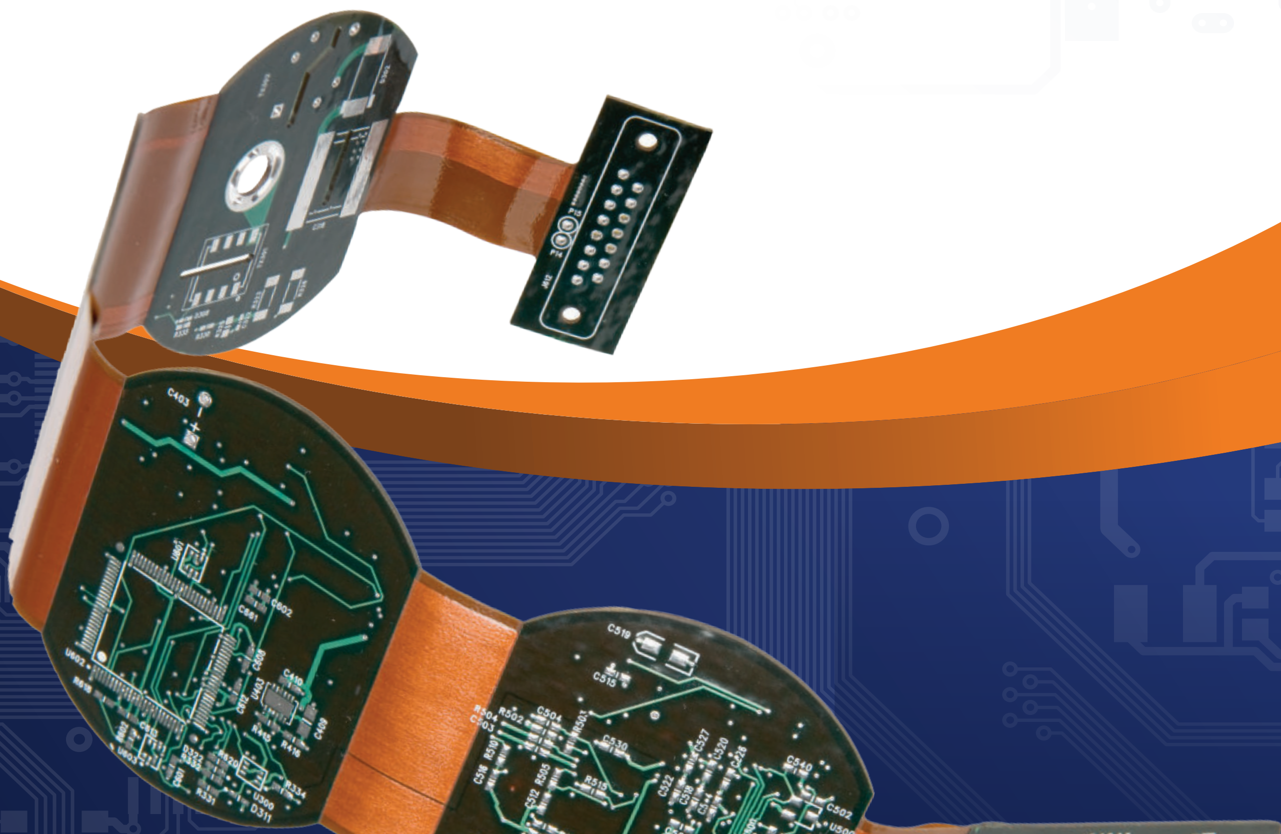
RIGID FLEX

DESIGN FOR MANUFACTURING GUIDE



8 Layer
Rigid Flex

RIGID FLEX

DESIGN FOR MANUFACTURING GUIDE

TABLE OF CONTENTS

1. SCOPE AND PURPOSE

This guide provides a tool for product development and design that emphasizes manufacturability. This guide aids the concurrent engineering process in creating optimum designs that are not only manufacturable, but also high in quality, reliability and cost effective.

This guide is intended to be used early in the design process to guide the designer towards producible and cost-effective designs. Following the rules and recommendations of this guide will lead to cost-effective designs with high yields and reliability.

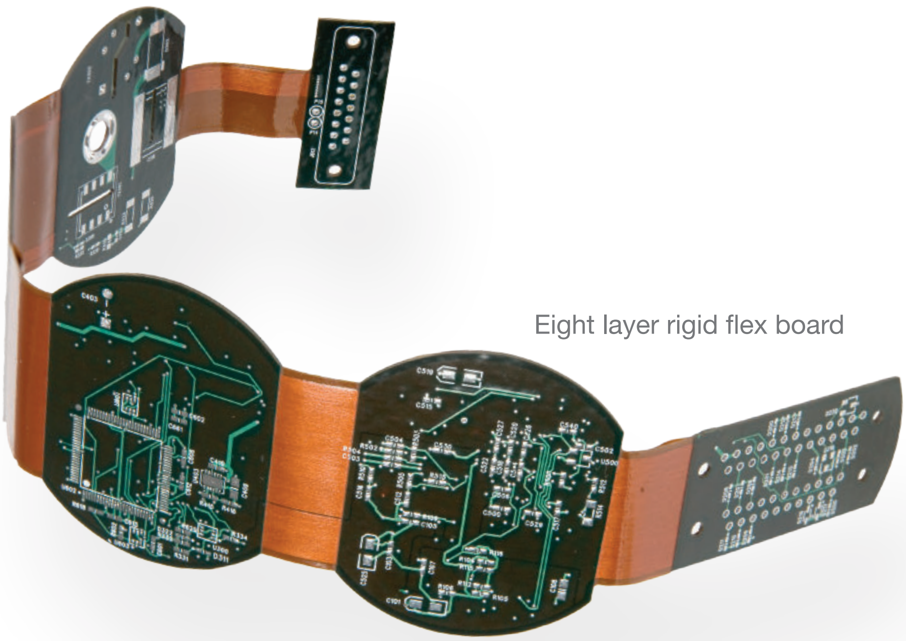
Deviation from this Design for Manufacturing (DFM) Guide will result in one or more of the following: greater number of manual operations, longer manufacturing cycle times, increased rework, higher scrap, lower yields, and greater risks to quality and reliability. Each of these items contributes to increased costs.

Ten layer rigid flex board

2. RIGID FLEX DEFINITION

A rigid flex board is a combination of hardboard(s) and flexible circuit(s), forming a hybrid circuit board that can be assembled similar to a rigid board, but has flexible areas allowing the board to be folded into place, or used in a continuously flexing application (dynamic flex).

The principal differences between a rigid flex board, and a flex board with stiffener, is that the rigid flex board will have plated through holes in the rigid areas of the board, making connections between layers including the flex layers. A rigid flex board can also have components mounted to both sides of the board in a density and complexity that is similar to hardboards.



Eight layer rigid flex board

3. RIGID FLEX APPLICATIONS

Rigid flex boards are more expensive than hardboards (usually about 7 times the cost of an equivalent hardboard) and flex boards (usually 2 to 2.5 times the cost of an equivalent flex board), but there are times when the increased cost is justified:

- High reliability applications, such as high shock and high vibration environments where flex cables and their connectors will fail.
- High density applications, where there is just not enough real estate on the circuit boards, or room within the assembly for connectors and cables.
- Assemblies with four or more rigid boards connected with flex cables. On these assemblies rigid flex can sometimes offer a lower cost option when all costs are considered.

4. RIGID FLEX TYPES

There are two main types of applications for rigid flex circuit boards. The predominant application is called flex to install. The bare board is manufactured, assembled with components, removed from the supporting array, and then folded to fit within the device.

The second application is less common and is called dynamic flex. In this application the flexible area(s) of the boards will be flexed back and forth throughout the life of the circuit board. Boards that undergo a life of continuous flexing, can be expected to have a service life of hundreds of thousands of flex cycles, without failure, as long as the recommended minimum bend radius requirements are observed and met (see 7.6 Minimum Bend Radius Guidelines).

Also note, that sometimes a board that is intended for a flex to install application, is actually going through a dynamic flex cycling because of the environment or requirements. High vibration environments will create dynamic flexing, out of a flex to install application. High(er) current carrying capacity can have the same affect, by causing the flex to expand and contract repeatedly during on/off cycling.

5. RIGID FLEX ADVANTAGES AND DISADVANTAGES

ADVANTAGES:

- Excellent survivability in high shock applications
- Excellent survivability in high vibration environments
- Opportunity to reduce overall package weight
- Opportunity to increase package density
- Opportunity to incorporate dynamic flex connections offering hundreds of thousands of flex cycles without failure
- Fewer assembly operations on boards with multiple flex interconnections

DISADVANTAGES:

- Higher cost
- Lower manufacturing yields
- Longer manufacturing cycles compared to rigid and flex boards

6. RIGID FLEX MANUFACTURING PROCESS FLOW

Rigid flex boards use the same manufacturing techniques as hardboards and flex boards but with much greater attention to material scaling, and with greater emphasis on conservative processing techniques that generally are slower, to accommodate the different material properties between glass reinforced materials and flexible materials.

Rigid flex manufacturing is also not a linear process flow like hardboards, always requiring at least one subassembly through lamination, and often more.

A SIMPLIFIED RIGID FLEX PROCESS FLOW CHART IS BELOW:

7. RIGID FLEX DESIGN

Rigid flex designs are very similar to hardboards. A fabrication package should consist of artwork Gerber files for each layer, drill(s), soldermask, nomenclature, etc. The flex layers go all the way through the rigid area, just like conventional hardboard layers.

Rigid Flex is a combination of Rigid and Flexible substrate materials laminated to form a single printed circuit board outline which can then be assembled and formed into a 3 dimensional subassembly which includes all interconnections.

7.1 DESIGN REQUIREMENTS:

General design requirements should be in accordance with IPC-2223. Sectional Design Standard for Flexible Printed Boards.

The principal differences between a rigid flex fabrication package and a hardboard fabrication package are:

- The print must accurately define the dimensions for the rigid flex transition areas, which are not always apparent by viewing the Gerber files.
- The material layup should be carefully defined for each section, with more attention to detail.
- You will have soldermask layers for layers 1 and X, but you will also have coverlayer artwork for the outside of the flex layers, and in the case of multilayer flex sections you will have artwork layer(s) for the bondply that bonds the flex layers together.

7.2 MATERIAL LAYUP OVERVIEW:

Board stackup is critical to both manufacturability and final performance. It is essential that the PWB designer work closely with the fabricator to determine the correct material selection, physical stackup and to ensure that signal integrity requirements are met. Controlled impedances, resistance and current carrying requirements will all impact copper weight and therefore stackup. As soon as the initial stackup calculations are available they shall be forwarded to the fabricator for confirmation of impedance values, materials, delivery and estimated cost.

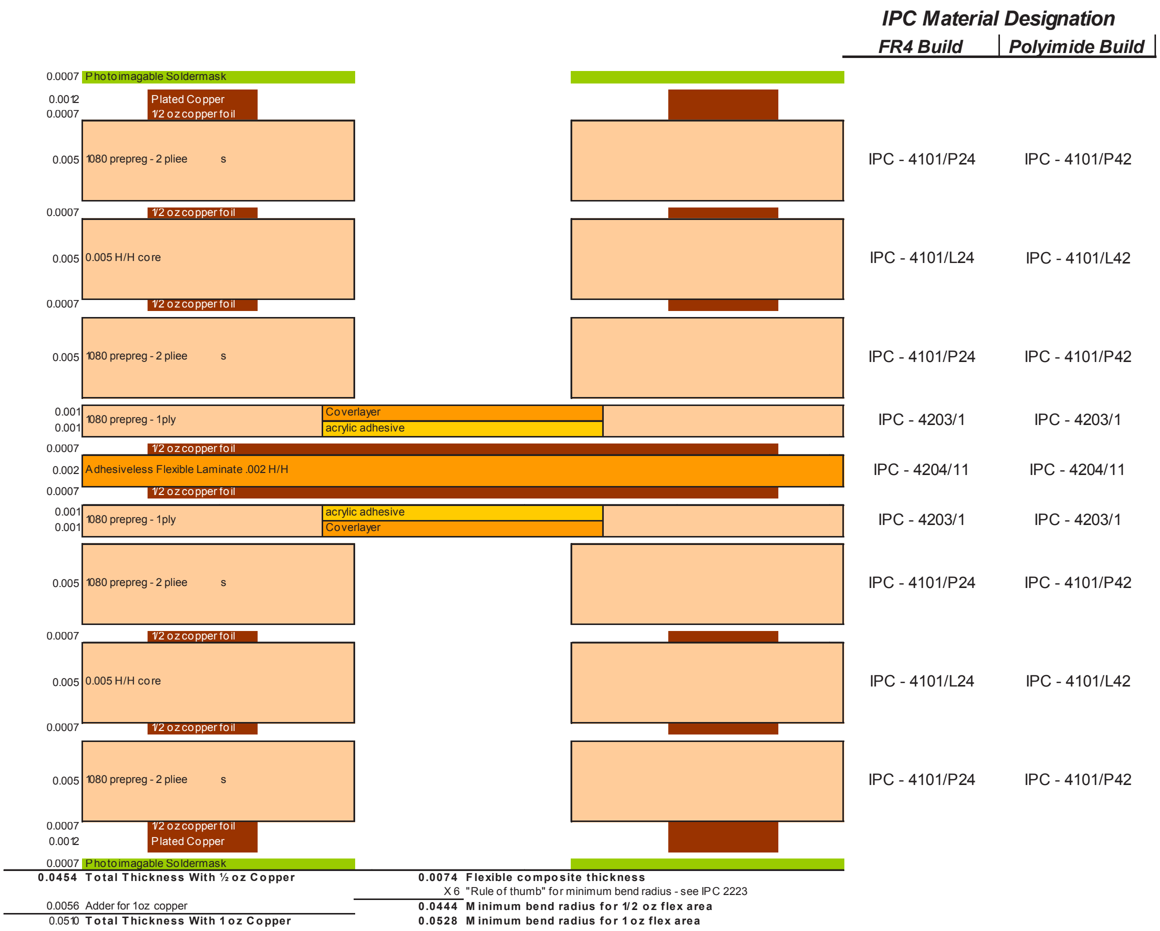
Rigid flex designs are usually 20 layers or less, though sometimes the designs have higher layer counts.

