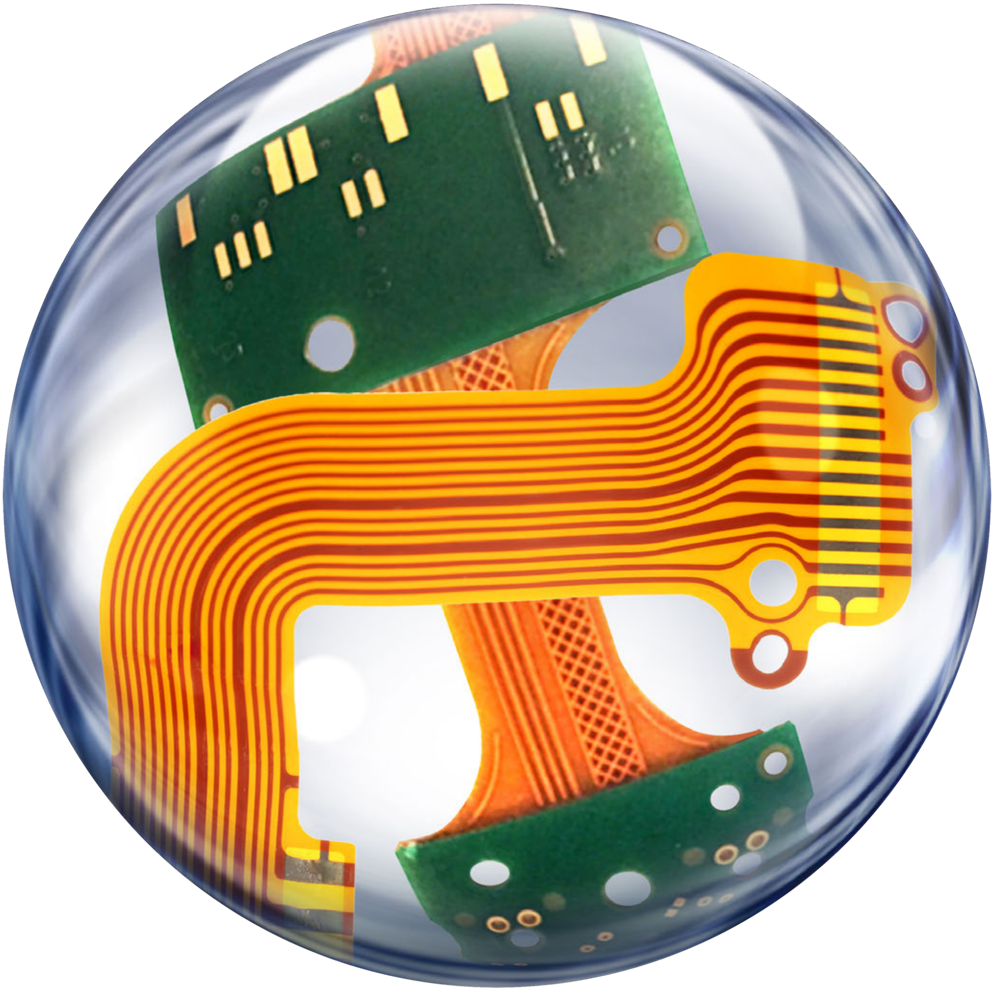
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™

Flex and Rigid-Flex Fundamentals



Anaya Vardya and David Lackey
American Standard Circuits

I-007e Books

PEER REVIEWERS

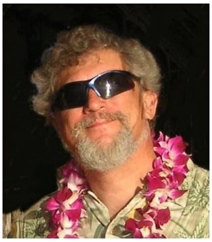
This book has been technically reviewed by the following experts in the PCB industry.

Lee Ritchey

Lee Ritchey is considered to be one of the industry's premier authorities on high-speed PCB and system design. He is the founder and president of Speeding Edge, an engineering consulting and training company. Ritchey conducts on-site private training courses for high-technology companies and also teaches courses through Speeding Edge and its partner companies. In addition, he provides consulting services to top manufacturers of many different types of technology products including Internet, server, video display and camera tracking/scanning products.

Prior to founding Speeding Edge, Ritchey held a number of hardware engineering management positions, including program manager for 3Com Corporation and engineering manager for Maxtor. Earlier in his career, he designed RF and microwave components for the Apollo space program among others. Ritchey holds a BSEE degree from California State University, Sacramento.

