2: November2009
Revision 3: February 2011