Rigid flex designs usually do not have the same number of layers in all of the hardboards. For example, you could have a 14 layer board with one rigid board at 14 layers, another at 10, and two at four layers. However, it is important that hardboards all be the same thickness.

It is possible to design and fabricate rigid flex boards, where the hard boards are at varying thicknesses, but manufacturing yields will go down appreciably depending on the design, and some designs are not manufacturable at all.

The flexible sections are typically single layers (singlets), two layers (doublets), three layers (triplets) or four layers (quads). More than four layers is very unusual as the flex composite becomes very stiff and loses flexibility for most applications.

An example of an 8 layer rigid flex with a doublet is below including IPC slash sheet numbers, thicknesses for each component, overall thicknesses for the rigid section and the flex section, with minimum bend radius calculations (see 7.6 Minimum Bend Radius Guidelines).

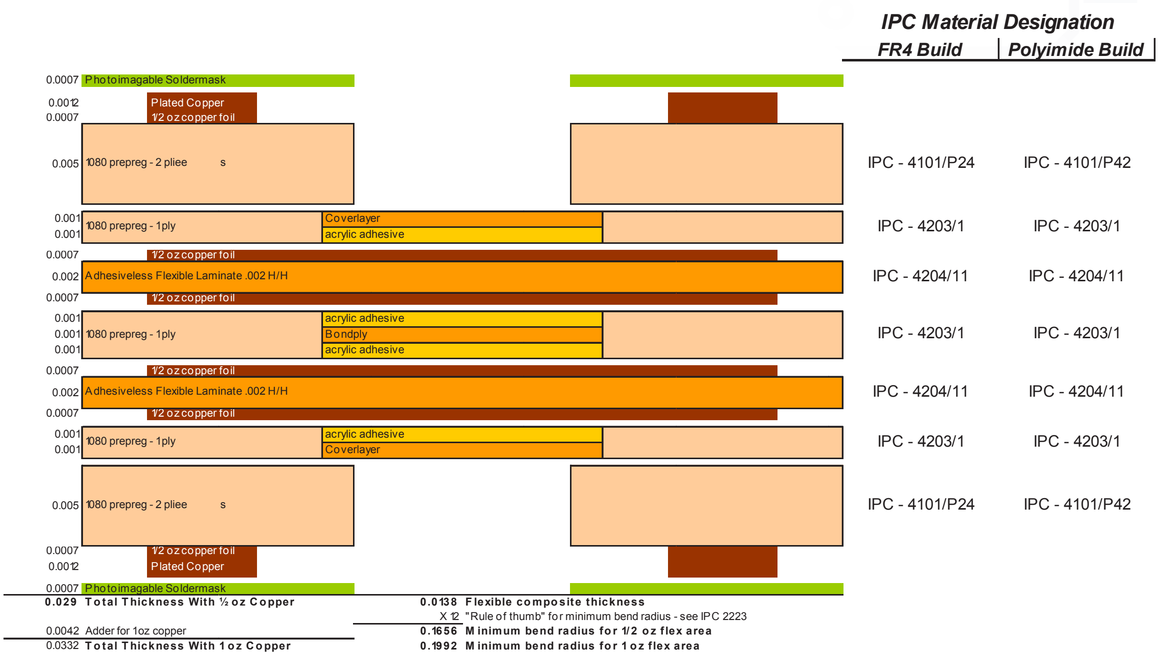


7.3 CUT BACK COVERLAYER AND BONDPLY:

High reliability rigid flex designs almost always use a technique called cut back coverlayer in their construction (see IPC 2223 5.2.2.2). The acrylic adhesive in the coverlayer has a much higher z axis CTE rate than the other materials, which if included in the hardboard package, will exert z axis force on the vias during thermal excursion, such as lead free assembly temperatures.

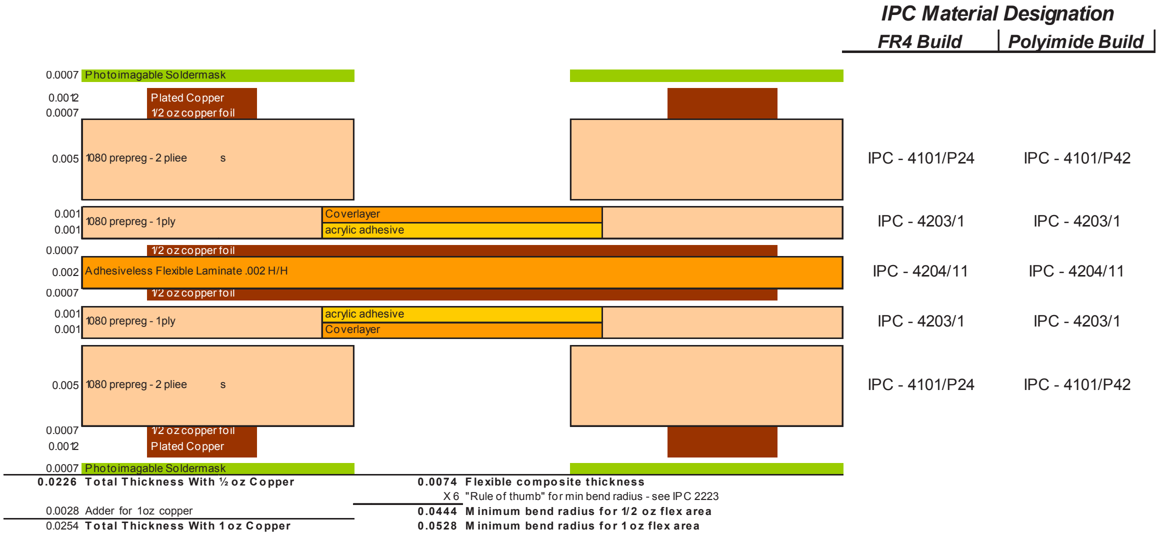
To address this issue, rigid flex fabricators allow the coverlayers and bondplies to go .050" into the hardboards and no further. The coverlayer and bondplies are then mated with a sheet of prepreg of roughly the same thickness, as shown in the material layup below. This area then becomes a "keep out" area for vias, to prevent drilling through the coverlayer in the hardboard sections (see IPC 2223 5.2.2.3).

An example of a 6 layer rigid flex with a quad is shown below. Note the bondply that is used to glue the two flex laminates together.



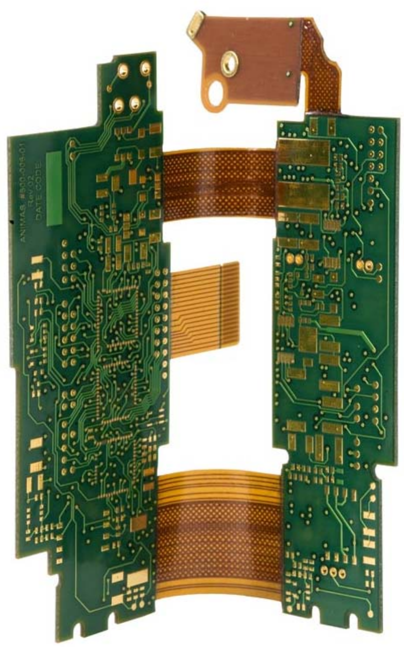
7.4 TYPES OF RIGID FLEX:

There are two main categories of rigid flex PWB's. The most common is where every flex arm terminates in a rigid board. An example of that construction is shown below. These designs are very straightforward, high(er) yielding, and progress through the manufacturing sequence relatively quickly and without issue, and at lower cost.

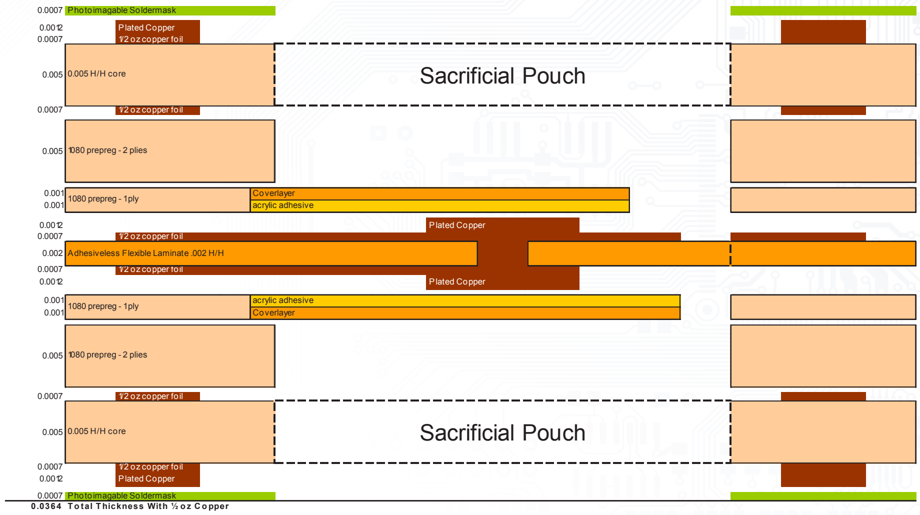


The second type is where one or more arms terminate in a flex arm without a hardboard on the end. These are usually flex pigtails that often have stiffeners applied on the ends. In these boards (example shown at left), the manufacturer builds the internal flex layers to completion. They are themselves a complete PWB in their own right.

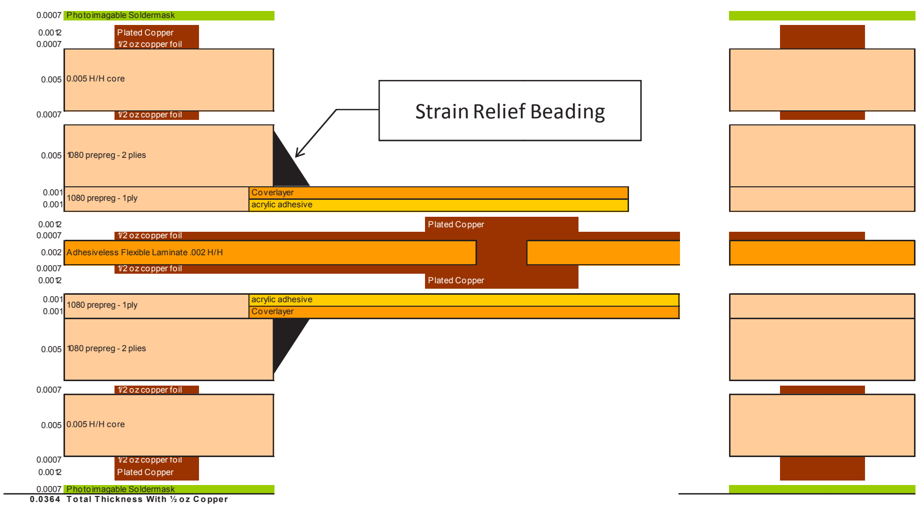
The manufacturer then "builds" the outer layer structure around the internal flex layers, and embeds the internal flex layers in what is called a pouch. The pouch protects the internal flex layers during outer layer processing. The pouch is then removed at the conclusion of manufacturing.