Tom Woznicki, *"The Flexdude"*



Tom Woznicki, CID+, is the founder of Flex Circuit Design Company in San Jose, California. He founded the company in 1992 after serving as a technical sales engineer for Rogers Corp.

He has designed over 1,000 flex and rigid flex circuits for fiberoptics, disk drives, medical equipment, telecommunications, military/aerospace and many other applications. Many of these circuits require leading-edge manufacturing methods, such as additive circuitry, laser processing and microvias, as well as assembly technologies such as flip-chip and chip-on-flex.

Tom served on the team that worked on the flex circuitry for the Mars Curiosity Rover. Since 2014, Tom has been part of the team working with MIT on the Transiting Exoplanet Survey Satellite (TESS), providing PCB design services, PCB design reviews, and project management services for the focal plane circuit boards for the satellite.

MEET THE AUTHORS

Anaya Vardya, President and CEO of American Standard Circuits

Anaya has over 30 years of experience in electronics manufacturing with experiences in the US, Canada and the Far East. He has over a decade of executive management experience in public companies manufacturing PWBs.

Anaya has managed a variety of facets of PCB manufacturing at companies like IBM, Continental Circuits, Merix, Coretec and now American Standard Circuits.

One of Anaya's peers described him as "...one of the most driven engineering managers I have ever worked with. He has a keen eye for details and recognizes the big picture." He has extensive experience in virtually every aspect of PCB manufacturing operations including supply chain management and quality control. It is this background that allows Anaya to not only serve as ASC's CEO, but as an expert ready to assist you with whatever questions you may have.

David Lackey, VP of Business Development and Flex Expert

David has been involved with manufacturing PCBs since 1980 and has worked in various shops, most of which specialized in high-end technology or military requirements. David has extensive experience building metal-core boards and PCBs requiring thermal management solutions, as well as flex and rigid-flex boards.

Having work experience in nearly all departments throughout the years, David has developed a strong engineering background and is knowledgeable in most industry technologies. His background enables him to work not only with buyers but with design engineers and quality and manufacturing personnel as well. Most questions can be answered on the spot without having to deal with multiple visits, e-mails, or calls.

The Printed Circuit Designer's Guide to...™ Flex and Rigid-Flex Fundamentals

Recognizing Important Recurring Issues Related to the Design and Manufacture of Flex and Rigid-Flex Circuits

By: Anaya Vardya and David Lackey,
American Standard Circuits

BR Publishing, Inc.
dba: I-Connect007
PO Box 50
Seaside, OR 97138-0050

ISBN: 978-0-9796189-4-9

Visit I-007eBooks.com for more books in this series.

Table of Contents


S-SIDE
STIFFENER SIDE
L-3 AVIONICS

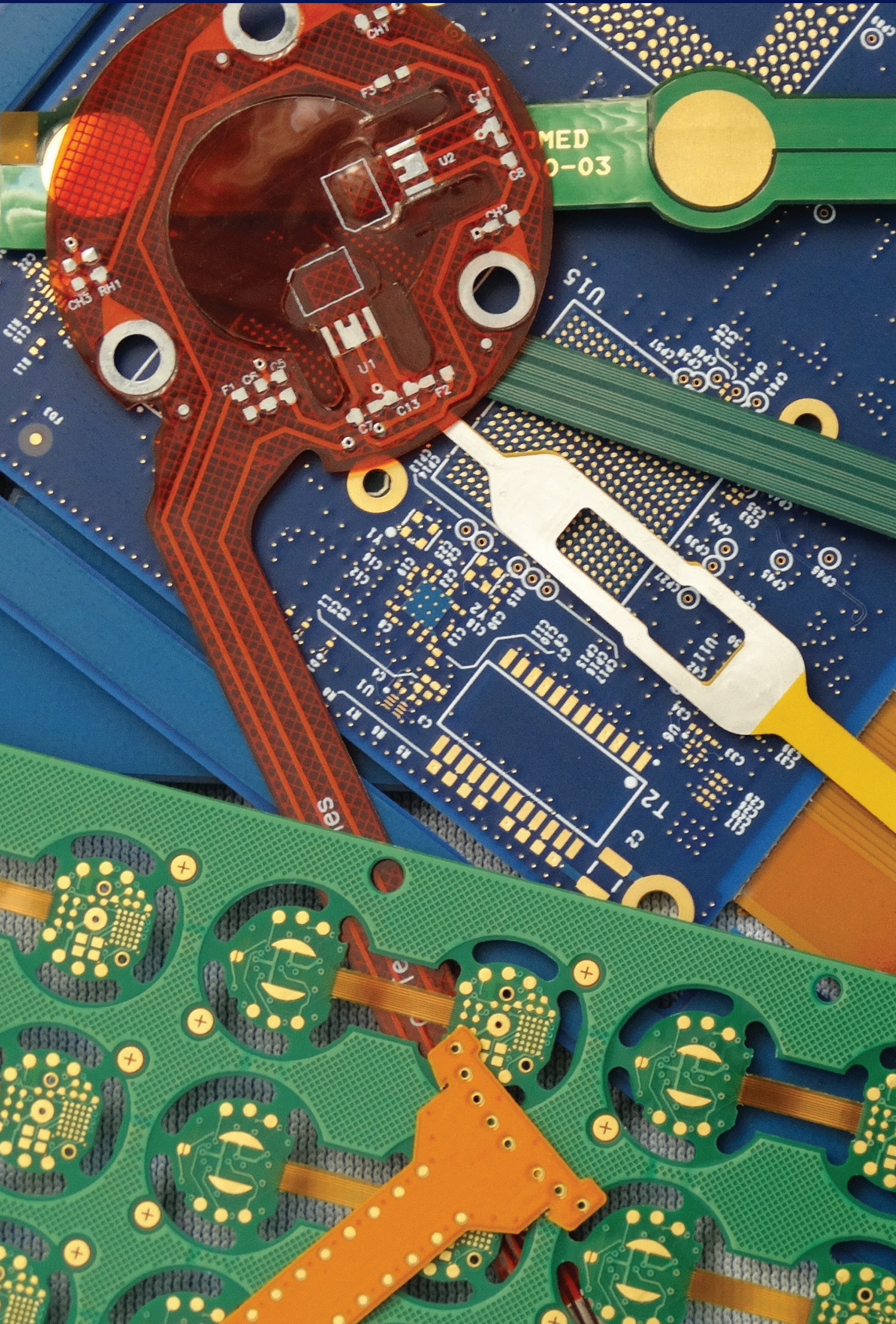
INTRODUCTION

Flex and rigid-flex circuits have proven to be among the most versatile and useful electronic interconnection technologies. They are also among the most complex. We conceived, developed, and published this guidebook in an effort to alert the PCB designer to some of the common issues that, if left unaddressed, can cause delay in getting the circuit manufactured. The book is most fundamentally about getting a proper documentation package into the hands of your flex and rigid-flex fabricators, and providing the knowledge you need to work with the manufacturer to solve problems before they occur. It is important to point out there are many benefits of flex and rigid-flex designs that allow for smaller and more complex packaging. A detailed discussion of the various benefits of flex and rigid-flex circuits are in Appendix A.

In truth, documentation may not seem to be an overly interesting topic, but it is a critical element of the business transaction that helps assure accurate communication between the flex circuit designer / customer and the vendor. If the information presented to the flex fabricator is clear, complete, and easily understood, the chances of getting the product right on the first pass are greatly enhanced. Unfortunately, flex circuit manufacturers frequently complain that up to 75% of the documentation packages they receive require some sort of clarification from the customer. As a result, precious time is wasted pursuing important missing information or clarifying unclear information that is vital to meeting delivery targets with a quality product. The missing or unclear information is potentially injurious to the bottom lines and reputations of both customer and vendor.

The detailed list of issues requiring attention will typically vary from design to design; however, there are some common issues that come up time and again. This book addresses the main issues of common concern that often require attention by the customer to assure a smooth journey through the manufacturing process.

The following are discussions of common errors of omission and commission which frequently hold up the production of a flex or rigid-flex circuit, or make it vulnerable to substandard manufacturing. These discussions are broken down into two main sections: material-related issues, and circuit design features and layout.



CHAPTER 1

CALLING OUT MATERIALS FOR FLEX CIRCUITS AND GETTING IT RIGHT

Only a limited number of material types are used in flex circuit manufacturing; however, materials and material callouts are the source of many phone calls between designers and fabricators, mostly for the purpose of seeking clarification. The following are some of the top recurring issues.