Eight layer rigid flex, with the center flex arm terminating in a flex with stiffener rather than hardboard



These designs are slow in manufacturing for a number of reasons. First, is that they are akin to building two boards - a board within a board. Second, the pouch has to be removed by hand, and on both the top and bottom sides of the board. Third, removing the pouch leaves a glass fabric edge, which for high reliability applications, should be covered with a strain relief fillet to prevent the flex from abrading against the glass fabric edge. Strain relief fillets, or beading, is also done by hand and requires a day to air dry. Beads or beading is illustrated below:

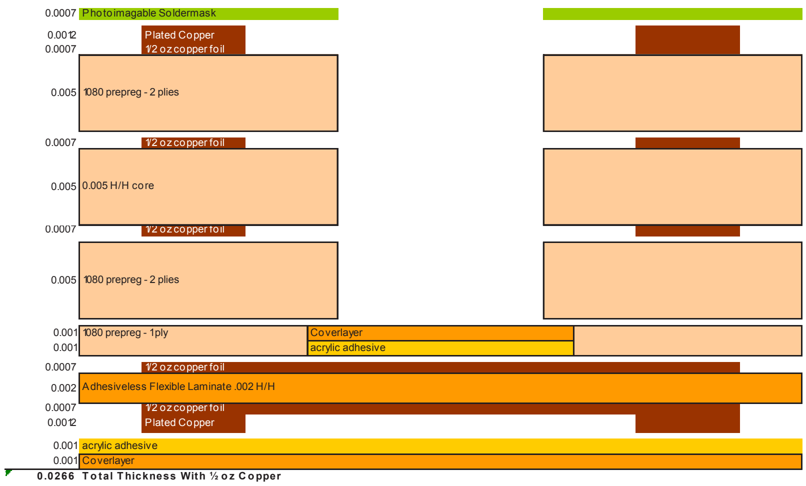


There are some other considerations when designing a pouch construction rigid flex.
Pouch constructions always have laminate on the outerlayers. Fabricators cannot do a pouch construction using copper foil on the outerlayers. The external laminate material needs to be more than .005” thick to withstand outerlayer processing. Thinner laminates cannot survive the vacuum pressure of plasma and subsequent outerlayer processing. They will break down, allowing chemistry into the pouch and ruining internal layers of the board. The outer laminate combined with the prepreg thickness needs to be at least .020” (from surface of rigid sections to outer surface of cover layers) thick to accommodate the strain relief beading – see section 9.3.
The beading material has a very thin viscosity and adheres to the rigid flex transition area using surface tension. If this area of the rigid to flex transition area is too thin, the beading material will run up onto the outerlayers of your PCB.

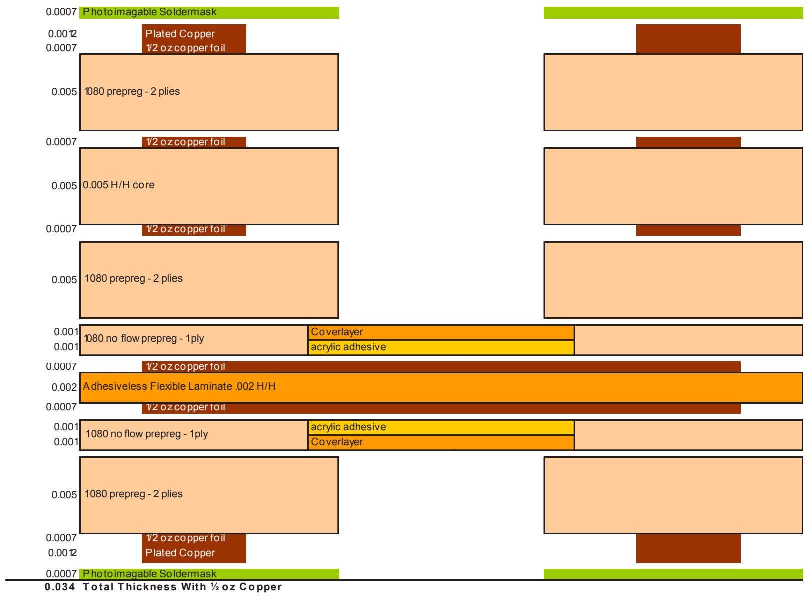
7.5 ASYMMETRICAL AND OTHER RIGID FLEX CONSTRUCTIONS:

Asymmetrical rigid flex designs are not recommended (IPC 2223 A.8.1). They are prone to warp both in bare board manufacturing and in assembly. They typically have high fallout in outer layer yields and sometimes can't be manufactured at all.

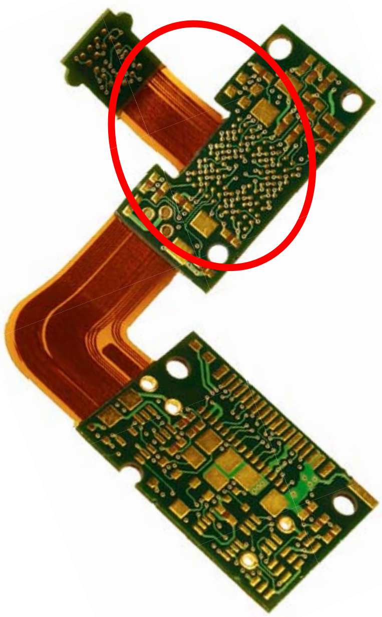
Two examples of asymmetrical designs are shown below. The first one shows the flex layers on the outer layers of the board. These designs are prone to warp and are also difficult to image and etch faithfully, as the outer layer goes across the flexible portion of the board. These designs typically have significant yield loss – 40% to 50% due to this feature.



The second shows flex layers within the material layup, not on external layers. These types of boards will not have the imaging issues that the first one has, but nonetheless are prone to warp and yield issues.



7.6 MINIMUM BEND RADIUS GUIDELINES:



High reliability flex circuits have minimum bend radius guidelines to ensure long life whether in dynamic (continuous flexing) or flex to install applications (bent once to install). Even in flex to install applications, the flexible areas of the board may be exposed to dynamic flexing (see 4.1 Rigid Flex Types). Care should be taken to follow the minimum bend radius guidelines (originally derived from Mil-Std-2118 and DuPont Technical Data Sheet on Minimum Bend Radius Guidelines for Pyralux Materials).

The minimum bend radius requirement for single and double sided flex sections of rigid flex PWB's is 6X the composite material thickness of the flexible area of the board (laminate, plus copper, plus coverlayer).

The minimum bend radius requirement for multilayered flex sections of three or more layers of copper is 12X the composite material thickness of the flexible area of the board (laminate, plus copper, coverlayer and bondply).

Note, on multilayered flex designs, it is usually best to put the circuit layers internally and the ground/power planes on the external layers of the flexible areas of the boards. This reduces the stress on the circuits on external layers, and on the outside of the radius bend, which is typically the most stressed layer of the construction.

Cross hatched ground/power/shielding planes can improve flexibility in flex designs, but need to be accommodated in impedance modeling for controlled impedance designs.

There are ways to increase the minimum bend radius within a design. A common practice is to rout slots on each side of the flex arm so that the rigid to flex transition area is moved back from the board edge. An illustration is shown to the above/left.

Another method, although less common and harder to manufacture, is to have the flex peel away from the hardboard prior to the edge of the board. A photo of that technique is shown below/left. The flex portion is peeling away from the hardboard on the bottom side of the board. It does create other manufacturing issues, but can be done when relief is needed for a minimum bend radius.

A third method, which is rarer still and much more expensive, is the bookbinder rigid flex. In bookbinder boards, the outer flex sections are increased in length compared to the inner flex sections, creating a hump when the board is flat. When the board is flexed to its final position, stress on the outermost layers is eliminated (when sized properly). This technique is helpful when there are many conductors and layers in the flex area and there is no other way to reduce the stress on the outer layers.

Bookbinders are extremely expensive, have very long manufacturing lead times, are poor yielding, have their own set of design rules, and require custom tooling which can also be expensive. A photo of a 26 layer bookbinder rigid flex is above.

7.7 CONTROLLED IMPEDANCE GUIDELINES:

Rigid flex circuit boards can have controlled impedance traces, similar to rigid and flex boards. Circuits needing impedance control can be modeled on Polar Speedflex software, and should be checked by the fabricator prior to build, and usually prior to quote as well. Check with fabricator for material Dk values, or ask for their material library if you already have Polar Instruments Speedflex software.

If you do not have Polar's Speedflex software, a free impedance modeling program is available from Saturn PCB Design at https://www.Saturnpcb.com/pcb_toolkit.htm, for limited impedance calculations.

Any controlled impedance traces that are in the flexible as well as the rigid sections of the board, must be modeled in Speedflex separately. That also means the designer usually must modify the trace dimensions to accommodate the requirements in each section of the board. For example, a .006" trace in the rigid section might need to be modified to .007" to achieve the same impedance value in the flexible section as the rigid section. Additionally, the transition point from one dimension to the other should be placed .050" within the rigid board to reduce stress on the circuits in the transition area.

Impedance controlled circuits should be clearly defined in a table on the print - showing layer number, type, value desired and the trace/spacing geometries - for the fabricator. The Polar Speedflex calculation models should also be included within the fab package if available. It is also helpful if you "mark" impedance controlled circuits with a unique number for identification for the fabricator. For example, if your calculator recommends a .005" trace to meet your values, make the circuits requiring impedance control .0051" so the fabricator can easily find the affected traces for adjustment. Differential paired traces are usually easy to recognize; single ended characteristic traces are not. Using an unusual size helps the fabricator find your circuits.

Care should be taken to avoid over specifying impedance requirements particularly in rigid flex. Each value must be tested in manufacturing test coupons that are included in every panel in the lot. Each value also has a twin, in a separate coupon in case anything goes wrong with the first coupon. And, values that transition from the rigid to flex areas, must be modeled for each area. With that many values, the impedance test coupons grow quickly, pushing parts off of the production panel and increasing your cost per part. It is important to specify those values that you need and limit impedance requirements as much as possible.

Controlled impedance circuits within the flex sections of the boards usually require thicker flex dielectrics to make the values required. Sometimes .004", .005" or .006" thick laminates are required to make the values work. Thicker flex laminates are less flexible, and cost quite a bit more than standard .001", .002" and .003" laminates. Adhesiveless flex laminates follow an exponential price curve as thickness increases, and are often special order materials. Care should be taken during design to avoid these materials if possible.

8. RIGID FLEX MATERIALS SELECTION

Rigid flex manufacturers have to use prepreg, to keep the resin from flowing out onto the flexible area of the PWB.

Many of the most popular laminate systems do not come with a prepreg – isola's FR408, 370HR, and Nelco laminates are all examples of systems that are not offered with prepreg.

Rigid flex manufacturers start with a prepreg that is compatible within their processing window and then often choose the corresponding laminate system, so should a problem occur, they are only working with one vendor.

Some popular laminate systems with their corresponding prepregs are shown below, in a table comparing their mechanical properties. The ones highlighted in blue we use the most often, so there is a greater likelihood of having them in stock. The materials not highlighted in blue, need to be ordered in.

Polyimide Products:	Manufacturer	Product	Type	Applicable Slash Sheet	Tg	Td
	Showa Denko	MCL-I-671	Laminate	/42	200 - 240°C	330 - 350° C
	(Hitachi)	GIA-671N(n)	No Flow Prepreg	/42	200°C	330 - 350° C
	Arlon	85N	Laminate	/40 and /41	250° C	>407° C
		35N	Laminate	/40 and /41	250° C	>407° C
		38N	Low Flow Prepreg	/42	200° C	>311° C
	isola	P95	Laminate	/40 and /41	260° C	416° C
		P25N	No Flow Prepreg	/42	200° C	383° C

FR4 Products:	Manufacturer	Product	Type	Applicable Slash Sheet	Tg	Td
	isola	FR406	Laminate	/21, /24 and /26	170° C	300° C
		FR406N	No Flow Prepreg	/21, /24 and /26	170° C	300° C
	Arlon	45N	Laminate	/26	175° C	
		51N	Low Flow Prepreg	/126	170° C	>350° C

* Laminate/no flow prepreg systems highlighted in blue are our most common materials and readily available. Items not highlighted in blue, need to be ordered in.

8.1 LAMINATE MATERIALS:

Showa Denko Hitachi's MCL-671/GIA-671 is an excellent polyimide system for rigid flex PWB's. You might also choose isola's FR 406 for lower layer count, and lower cost rigid flex constructions.

Core constructions are below. Two ply constructions are preferred, and for the thicker cores to still specify thinner glass fabrics. 7628, 7629, 2116 glass fabrics etc. can cause excessive drill wander and wear in typical rigid flex constructions.

Copper weights greater than 2 ounce should be avoided on rigid flex constructions. It is very difficult to get prepreg to fill adequately on 3 ounce coppers and above, resulting in voids during lamination and electrical shorts at electrical test and in the field. Consult your fabricator for material layup recommendations to accomodate 3 ounce copper features.

Materials marked in blue are the most common thicknesses.