Missing Dielectric Material Callouts

Flexible and rigid-flex circuits are manufactured using numerous types of materials to meet a wide array of cost targets and performance requirements, both physical and electrical. Because of this variety, relative to the prospective concerns related to each choice, it is vitally important that the designer provide detailed information about the dielectric materials to be used. It is recommended that designers educate themselves about the choices available in terms of cost and performance. The Internet is packed with easily tapped information about flexible circuit materials and how they might be used. The PCB fabricator can also help with this topic. The basic flex material types are:

- Adhesiveless materials, which have no acrylic bonding the copper to the polyimide dielectric
- Adhesive materials, which have acrylic bonding the copper to the polyimide dielectric
- Flame retardant and non-flame retardant laminates, coverlayers, and bond plys

Figure 1 illustrates the difference in adhesiveless flex cores vs. adhesive flex cores.

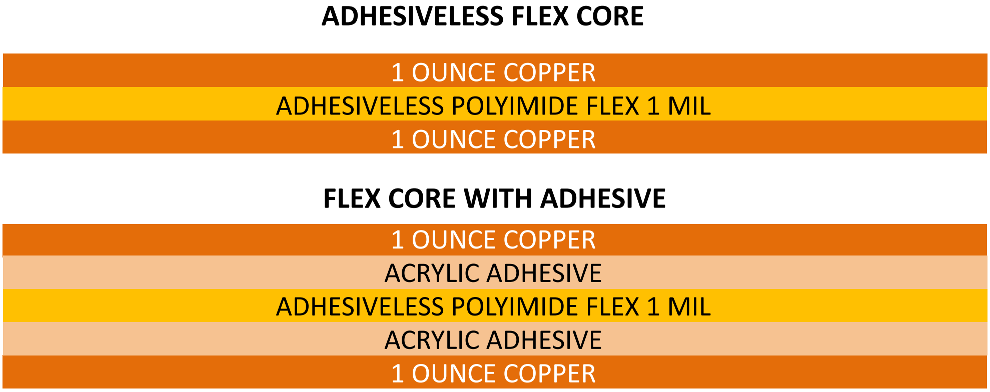


Figure 1: Adhesiveless vs. adhesive flex.

Each of the above have specific uses dependent on the end use needs. However, if designers are uncertain as to what their options are and what material might be best, they can always contact the PCB fabricator's engineering staff and ask for suggestions and recommendations.

Incomplete or Insufficient Rigid-Flex Base Material Type Definition

The base material chosen defines the performance limits of the rigid-flex circuit in process and in field operation in many applications. With most lead-free solders, the upper temperature excursions required for soldering can be as high as 260°C, which normally mandates the use of polyimide laminates. However, the material choice and its electrical properties can affect other performance issues. One key area is in the management of characteristic impedance of the circuit, and assurance of signal integrity, with higher-frequency circuit designs becoming ever more common (these latter subjects will be given more attention later). Temperature range requirements for the rigid laminates used in rigid-flex constructions must also be considered and addressed. The rigid material should be high-temperature capable. Polyimide glass laminate is a common callout, but available improved epoxy resins are often suitable.

Copper Type and Thickness Callout

While a number of different metal foils are available for making flexible circuits, copper is the most commonly used metal for making electronic interconnections. It is highly conductive, malleable (making it both flexible and foldable), relatively easily processed by etching and plating, and relatively inexpensive. The type of copper used most often for flexible circuits is rolled and annealed copper (RA copper), which has the best properties for dynamic flex applications. When designing the copper, type and thickness choice should match the electrical and mechanical requirements for the application.

Thicker copper is typically used for higher-power applications, and thinner copper for circuits that require repeated bending (dynamic flexing). There are many choices of copper thickness, but the most commonly used in the creation of flexible circuit laminates are presently ½ ounce (17µm or 0.7mils) and 1 ounce (35µm or 1 mil).

Often additional copper may be plated onto the circuit, and this should be accounted for in the specification as well. If the designer is uncertain, he should contact engineering support for guidance.