8.1.1 Showa Denko Hitachi MCL-I-671N Polyimide (IPC 4101/42L)

MCL-I-671N laminate

DIELECTRIC THICKNESS			Glass Construction	Dk						Df					
INCHES		MM		(100MHz)	(500MHz)	(1GHz)	(2GHz)	(5GHz)	(10GHz)	(100MHz)	(500MHz)	(1GHz)	(2GHz)	(5GHz)	(10GHz)
0.0025		0.06	#1080x1ply	4.33	4.27	4.25	4.23	4.20	4.16	0.0101	0.0109	0.0110	0.0124	0.0132	0.0145
0.0030		0.08	#1080x1ply	4.31	4.25	4.23	4.21	4.18	4.14	0.0102	0.011	0.0111	0.0125	0.0133	0.0146
0.0040		0.10	#1080x2ply	4.33	4.27	4.25	4.23	4.20	4.16	0.0101	0.0109	0.0110	0.0124	0.0132	0.0145
0.0050		0.13	#1080x2ply	4.31	4.25	4.23	4.21	4.18	4.14	0.0102	0.011	0.0111	0.0125	0.0133	0.0146
0.0055		0.14	#1080x2ply	4.31	4.25	4.23	4.21	4.18	4.14	0.0102	0.011	0.0111	0.0125	0.0133	0.0146
0.0060		0.15	#2116x1ply+#1080x1ply	4.36	4.30	4.28	4.26	4.23	4.19	0.0100	0.0108	0.0109	0.0123	0.0131	0.0144
0.0062		0.16	#2116x1ply+#1080x1ply	4.36	4.30	4.28	4.26	4.23	4.19	0.0100	0.0108	0.0109	0.0123	0.0131	0.0144
0.0070		0.18	#2116x1ply+#1080x1ply	4.34	4.28	4.26	4.24	4.21	4.17	0.0101	0.0109	0.0110	0.0124	0.0132	0.0145
0.0080		0.20	#2116x2ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0082		0.21	#2116x2ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0090		0.23	#2116x1ply+#1080x2ply	4.34	4.28	4.26	4.24	4.21	4.17	0.0101	0.0109	0.0110	0.0124	0.0132	0.0145
0.0095		0.24	#2116x1ply+#1080x2ply	4.34	4.28	4.26	4.24	4.21	4.17	0.0101	0.0109	0.0110	0.0124	0.0132	0.0145
0.0100		0.25	#2116x2ply+#1080x1ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0106		0.27	#2116x2ply+#1080x1ply	4.38	4.32	4.30	4.28	4.25	4.21	0.0099	0.107	0.0108	0.0122	0.013	0.0143
0.0110		0.28	#2116x2ply+#1080x1ply	4.38	4.32	4.30	4.28	4.25	4.21	0.0099	0.107	0.0108	0.0122	0.013	0.0143
0.0120		0.30	#2116x3ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0130		0.33	#2116x2ply+#1080x2ply	4.34	4.28	4.26	4.24	4.21	4.17	0.0101	0.0109	0.0110	0.0124	0.0132	0.0145
0.0140		0.36	#2116x2ply+#1080x2ply	4.34	4.28	4.26	4.24	4.21	4.17	0.0101	0.0109	0.0110	0.0124	0.0132	0.0145
0.0150		0.38	#2116x2ply+#1080x3ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0160		0.41	#7629x1ply+#2116x2ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0180		0.46	#2116x4ply+#1080x1ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0200		0.51	#7629x1ply+#2116x3ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0240		0.61	#7629x2ply+#2116x2ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0250		0.64	#7629x2ply+#2116x2ply+#1080x1ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0280		0.71	#7629x2ply+#2116x3ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0300		0.76	#7629x2ply+#2116x3ply+#1080x1ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0310		0.79	#7629x3ply+#2116x2ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0400		1.02	#7629x4ply+#2116x2ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0500		1.27	#7629x5ply+#2116x2ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0580		1.47	#7629x6ply+#2116x3ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0620		1.57	#7629x7ply+#2116x2ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142
0.0930		2.36	#7629x11ply+#2116x3ply	4.39	4.33	4.31	4.29	4.26	4.22	0.0098	0.0106	0.0107	0.0121	0.0129	0.0142

8.1.2 isola FR 406 (IPC 4101/24L)

Material construction availability of isola FR 406 is similar to the Showa Denko Hitachi materials shown above.

8.2 PREPREG MATERIALS:

In rigid flex, there are only two glass fabrics available - 1080 which presses out around .0025” and 106 which presses out around .002”. Prepreg is usually the number one cost driver in most rigid flex designs and 106 carries about a 50 percent premium over 1080 pricing.

8.2.1 Hitachi GIA-671N(N) (IPC 4101/42P)

GIA-671N(N) prepreg

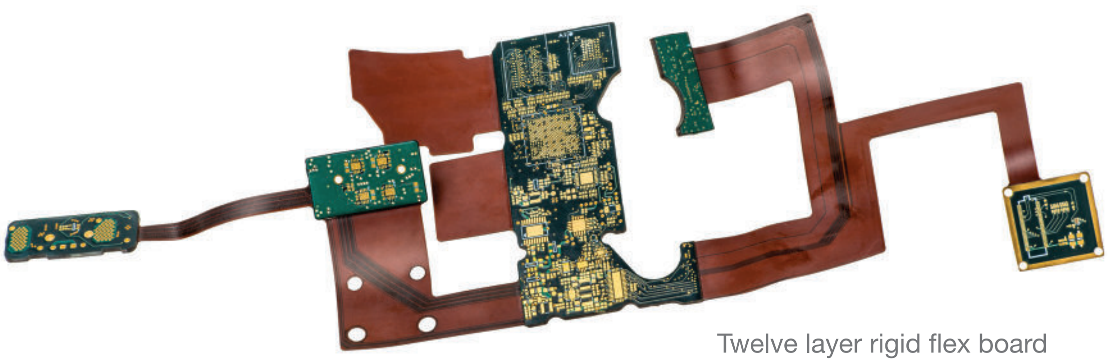
Nominal Thickness	Glass Style	RC (wt%)	Dk					Df				
Inch			1MHz	10MHz	100MHz	500MHz	1GHz	1MHz	10MHz	100MHz	500MHz	1GHz
0.002	#106	65-71	3.85	3.75	3.7	3.64	3.61	0.0087	0.0102	0.0117	0.0121	0.0123
0.0025	#1080	56-62	4.12	4.02	3.97	3.91	3.88	0.0083	0.0098	0.0113	0.0117	0.0119

8.2.2 isola FR 406 (IPC 4101/24P)

isola’s FR406 prepreg also comes in 106 and 1080 glass fabrics, which press out at .002” and .0025” respectively.

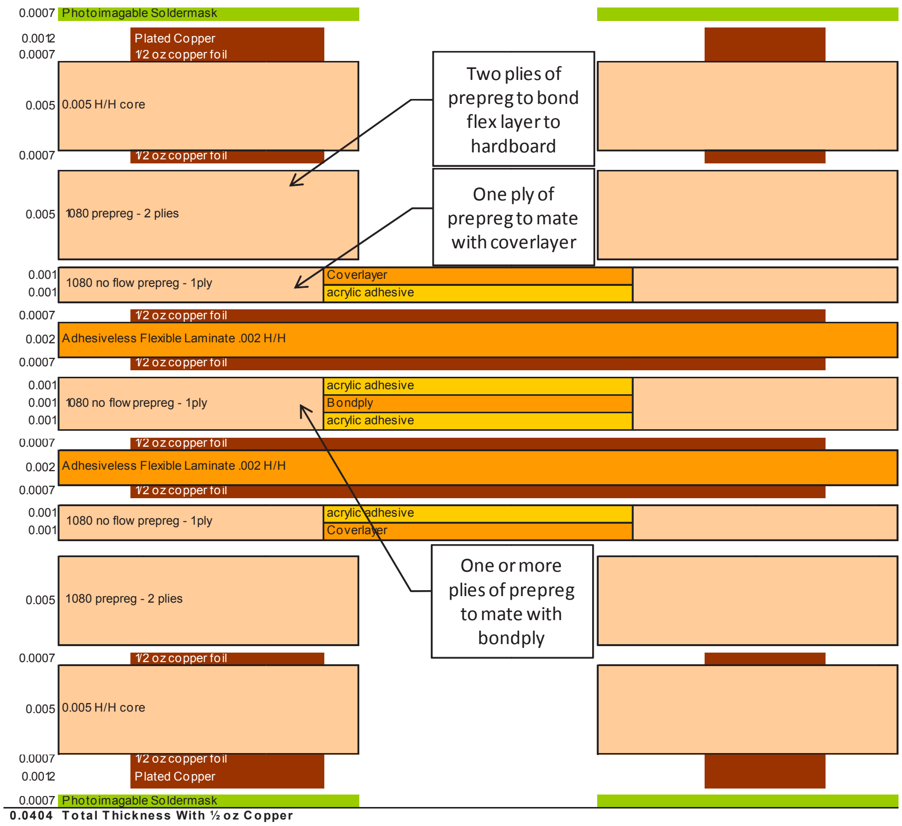
8.2.3 Design Considerations When Using Prepreg:

There are some considerations on specifying prepreg constructions. We need a minimum of two sheets of prepreg between each laminate opening. Designing rigid flex PCB’s with fewer than two sheets can limit how much resin is in the opening, resulting in air entrapment. Air entrapment will cause high resistance shorts at electrical test, and ultimately cause reliability issues after assembly and in the end application.



Twelve layer rigid flex board

Three sheets of prepreg should be used in the stackup wherever the flex is laminated to the rigid board materials for the following reasons: One sheet of prepreg mates with the coverlayer or bondply at the rigid flex transition area. The other two bond the flex layers to the hardboard layers (see illustration below). Attempting to accomplish bonding in these areas with just two plies of prepreg, can create air entrapment, because the prepreg does not have enough flow to encapsulate your circuitry. Air inside of a circuit board will cause high resistance shorts at electrical test, and ultimately cause reliability issues after assembly and in the end application. This is especially true, when the outer core materials are thicker than .012” where the core limits the lamination press’ ability to conform the prepreg to the internal circuitry.



Specifying too many plies of prepreg, can also cause issues with your design. Prepregs are designed not to flow, but they still flow a bit, providing encapsulation of your circuit features. FR4 prepregs tend to flow a bit more than polyimide prepregs. In either case, if too many plies are used, the excess resin will flow out onto the flexible area of the PCB, harden, and render it inflexible. A good general rule is one sheet of prepreg to meet up with the coverlayer and/or bondply, and then two more sheets to bond the flexible layers to the next layer in your design. More or less than this will create issues with your design and its reliability. See All Flex Solutions Valu Build Program for very low cost, very stable, rigid flex material layups.

Some other considerations on bonding your design with prepreg:
When you are using a pouch rigid flex construction (see section 7.4) the rigid to flex transition area is defined by how much or how little the resin flows out onto the flexible area of the PCB. Your rigid to flex transition may not end up exactly where you wanted it. The IPC 6013 gives a ± 0.059” no inspection zone on the rigid to flex transition area for this, and other reasons.

This is also true for rigid flex constructions using copper foil, or a cap piece of laminate (copper on only one side of the laminate, and no copper features on the other side) on the outerlayers of your board. These materials do not have any copper features, which work to restrict resin flow. Without those features, you may also encounter excessive resin flow out onto the flexible areas of your design. Consult your fabricator for their advice, if you are unsure.

Because of the nature of prepregs, the flexible areas of your design, between rigid boards, should be at least .100” long, and preferably more. Designs with flex sections less than .100” long are hard to get to yield, due to the variation in flow of prepreg.

Another concern is flex arms that are adjacent to rigid sections of your design. Care must be taken to allow enough space between the flexible area of the board and the rigid section, see illustration below. Even though these areas are routed away, if the prepreg flows beyond the rout path, your part will have hardened resin on the flex arm, limiting flexibility. Best practice is to leave at least .062” between these features, and preferably more.

Note, all layups shown are for half and one ounce copper weighted materials. Constructions using copper weights greater than one ounce, require coverlayers and bondplies with thicker adhesive, and usually more and thinner prepreg plies, such as 106 – which contains more resin than 1080, to improve encapsulation of the circuitry. Consult your fabricator for recommended layups for copper weights greater than one ounce.

It is important that the thickness of the prepreg that mates with either bondply or coverlayer, needs to be a similar thickness as much as possible. For example, if you have chosen an LF/FR0222 which is .006” thick, the prepreg in the rigid section needs to be close to that thickness. In this case two plies of 1080 prepreg at .005” thickness or three plies of 106 at .006” would be sufficient.

8.3 FLEXIBLE COPPER CLAD LAMINATE MATERIALS (IPC 4204/11):

Flexible Laminates (IPC 4204/11) with rolled annealed copper foil are recommended for rigid flex constructions.

The IPC recommends (IPC 2223 4.2.4.1) the use of adhesiveless flexible laminates in rigid flex constructions to reduce the deleterious effects of acrylic adhesive within the rigid portion of the board. Acrylic adhesive has high z axis CTE rates, putting unnecessary stress on the vias and can cause via failure, resulting in random open circuits. It also has inconsistent mechanical properties in the subsequent drilling, plasma and plating processes.

For the same reason, high reliability rigid flex designs almost always use a technique called cut back coverlayer in their construction (see 7.3 Cutback Coverlayer and Bondply and IPC 2223 5.2.2.2). The acrylic adhesive in the coverlayer has a much higher z axis CTE rate than the other materials, which if included in the rigid part of the PWB, will exert z axis force on the vias during thermal excursion.

To address this issue, rigid flex fabricators allow the coverlayers and bondplies to go .050” into the hardboards and no further. The coverlayer and bondplies are then mated with a sheet, or sheets, of prepreg of roughly the same thickness. This area then becomes a “keep out” area for vias, to prevent drilling through the coverlayer in the hardboard sections (see IPC 2223 5.2.2.3).

Fourteen layer rigid flex board

Following is a list of common adhesiveless flex laminate constructions. Three items of note: flexible laminates should always be specified with rolled annealed copper foil for ultimate flex endurance life in high reliability applications.

It is often necessary with controlled impedance designs to use thicker dielectrics (e.g. .003” or .004”) to get the impedance calculations to work. And, pricing on adhesiveless flexible laminate is not linear like core laminates. The cost for the material follows an exponential curve with each .001” in dielectric thickness.

Unlike hardboard materials it is very unusual to specify adhesiveless flexible laminates with unbalanced copper constructions e.g. half ounce/one ounce laminates. These constructions are very rare, usually a special order, and are expensive.

Please ask for a Polar library of common Dk values for flexible laminates, or use a base value of 2.9 and not the 3.5 on the manufacturer’s product data sheet.

Materials marked in blue are the most common or popular.

8.3.1 DuPont AP Adhesiveless flexible copper clad laminates

Product	Kapton® Thickness	Copper Thickness
AP8515	.001"	1/2 ounce (.0007")
AP9111	.001"	1 ounce (.0014")
AP9212	.001"	2 ounce (.0028")
AP8525	.002"	1/2 ounce (.0007")
AP9121	.002"	1 ounce (.0014")
AP9222	.002"	2 ounce (.0028")
AP8535	.003"	1/2 ounce (.0007")
AP9131	.003"	1 ounce (.0014")
AP9232	.003"	2 ounce (.0028")
AP8545	.004"	1/2 ounce (.0007")
AP9141	.004"	1 ounce (.0014")
AP9242	.004"	2 ounce (.0028")
AP8555	.005"	1/2 ounce (.0007")
AP9151	.005"	1 ounce (.0014")
AP9252	.005"	2 ounce (.0028")
AP8565	.006"	1/2 ounce (.0007")
AP9161	.006"	1 ounce (.0014")
AP9262	.006"	2 ounce (.0028")

8.3.2 Panasonic RF775 Adhesiveless flexible copper clad laminates

Product	Kapton® Thickness	Copper Thickness
RF775 H/1/H	.001"	1/2 ounce (.0007")
RF775 1/1/1	.001"	1 ounce (.0014")
RF775 2/1/2	.001"	2 ounce (.0028")
RF775 H/2/H	.002"	1/2 ounce (.0007")
RF775 1/2/1	.002"	1 ounce (.0014")
RF775 2/2/2	.002"	2 ounce (.0028")
RF775 H/3/H	.003"	1/2 ounce (.0007")
RF775 1/3/1	.003"	1 ounce (.0014")
RF775 2/3/2	.003"	2 ounce (.0028")
RF775 H/4/H	.004"	1/2 ounce (.0007")
RF775 1/4/1	.004"	1 ounce (.0014")
RF775 2/4/2	.004"	2 ounce (.0028")
RF775 H/5/H	.005"	1/2 ounce (.0007")
RF775 1/5/1	.005"	1 ounce (.0014")
RF775 2/5/2	.005"	2 ounce (.0028")
RF775 H/6/H	.006"	1/2 ounce (.0007")
RF775 1/6/1	.006"	1 ounce (.0014")
RF775 2/6/2	.006"	2 ounce (.0028")

8.4 COVERLAY, BONDPLY AND CAST ADHESIVE MATERIALS - (IPC 4203/1):

Coverlayer and bondply are most often .001” Kapton® with acrylic adhesive of varying thickness (IPC 4203/1). Thicker Kaptons ® are available, but aren’t used very often. Typically you should use .001” of acrylic adhesive for every .001” of copper fill – e.g. half ounce copper needs .001” acrylic, one ounce .002” of acrylic, etc.

You can also get LF or FR bondply without Kapton® supporting film. It is called cast adhesive, and it is just pure acrylic adhesive. Both are shown in the chart on the next page.

Materials marked in blue are the most common or popular.

Note, if your product requires UL approval, or flame resistance, specify DuPont FR coverlayer, bondply and/or cast adhesives. The material selection is the same as LF materials shown on the next page, just change the “LF” to “FR”.

DuPont LF Coverlayer

Product	Kapton® Thickness	Adhesive Thickness
LF0110/FR0110	.001"	.001"
LF0120/FR0120	.002"	.001"
LF0130/FR0130	.003"	.001"
LF0150/FR0150	.005"	.001"
LF0210/FR0210	.001"	.002"
LF0220/FR0220	.002"	.002"
LF0230/FR0230	.003"	.002"
LF0250/FR0250	.005"	.002"
LF0310/FR0310	.001"	.003"

DuPont LF Cast Adhesives and Bondplies

Product	Kapton® Thickness	Adhesive Thickness
LF0100/FR0100	N/A	.001"
LF0200/FR0200	N/A	.002"
LF0300/FR0300	N/A	.003"
LF0111/FR0111	.001"	.001"/.001"
LF0121/FR0121	.002"	.001"/.001"
LF0212/FR0212	.001"	.002"/.002"
LF0222/FR0222	.002"	.002"/.002"

Note, on multilayered flex designs, it is usually best to put the circuit layers internally and the ground/power planes on the external layers of the flexible areas of the boards. This reduces the stress on the circuits on external layers, and on the outside of the radius of bend, which is typically the most stressed layer of the construction.

Cross hatched ground/power/shielding planes can improve flexibility in flex designs, but need to be accommodated in impedance modeling for controlled impedance designs.

8.5 HIGH TEMPERATURE COVERLAYER AND BONDPLY:

DuPont has developed a flexible high temperature material called Pyralux® HT offering very high temperature performance > 225° C, that can be used as either a coverlayer or a bondply for multilayered flexible circuits or rigid flex constructions. Pyralux® HT is available in .0015”, .002”, and .003” thicknesses.

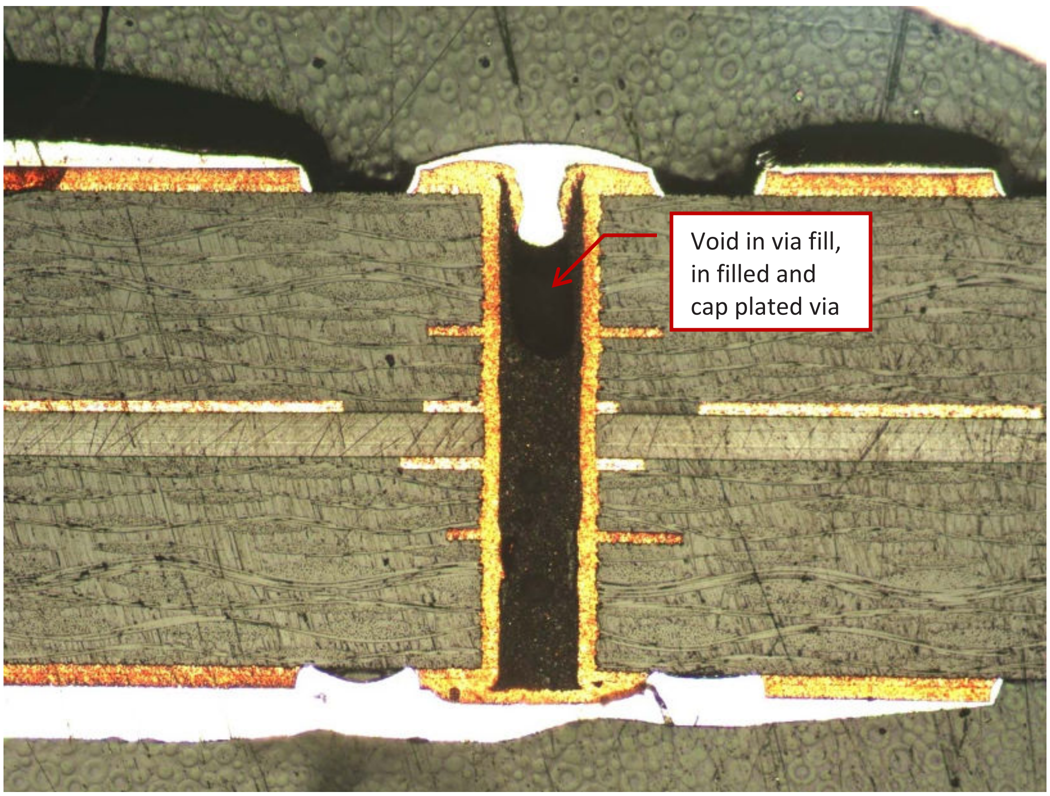
9. ADDITIONAL RIGID FLEX PROCESSING ISSUES

9.1 VIA FILLED THROUGH HOLES ON RIGID FLEX PWB'S:

Specifying filled and cap plated vias on rigid flex boards should be avoided if possible. Filled and cap plated vias are common in rigid board designs, but are difficult to do with rigid flex boards due to the surface topography that is inherent in rigid flex outer layer surfaces.

Boards with filled through vias will typically have three weeks additional lead time and a yield loss of 20% to 30% more than boards without via fill.

Filled and cap plated vias are common in hardboard manufacturing, as the typical hardboard has a very smooth and planar surface coming out of lamination. After filling the vias with a conductive or non-conductive via fill material, it can easily be removed from the surface using an aggressive sanding machine, such as a Timesaver.



In rigid flex manufacturing, the fabricators use prepreg, which is designed to limit flow out onto the flexible areas of the boards. Because the prepreg is designed not to flow, the fabricator must use conformable products in lamination that are designed to drive the multilayer package into intimate contact. The result is a highly topographical surface, typically ± .006" across the panel. Also, there are open cavities in the flexible areas of the board, which fill with the epoxy based via fill materials.

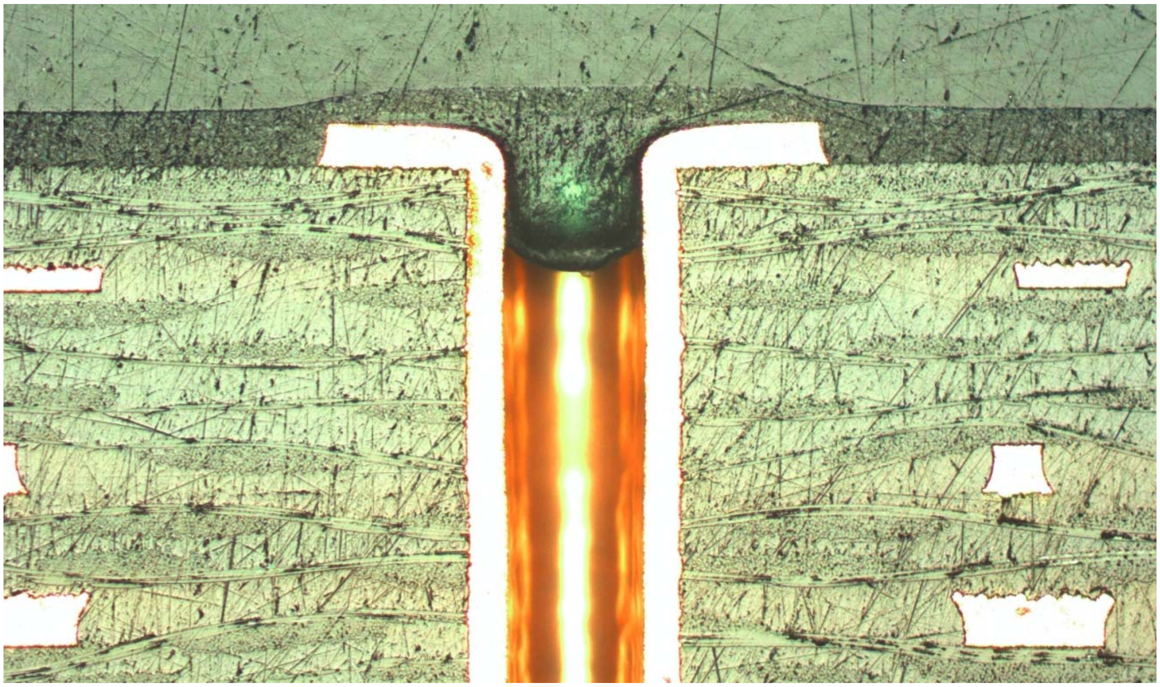
If an aggressive machine is used to remove the cured epoxy material, it will remove the base copper from the raised features, destroying the parts. Much of the process for removing the epoxy material is done by hand. Every effort should be made to design boards that do not require filled through vias.