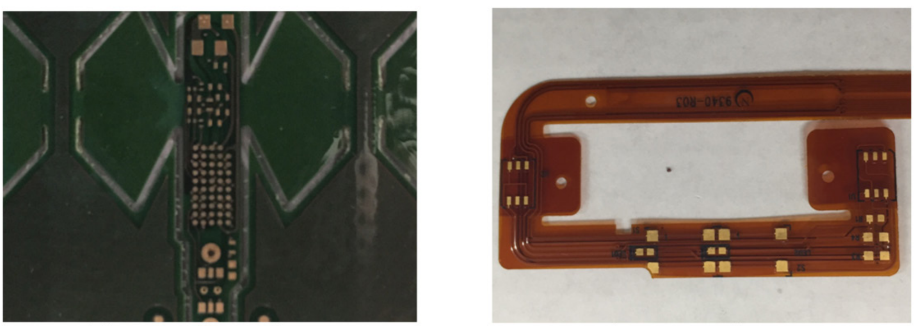
Coverlayer or Solder Mask Over Flex Circuits

Coverlayers, or cover coats, are polymer materials used to cover and protect the copper traces of the flex circuit product. As implied, there are a number of different options available for protecting the circuits, and they serve different design requirements in terms of cost, performance, and flexural endurance optimization. When specifying the choice, it is important to call out not just the type of coverlayer material but also the thickness requirement. This can be very important in certain types of constructions, especially when a flex circuit will experience dynamic flexing during use.

In terms of cost, a flexible solder mask is generally the least expensive. Some one- or two-layer flex circuits, that will not be subject to multiple flex cycles or extreme radius bends, can be coated with an epoxy-based solder mask that is designed to flex without cracking. This, however, is not recommended when the design requires any dynamic or extreme flexing. The other option is the laminated coverlayer. These are typically materials that have a makeup that is identical to the flex core material's and are best suited for dynamic flexible circuit applications.

The coverlayer material is a polyimide sheet with acrylic adhesive on one side. It is typically pre-machined to create openings in the sheet where the final finish is required. The coverlayer sheets are usually applied in a lamination press using special pads to assure conformity around the copper features on the flex layer. For rigid-flex circuits, the coverlayer is typically cut to only protrude into the rigid portion by no more than 50 mils. The purpose of this is to allow all the plated holes in the rigid-flex to be void of any acrylic adhesive, as it can affect the hole wall plating integrity.

Figure 2 shows an example of flexible solder mask and coverlayer being used in flex circuits.



Solder mask on flex circuit | Coverlayer on flex circuit

Figure 2: Flexible solder mask and coverlayer used in flex circuits.

It is worthwhile to note here that bond ply used to laminate flex layers together is like a coverlayer, but it has adhesive on two sides. It is further worth noting that prepregs (glass cloth which has been pre-impregnated with a thermosetting resin), used for making rigid circuits, are used in the construction of rigid-flex circuits where they serve in the role of bond ply.

It is important to note that coverlayer material can come in typical thickness intervals from .5 mil – 5 mils (12-125 μm) of polyimide and .5 mil – 3 mils (12-75 μm) of adhesive. Based on your design, the adhesive thickness requirement is typically decided by the copper thickness that it is being bonded to. The higher copper weight the more adhesive is needed. The same holds true for bond ply. Figure 3 illustrates a four-layer flex circuit and demonstrates the use of coverlayer and bond ply.