Additionally, the IPC has a “wrap” requirement for Class III boards, requiring the fabricator to maintain a minimum of .0005” of copper plating on the via land, from the first plating process. This encourages the fabricator to plate extra copper on the outer layers during the first plating of the via, making sub .004/.004” circuit construction on the outer layers very difficult and sometimes impossible.

Filled and cap plated vias cannot be accomplished on multilayered flex boards, nor can it be done on the flexible areas of rigid flex designs. When used, it is limited to the rigid portions of rigid flex designs.

An acceptable alternative to specifying filled vias, is to design with dog bone structures, and have the fabricator use a full body ENIG/ENEPIG process, and then use dry film soldermask to cap the vias adjacent to the surface mount pad. An example is shown below.

All filled vias, that are not lands for component assembly (e.g. via in pad) should be covered with soldermask.



9.1.1 Via Fill Recommendations:

Approximately .005 - .006” finished filled via, drilled at .0098” in .062” board.

Approximately .007 - .008” finished filled via, drilled at .0118” in .080” board.

9.2 LASER VIAS:

Laser vias can be drilled down to .004”, but no greater than a 1:1 ratio of drill depth to drill diameter. Laser vias do add cost and turn time. If laser vias are used, they should be used on both sides of the PWB, and not just on one of the outerlayers.

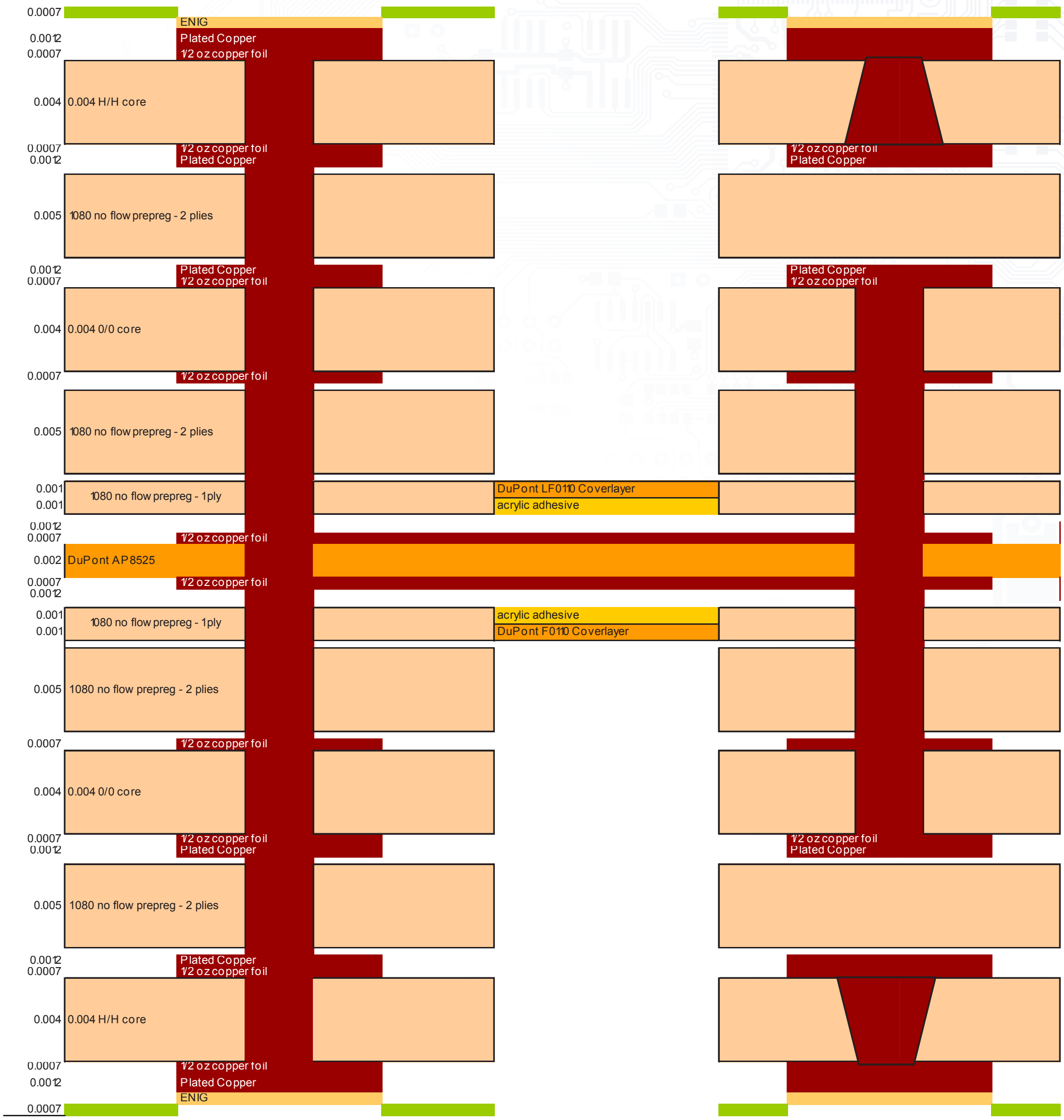
Inverted laser vias can be used on specific layer constructions and are preferred for cost and yield over conventional laser vias. With inverted laser vias, the fabricator drills from layer 2 to layer 1, rather than the conventional method of drilling from layer 1 to 2.

Layers 1 and 2 then create a subassembly, which is laminated onto the rest of the package.

The advantage to the fabricator is a flatter, less topographic surface for laser drilling (rigid flex has a great deal of topography typically, and lasers have a focal length similar to cameras, sometimes making laser vias difficult to yield predictably).

The advantage in assembly is that the external pad is completely flat without any plated dimple.

A diagram showing the typical inverted laser via build is to the right. Inverted vias can only be used on 6, 10, 14 and 18 layer PWB’s.



9.3 RIGID FLEX BEADING/STRAIN RELIEF:

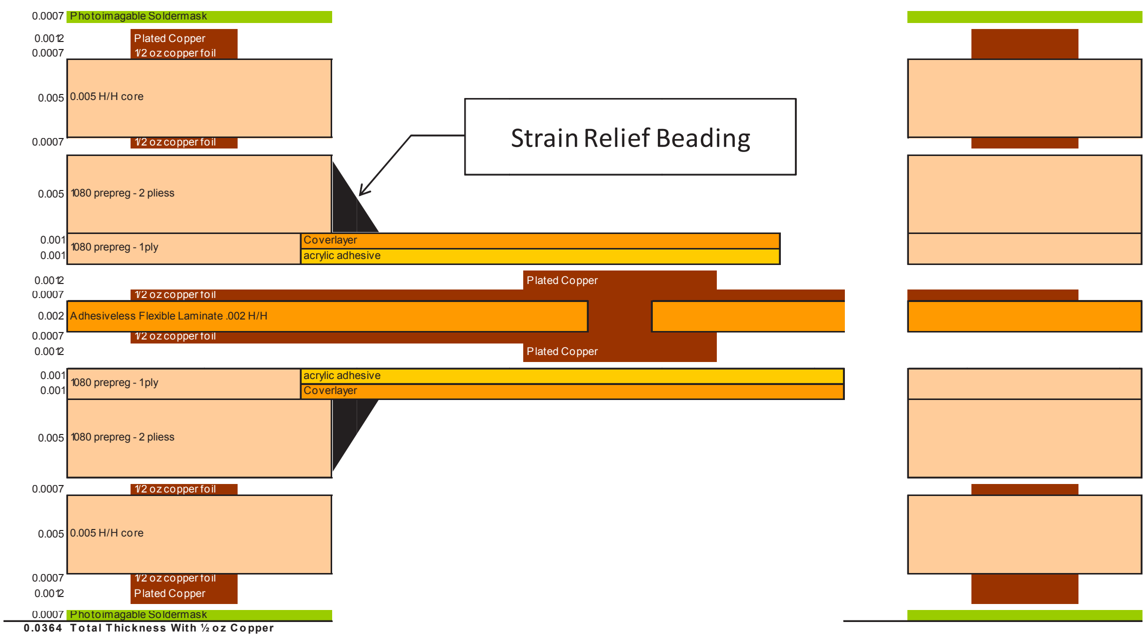
Rigid flex boards sometimes need an application of a soft compound, typically an epoxy system, along the rigid to flex interface(s). This compound is designed to reduce the amount of stress on the circuits within the flexible layers at this juncture, due to handling, vibration, etc. It is commonly used on boards fabricated with pouch constructions (see 7.4) due to the rough glass edge, in high vibration or dynamic flex applications, high reliability applications, or in cases where assembly and/or handling or end application may put undo force on the circuits in the flexible section of the board.

Applying strain relief beading is a hand operation, and usually adds a few days onto your expected lead time to perform the beading, and to allow the material to air dry and cure. Baking, often causes the beading material to run out onto the flex.

There are some important considerations for beading. The hand drawn beads are typically .030” to .060” wide and the material has a very low viscosity. It can be difficult to control the size of the bead. Because of this, the IPC describes the rigid to flex transition area - .059” on either side of that line – as a no inspection area. Calling out minimum and/or maximum size for your beads can be difficult or impossible to attain.

Also, if your rigid boards are close to each other – e.g. the flexible section of the arms is very short, the beading will further reduce that distance and can interfere with your bend radius. For example, if the flexible area on your flex arm is .100” long, and the bead on each side is .030” wide, the remaining flexible portion of your board is .040” and may not bend to meet your requirements.

If your design has a very short distance between the outerlayers of the PCB and the top of the coverlayer, accomplishing a bead can be very difficult. The beading material has a very thin viscosity, and needs an adequate lip to adhere to. This is done entirely by surface tension. If the lip is too thin – e.g. less than .020” the beading material will tend to run up on top of the outerlayers of your design.



9.3.1 Henkel Loctite Ablestik 45 black with Catalyst 15 black, or Henkel Loctite *Ablestik 45 clear with Catalyst 15 clear, are recommended*
(formerly Eccobond 45 and 15)

Henkel Loctite Ablestik 45 is typically mixed in a 1:1 ratio of resin to catalyst/hardener to produce a semi-rigid deposit. However, it can be mixed in other ratios to produce more or less flexible deposits. Consult Henkel Loctite Ablestik 45 product data sheet for specifics.

9.3.2 Other beading materials
Armstrong C7
3M Scotch-Weld 2216

9.4 ISOLATED PLATING ON FLEXIBLE AREAS OF RIGID FLEX PWB'S:

This note does not apply to typical rigid flex boards, where every flex arm terminates in a hardboard. In general, those boards should not have electroplated copper on the flexible areas of the boards.

However, boards that have one or more flex arms that terminate in just flex (see 7.4 Types of Rigid Flex), could possibly have the features on the flex arms plated with additional copper. In these types of boards, the flexible sections of the board are built to completion, often plating through holes in the flexible section of the board.

When plating the through holes, the fabricator, unless instructed otherwise, will plate up all of the other features on the flexible areas of the board as well (i.e. circuits, ground planes, etc.).

Plated copper is not always ideal in flexible circuit regions of rigid flex PWB's. The additional plating will make the flexible area of the board less flexible.

In these cases, the fabricator needs to be instructed to use what is called isolated plating, or button plating on the flexible sections of the board. Isolated plating limits the electroplated copper deposit to just the through holes and a .002" ring around the pads on the through holes, and does not put any copper on the circuits or other features on the flexible areas of the board.

This technique keeps the flexible areas of the board, as flexible as possible. Isolated plating does increase lead time by a few days, increases cost and reduces yield.

9.5 SOLDERMASK AND PHOTOIMAGEABLE COVERLAYER:

Minimum soldermask dam = .005". Dams less than .005" will typically be removed.

Type: Taiyo PSR-4000 LDI
Colors: green, red and black

If photoimageable soldermask is used to cover a via or vias, on one side of the board, those via or vias need to be covered on the other side as well. Covering vias on one side, and not the other side of the PWB, will cause skip plating at ENIG/ENEPIG, and will reduce yields considerably – 50% or more – and the fabricator cannot check design files for these anomalies in the soldermask Gerber files.

Photoimageable coverlayer (PIC): Minimum photoimageable coverlayer dam = .004”
Type: Taiyo PSR-4000 LDI
Colors: amber and green

Photoimageable coverlayer is sometimes used on flex arms to improve coverlayer imaging particularly around fine features. Conventional coverlayer rapidly reaches the end of its ability to provide solder isolation on higher density flex and rigid flex designs. As this limit is reached, the designer can elect to use photoimageable coverlayer near and around the higher density areas, and then put a layer of conventional coverlayer on top.