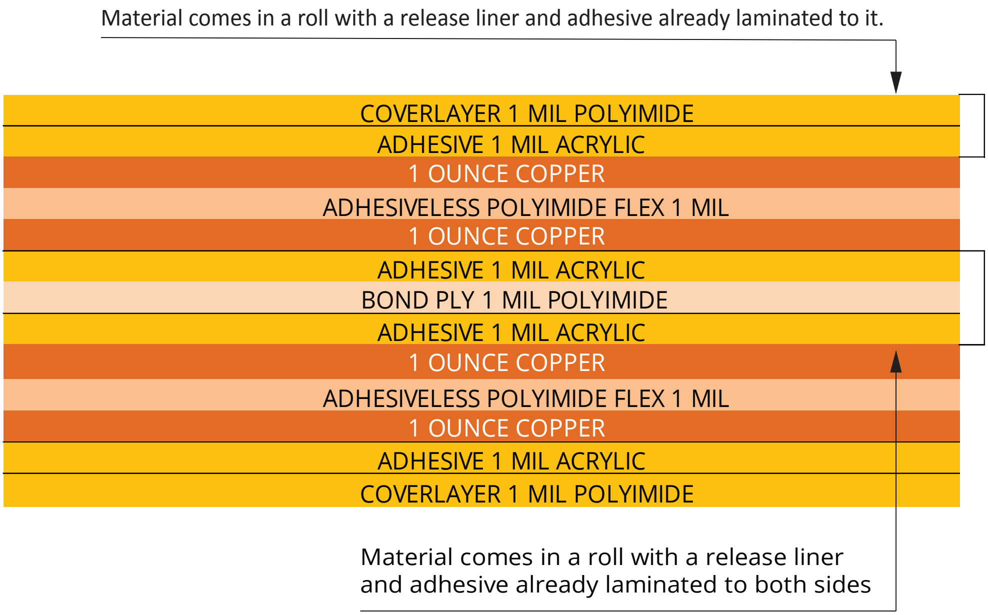
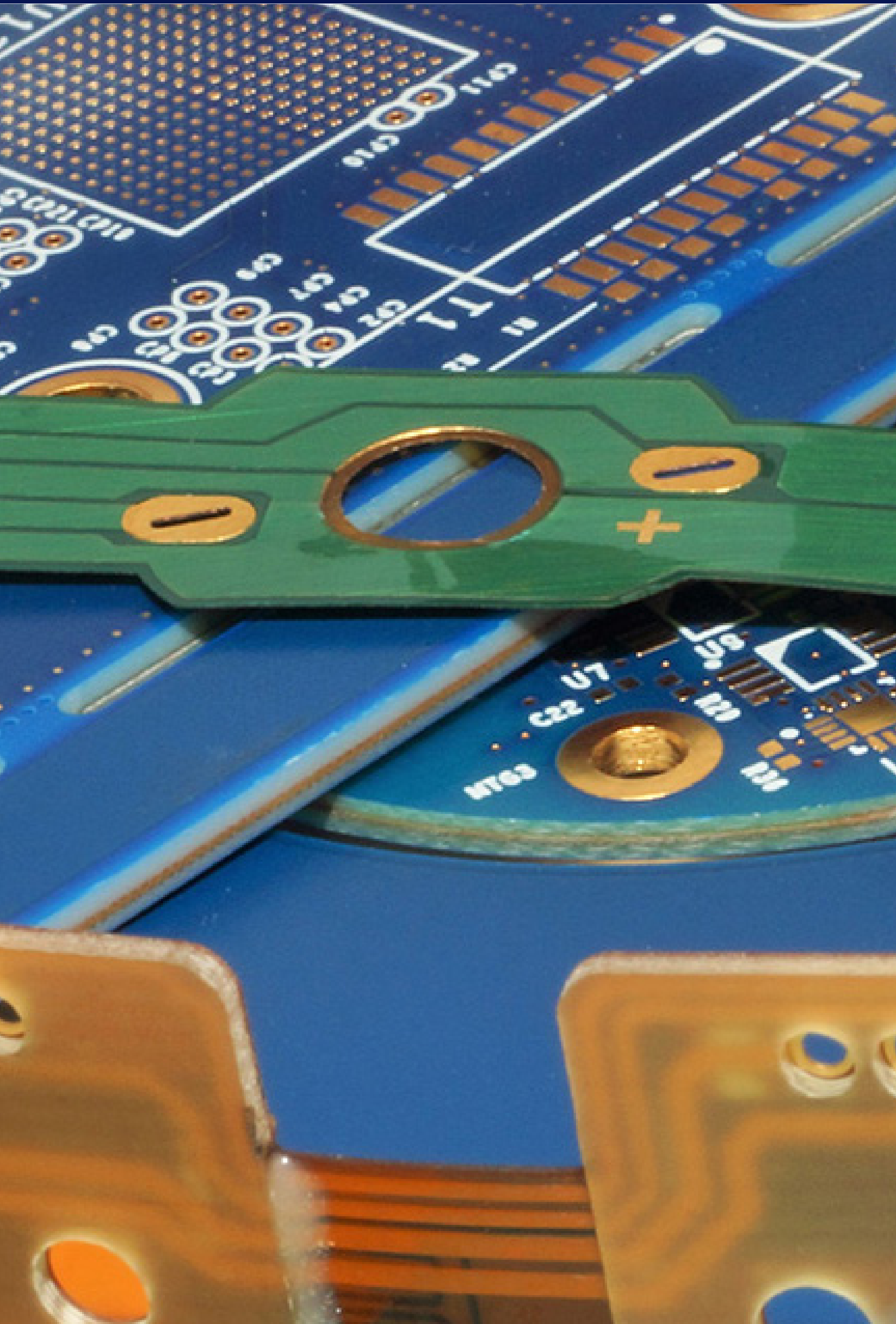


Figure 3: Four layer flex construction with bond ply and coverlayer.


U7
C22
MTG3

CHAPTER 2

DESIGNING FLEX CIRCUITS FOR FIRST-PASS SUCCESS

The design process is arguably the most important part of the flex circuit procurement process. The decisions made in the design process will have a lasting impact, for better or worse, throughout the manufacturing cycle. In advance of providing important details about the actual construction of the flex circuit, it is of value to provide some sort of understanding of the expected use environment for the finished product.

The electronics industry serves a number of different markets that do not always share the same product acceptability or reliability expectations. For this reason, the electronics industry, through IPC and other standards organizations, has developed a classification system that specifies what is expected of products for different classes. The system of classification is not intended to be a measure of quality. Rather, quality is a matter of conformance to a set of established requirements for a product in a given application. Therefore, quality products can be created in each of the classifications within the system. It is generally accepted that there are three classes of product. These have been defined by IPC standards as follows:

- Class 1 – Consumer products and products for non-critical applications where cost is normally the primary driver.
- Class 2 – Higher-order products in terms of quality and reliability expectations, including telecommunications, computers and general industrial.
- Class 3 – High-reliability applications including military, aerospace, automotive and medical products.

By defining the class of the product being designed, the purchaser is letting the manufacturer know what added controls to apply to the manufacturing process and the level of care they will need in the inspection process to ensure that the customer gets the product; one that is best suited to the application.

The following are discussions on matters of high importance to achieving first-pass success in securing quality flexible circuits from a flex circuit vendor. It is important to provide some information about the operational requirements for the flex circuit, especially if the circuit is to be used in a dynamic flexing application, such as for a disk drive read/write head assembly. The reason for this is the circuit vendor needs to provide a plan for proper layout strategy for manufacturing, one that accounts for the grain direction of the copper foil during the manufacturing process. This is because there is a measurable difference in terms of flexing performance between the machine and transverse directions of the copper foil.

Fabrication Specification Details

After the basic circuit design layout is completed, the next most important piece of information required is the fabrication specification. This document communicates to the fabricator all of the pertinent details for the physical construction of the circuit, and what is needed and expected in the final product. If this information is incomplete or inaccurate, or if a customer has requirements that cannot be reasonably met by a competent manufacturer, time will be unnecessarily lost, at a cost to both the customer and the vendor. For this reason, it is vitally important that the fabrication specifications are checked and rechecked before putting them out for bid. In the sage words of the master carpenter, “Measure twice, cut once.”

Manufacturing Tolerances

Manufacturing system operators need not only the dimensions of the part they are to manufacture, but also the tolerance for the important features of the product. With flexible circuits, this is something that must be done with thought, care, and consideration of the realities of flexible circuit materials.

With some features, design tolerances may be critical for the performance, fit, or further processing of the product (line widths, spaces, hole sizes, physical separation of features, positional accuracy, etc.). In these cases, the manufacturer can often employ methods to deal with the requirement on a localized basis. In the case of other features, the tolerance may be less critical, significantly less critical, or even non-critical. An important thing to keep in mind is that flexible materials are not as dimensionally stable as rigid materials, and while local features may be held in tight tolerance relative to each other, features from end to end may be less predictable. Given that flexible circuits are normally installed in some 3D form after assembly, the tight tolerances on planar measurements are often not necessary. If there are questions about a tolerance callout, the designer should contact the manufacturing engineer. It is always best to solve the problem before it becomes a problem.

Unclear Layer Designation (Rigid or Flex)

The purpose of a product specification is to provide clear, unambiguous instructions on the product's construction. In the case of a multilayer circuit design, this is vitally important. The relationship of internal circuit layers relative to one another has become increasingly important in not only assuring that correct interconnections are being made, but also in product performance, especially with controlled impedance designs and signal integrity issues. A number of different systems have been developed over the years to help assure that there is no uncertainty in the order of the circuit layers in the final construction. The fabricator's engineering staff can provide recommendations if needed. See the thickness and construction of each core in Figure 4.