Photoimageable coverlayer is not as robust as conventional coverlayer and cannot provide the flex endurance life of conventional coverlayer, but it is a good method of solder isolation in higher density applications. Photoimageable coverlayer is also not as robust as traditional soldermask or coverlayers in the ENIG/ENEPIG plating process and is prone to attack from the solutions used in plating.

Photoimageable coverlayer is usually used only on rigid flex boards that have a flex arm that does not terminate in a rigid board (see 7.4 Types of Rigid Flex). The additional processing steps will increase the cost of the board. Also, the photoimageable coverlayer cannot hold up in the plasma process that is used to condition hole walls prior to plating. For that reason, when photoimageable coverlayer is specified on the flex arm of a rigid flex design, that/those flex arm(s) must be pouched (see 7.4) to protect the photoimageable coverlayer during the plasma process, increasing cost and lead time.

Printed Circuits recently invested in an exciting new technology for soldermask and nomenclature application called direct printing.

We purchased an n.jet soldermask ink jet printer from Notion Systems Gmbh. It uses an additive technology to print soldermask and nomenclature directly onto the printed circuit board. This technology eliminates all of the coating, baking, imaging, developing and curing steps inherent in conventional photoimagable soldermasks and nomenclature – reducing what would typically be seven process steps, for each side of the panel, down to three. The soldermask and nomenclature inks are printed onto the production panel, followed by an inline UV cure, and then placed on a conveyorized tunnel oven for a final cure.

The printer applies soldermask in the areas indicated in your Gerber files and can scale the artwork according to panel movement allowing very precise registration. It uses a special soldermask developed by Taiyo that is very similar to conventional photoimageable masks, just without the photoinitiators. The technology will give us the capability to print mask dams down to .004, or less, and help us with mask colors that were always challenging to image with photoimageable masks, such as blue and black.

With this new technology we expect:

- Higher and accelerated throughput
- No soldermask in holes
- Excellent repeatability
- Adjustable soldermask thickness
- Greater mask adhesion
- No soldermask on pads
- Smaller and more reliable soldermask dams

9.6 FINAL FINISHES:

The final finish applied to a PWB is determined by the designer and is based on component assembly requirements and the end-use of the product.

Both ENIG and ENEPIG offer improved planarity over HASL, making them better suited for surface mount assembly, and both are defined by IPC standards (ENIG is IPC 4552, ENEPIG is IPC 4556).

Note: Some sources have attributed the occurrence of a chemical reaction known as “Black Pad” to the use of the ENIG process. “Black Pad” is a result of poor implementation and not the process itself.

Dual finishes, where a PWB has a finish on one part of the board, and an alternative finish in other location(s) should be avoided if at all possible. Current photoresists that are available have a hard time holding up during the electroplating nickel gold or ENIG/ENEPIG processing, and often break down at the foot of the photoresist, creating shorts. PWB’s with dual finishes typically have 20% to 30% yield reduction compared to single finish boards. Additionally, the exposed photo plating resists comtaminate the nickel and gold plating baths.

9.6.1 Hot air solder level (HASL) - HASL finishes are becoming less common, but can be provided if desired. HASL has limitations on thickness control, deposit in the vias, consistent planarity, etc. making ENIG and ENEPIG finishes preferable for manufacturing and overall yield.

9.6.2 Electroless nickel immersion gold (ENIG) - ENIG is a preferred finish. Nickel is typically specified as a minimum thickness of 118 microinches and gold a minimum of 1.97 microinches. See IPC 4552 or IPC 6013 Table 3.2 for recommended thicknesses.

9.6.3 Electroless nickel electroless palladium, immersion gold (ENEPIG) - ENEPIG is the preferred finish for high reliability applications, or where mixed technology - soft wire bondable gold and surface mount assembly are used on the same PWB. ENEPIG has excellent soft wire gold bonding characteristics and pull strengths, often better than soft electroplated gold, and significantly improves yields compared to dual finishes (see 9.6 Final Finishes:). Nickel is typically specified as a thickness of 118 to 236 microinches, palladium 2 to 12 microinches, and gold a minimum of 1.2 microinches. See IPC 4556 or IPC 6013 Table 3.2 for recommended thicknesses.

9.6.4 Electroplated nickel hard gold - nickel and hard gold can be electroplated onto copper features and are typically used whenever a final finish is desired that is resistant to mechanical abrasion - typically PWB edge fingers, mechanical switches, etc. - areas that are not soldered. The nickel plating is typically specified as a minimum of 100 microinches and the gold is typically specified as a minimum of 50 microinches thick.

9.6.5 Other - Immersion Silver, Tin and OSP coatings are available, but are not recommended.

9.7 NOMENCLATURE:

White is recommended - 0.030" high with .003" draw minimum.
We use an Orbotech direct legend ink printing machine to print Taiyo IJR-4000 FW 200 ink.

Often designers put nomenclature on the flex arms and the rigid sections, or even flex arms and stiffener sections assuming that these features are all printed by the fabricator at the same time. Due to the distance between the surfaces of the rigid board and the coverlayer and/or stiffeners, each layer/feature must be printed independently. If we print all at the same time, one or more sets of features will not print clearly. Printing your nomenclature four or five different times, adds processing steps, which adds to your manufacturing lead time as well as cost. If possible, limit nomenclature to just the rigid sections of your design, or just the flexible sections.

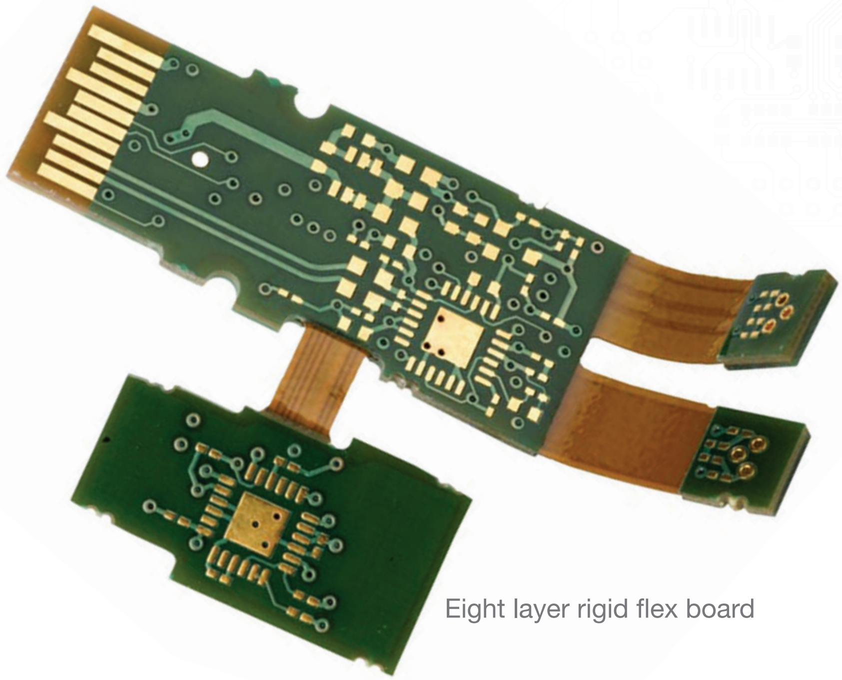
9.8 FRAMES AND FIDUCIALS:

Rigid flex PWB's are usually procured with a frame or array around the external dimensions of the part or parts. The frames are used to provide stability for the part(s) during assembly and are removed afterwards. Frames, or individual rails, are typically .400" or wider. Fiducials are typically placed within this frame area, typically three round copper features .125" in diameter, with .250" in soldermask clearance. With very high density assemblies, fiducials can also be located on each rigid section, or within the part to aid machine vision on pick and place machines for BGA assembly, etc.

9.9 OTHER DESIGN CONSIDERATIONS:

Some considerations for your rigid flex PCB print:
Rigid flex designs are very similar to rigid PCB designs in that the layout engineer provides artwork, or gerber layers for each of the circuit layers, and a manufacturing print detailing their requirements. Where they differ, is that rigid flex designs need a very accurate material layup, to be sure you are getting a PCB that will perform mechanically and electrically as desired. Rigid flex designs also need accurate dimensioning, particularly where the rigid to flex transition areas are located, which might not always be apparent in the gerber files. And, in general rigid flex boards typically have more dimensions than a rigid board, because they are usually used in three dimensional applications.

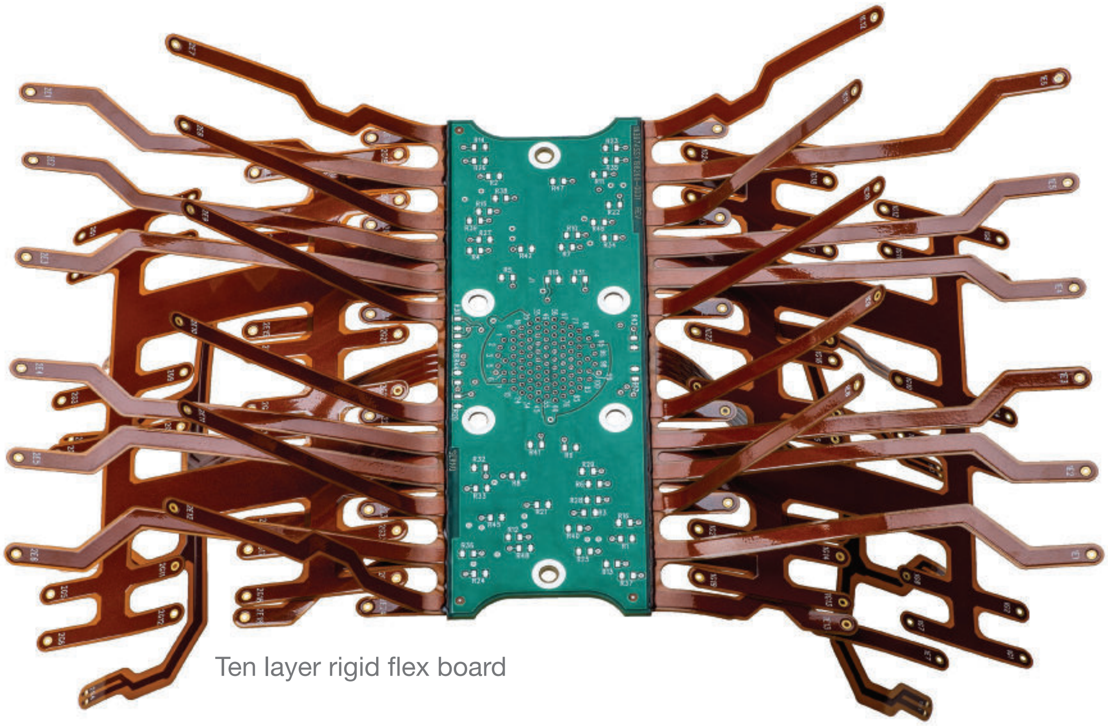
A common mistake is features that are dimensioned from one Rigid area to another Rigid area – across the flexible arm. It is perfectly acceptable to have features within a rigid area defined with tolerances normally seen on a rigid PCB. It is not a good idea to have tight tolerances from Rigid section to Rigid section. Reference dimensions in these instances are acceptable. Whenever possible, include an iso view, showing how the circuit board will be installed, on your drawing. This helps verify the design is robust and designed well for its intended use.



Eight layer rigid flex board

Rigid flex manufacturers use special prepregs. It is different than what is typically used in rigid board constructions as it is purposefully designed for very little resin flow, to keep the resin from flowing out onto the flexible areas of the board. Because of this feature, care should be taken to avoid high and low pressure areas within the board. High pressure areas are caused by stacking of innerlayer features. High layer count pad stacking can cause localized high pressure areas adjacent to very low pressure areas, which can make it difficult for the resin to fill adequately, reducing manufacturing yields. Putting multiple ground and power planes on one side of the board, and all signal layers on the other, can also cause high and low pressure areas, which will create air entrapment and lower yields. A rigid flex design should be crafted with as much symmetry as possible to improve manufacturing yields.

The flexible sections on your rigid flex design should not be less than .250” from one rigid board to the adjoining one. It is possible to produce designs even down to .100” between hard boards, but it can be difficult to control the resin flow on prepreg (particularly FR4 prepreg), so your .100” dimension can be reduced to .060” preventing the board from meeting your mechanical requirements. Also, this area of the PCB per IPC is considered a “no inspect zone” for .059” on either side of the rigid flex transition line. The part could be manufactured and meet the IPC requirements, but still not be mechanically suitable for your application.



Ten layer rigid flex board

Putting ground planes along with high density circuitry on outerlayers of rigid flex PCB’s can cause issues. Rigid flex PCB’s, unlike hardboards, have a great deal of surface topography, sometimes as much as ± .006”. This can make imaging them faithfully more difficult than rigid boards due the the focal length inconsistency in imaging. Parts with ground planes and higher density circuitry can be built successfully if at least .010” – or more - spacing is kept between the ground plane and any adjacent feature.

9.10 ACCOMMODATING MANUFACTURING PROCESSING:

Fabricators have to alter your Gerber layers in order to meet the requirements for the final product. Circuits are typically oversized, or enlarged, so that when the circuits are etched, they end up at the nominal size in the design.

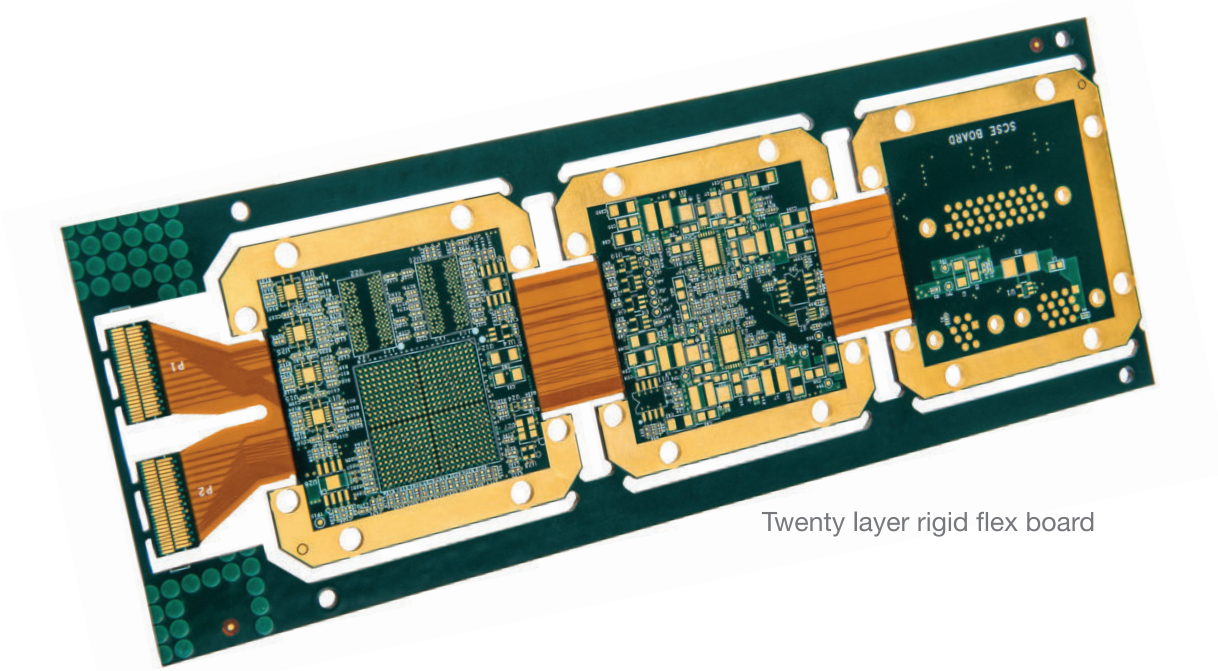
Half once copper features – both circuits, ground planes and pads – are typically increased by .001”. One ounce copper features are typically increased by .002”. And, two ounce copper features are typically increased by .003”.

Drilled vias, are typically drilled .004” to .006” larger than the nominal finished via size, to accommodate the plated copper that goes into the vias.

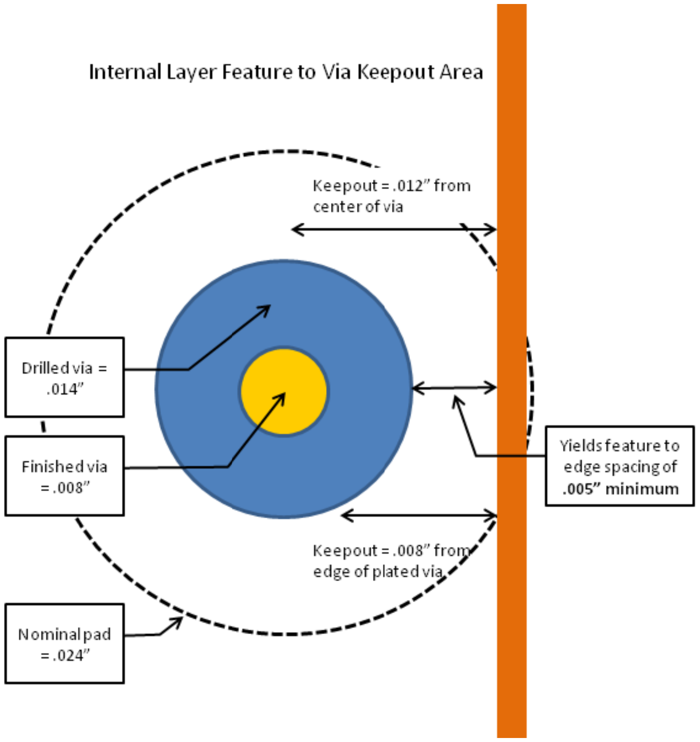
Because of this, there is a keepout area on all internal layers for copper features, whether circuits or ground planes that needs to be observed. Fabricators drill the vias larger than the specified finished via size, in order to accommodate the electroplated copper in the hole. The larger sized drilled via can intersect with copper features if they are placed too close to the via.

For example, if you have a finished hole size of .008”, the fabricator will drill those vias at .014” which will reduce spacing to adjacent features. The keepout area is equivalent to what you would have if non-functional pads are added to the design – or .005” per side of the via, plus an additional spacing of at least .003”.

In the example above, where the .008” finished via is drilled at .014”, the keepout area would be .008” away from the edge of the plated via, or .012” from the center of the via, yielding a separation of .005” minimum from the edge of the drilled via to the nearest feature. See the representative diagram on the following page. Note, the dotted line represents a hypothetical pad (internal or external) and the features are being routed within the pad boundaries.



Twenty layer rigid flex board



There is a keepout area on all internal layers for copper features, whether circuits or ground planes, that needs to be observed

10.0 PROCESS CAPABILITIES AND DESIGN RULES:

10.1 PANEL SIZES:

18 x 24" yielding ~ 16" x 22" usable space less manufacturing and impedance coupons, etc.

12 x 18" yielding ~ 10" x 16" usable space less manufacturing and impedance coupons, etc.

10.2 LAYER COUNT:

Rigid section – 3 to 26 recommended

Flex section – 1 to 4 recommended, more can be accommadated using multiple doublets

10.3 THICKNESSES:

Thinnest board: .006" recommended measured over soldermask/coverlayer

Thickest board: .125" recommended measured over soldermask/coverlayer

Note, thickness callouts on the prints should be very clear, and preferably be measured from soldermask on layer 1 to soldermask on layer X. Material layups often are not clear whether the measurements are over laminate, base copper, plated copper, or soldermask.

10.4 THICKNESS TOLERANCE:

Rigid: ± 10%

Flex: ± 10%

10.5 LINE WIDTH AND SPACING:

(1/2 oz copper .0007” thick)
Minimum trace width inner layers = .003”
Minimum trace separation inner layers = .003”

(1 oz copper .0014” thick)
Minimum trace width inner layers = .005”
Minimum trace separation inner layers = .005”

(2 oz copper .0028” thick)
Minimum trace width inner layers = .008”
Minimum trace separation inner layers = .008”

(1/2 oz copper with plating for total thickness of .0021”)
Minimum trace width outer layers = .004”
Minimum trace separation outer layers = .004”

(1 oz copper with plating for total thickness of .0028”)
Minimum trace width outer layers = .005”
Minimum trace separation outer layers = .005”

Note, care should be taken to have either circuits, or ground planes on outerlayers, but not both at the same time. It should be one or the other and not mixed. If it cannot be avoided, traces should be greater or equal to 0.005” and spacing no less than 0.004”, with distances between copper pours and circuits and pads greater than at least .010”.

10.6 VIAS:

Conventional – down to .0079” drilled

Conventional aspect ratio – no more than 10:1 drill depth to drill diameter

Laser – down to .004” laser

Laser aspect ratio – no more than 1:1 drill depth to drill diameter

Pad Sizes – drilled via + .010” minimum to meet Class III requirements + .008” to meet Class II requirements

Copper plating - .001” minimum in the via

Plated through hole tolerance - ± .003” recommended

Filled Vias:
Conductive or non-conductive via fill is available.

Minimum filled vias:
Approximately .005 - .006” finished filled via, drilled at .0098” in .062” board

Approximately .007 - .008” finished filled via, drilled at .0118” in .080” board

10.6 FEATURE DIMENSIONS AND TOLERANCES:

Mechanical/routed dimension tolerance - ± .005” minimum
Copper feature to edge distance - .015” minimum
Copper feature to rigid/flex transition area - .025” minimum
Plated through hole to rigid/flex transition area (edge of drilled hole) – .050” minimum

Note, per IPC no dimensions on the print should be across a flexible section(s) of the PWB.

Transition area for controlled impedance lines when moving from rigid to flex should occur .050” from the rigid-flex interface on the rigid side.

10.7 ELECTRICAL TESTING:

Electrical testing typically is at 100 volts, isolation at 10 Meg ohms and 10 ohms continuity. Other testing can be accommodated, but needs to be specified on the print.

11.0 STANDARD PRINT NOTES:

Typical notes on a rigid flex print contain the following information:

1) This PWB is designed in accordance with IPC-2223.

2) Fabricate in accordance with IPC-6013, Class 2 or 3.

3) Material: Laminate and prepreg shall meet the requirements of IPC-4101 for the glass reinforced materials, IPC-4204 with rolled annealed copper foil for the flexible copper clad laminates and IPC-4203 for the flexible coverlayer, bondply and/or cast adhesive.

4) Final Finish to be (enter 60/40 Tin/Lead HASL, ENIG or ENEPIG as required).

5) Hole Requirements: Hole diameters specified are finished size unless otherwise stated. Diameter and Tolerance to be as defined in the Hole Data Table.

6) Etchback required or not with dimensions typically .0002” when required.

7) Trace widths to be as defined by the Master Pattern, or Gerber files.

8) Impedance requirements should be listed in a table - providing layer number, impedance type, value desired and trace geometries within the Gerber layers for those circuits. Alternatively, each requirement can be listed within the notes, similar to the following:

- Trace width x.xxx on Layer(s) x, x and x are 100 ohm differential controlled impedance
- Trace width x.xxx on Layer(s) x, x and x are 75 ohm characteristic controlled impedance (Edit or delete note as required).

9) Flex layers to be bonded together using item x.

10) Apply strain relief epoxy (item x) at rigid to flex transition(s) indicated. Epoxy shall not extend above the surface of the rigid sections, nor more than .060” away from the rigid to flex transition area.

11) Solder Mask: Apply over bare copper using Liquid Photo-Imageable (LPI) Epoxy per IPC-SM-840, Class H. Resizing for minimal pad to mask clearance is acceptable. Soldermask Dams less than 0.005” may be removed.

12) Silkscreen as defined by the Master Pattern using White non-conductive ink.

13) Electrical test in accordance with IPC-9252, compare using IPC-D-356 CAD data supplied.

12.0 QUICK TURN RIGID FLEX:

Lead time can vary, depending on the fabricator’s plant loading, and the technical aspects of the board. It is possible to get a straightforward rigid flex design in as little as a few weeks. The features listed below will increase your lead time by the amount indicated:

Materials – laminate: two to four weeks for non-standard materials (see 8.1)
Materials – prepreg: N/A, always on hand or available quickly
Materials – copper clad flex: two weeks for non-standard materials (see table in 8.3)
Materials – coverlayer: two weeks for non-stocked materials (see table in 8.4)
Buried vias: two to three weeks
Pouch constructions: two to three weeks
Stand alone flex arms (sublam): one to two weeks per flex arm
Filled through vias: three weeks
Isolated/button plating flex arm: one to two weeks per flex layer
Laser vias: one to two weeks
Coverlayer/PI coverlayer combo: one to two weeks per flex layer
Dual final finishes: one to three weeks depending on the finishes
Beading/strain relief: two days to a week depending on quantity of beads
Bookbinder rigid flex: generally twelve week lead time, plus custom tooling

all flex
SOLUTIONS

Flex Circuit Center of Excellence
1705 Cannon Lane
Northfield, MN 55057

Rigid Flex Center of Excellence
1200 W 96th St
Bloomington, MN 55431

Flex Heater & Assembly Center of Excellence
9401 James Ave S Suite #120
Bloomington, MN 55431

(507) 663-7162
www.allflexinc.com
info@allflexinc.com

AS9100 Certified
ISO 9001 Certified
ITAR Registered
MIL-PRF-31032/3 Qualified
MIL-PRF-31032/4 Qualified
RoHS Compliant
UL Recognized
IPC Member
JCP Certified