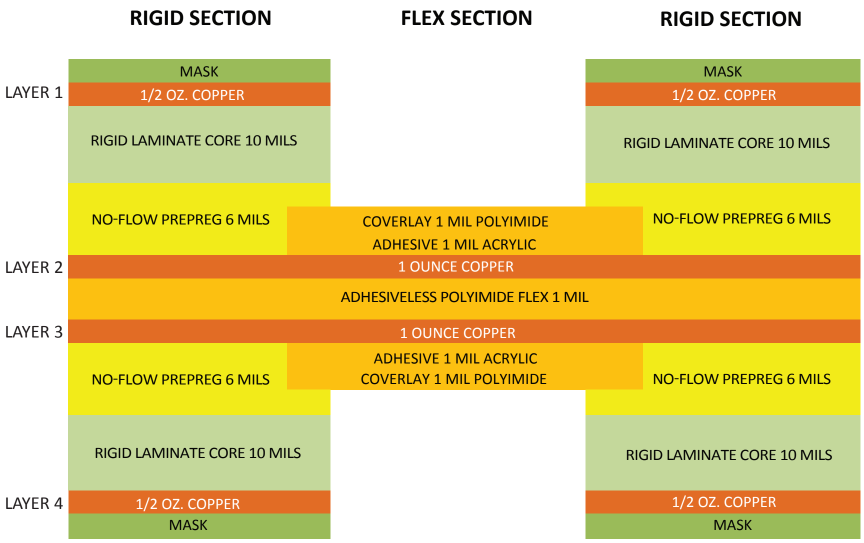


Figure 4: Example of four-layer flex construction.

Cover Layer Requirements Not Properly Called Out or Defined

Coverlayer and cover coat are terms normally reserved for flexible circuit constructions and they are by default a defining structural element of both flex and rigid-flex circuits. Coverlayers serve as a flexible solder mask of sorts, protecting the delicate circuits from damage and potential wicking of solder along circuit traces, while leaving open access to design features where interconnections are to be made to components by soldering. It is important to determine the thickness of a coverlayer to allow for maximum flexibility when desired, and also ensure you have chosen a coverlayer with a sufficient amount of adhesive on it to accommodate the copper weight. Coverlayers are also of particular importance in the design of areas where the circuit is to be bent either just one time, intermittently, or dynamically millions or even billions of times over its useful life. In the latter case, the dimensions and make of the flexible circuit coverlayer are critical. In dynamic flex circuits, there is a need to balance the amount of flexible materials on the sides of the conductors where flexing is to occur.

It is important to know and understand that there are different types of materials available for use as coverlayer materials, and that there is no single, ideal solution. These material choices include: materials that are laminated to the copper circuits using heat and pressure; materials that can be laminated and then photoimaged, like solder mask, to define points of connection; and materials that are simply screen printed on to seal traces, while leaving open features of interest for further processing or for making interconnections.

Number of Flex Layers

The vast majority of flexible circuits have just one or two metal layers. However, an increasing number of high-performance products now require high layer counts and high-density interconnect (HDI) design techniques. As layer count increases, so does the need for control in design generation to accommodate manufacturing process realities. It is also worth noting, while on the topic of layer count, that stiffness increases as a cube of thickness. That is, if one doubles its thickness, the stiffness goes up eightfold ($2^3 = 8$), and thus small increases in thickness due to increases in layer count can greatly decrease circuit flexibility. The converse is also true, of course. The following are some key concerns to be understood and addressed in the design process relative to flex layer count.

As is the case with any multilayer construction, core thickness must be provided with the assumption that copper is clad on at least one surface. The core thickness is generally understood to be the thickness of the dielectric material between the copper layers. The core material can be a simple single-sided piece of copper clad polymer, or it can be clad with copper on both sides. Many different core thicknesses are commonly available for flexible circuits, but the most common is 75µm, typically comprised of 25µm of base polymer (e.g., polyimide, polyester) with 25µm of adhesive (e.g., acrylic, modified epoxy) on either side to bond copper foil to the surface of the base polymer. Thinner and thicker core materials can be procured both with and without adhesive. It is recommended that designers check with their flex vendors for both their recommendations and the availability of the chosen material.

While the discussion so far been limited to flexible circuit core material, rigid materials are employed in the fabrication of rigid-flex circuits. Of course, any of the myriad core materials used in rigid multilayer circuits are also available to make rigid-flex circuits. However, once again, it is advisable to check with the flex manufacturer for advice as to what options are most common and readily available.

Separation Distance Between Flex Circuit Cores

When a product requires two or more cores, there is a need to define in the specification what the spacing requirements are between cores. The spacing can impact product performance (physical and electrical) and, most obviously, thickness. In some designs, the spacing between flex circuit cores may be filled with dielectric material, but with other designs the dielectric between flex cores in the flex area may be omitted in order to assure maximum flexibility (see Figure 5).

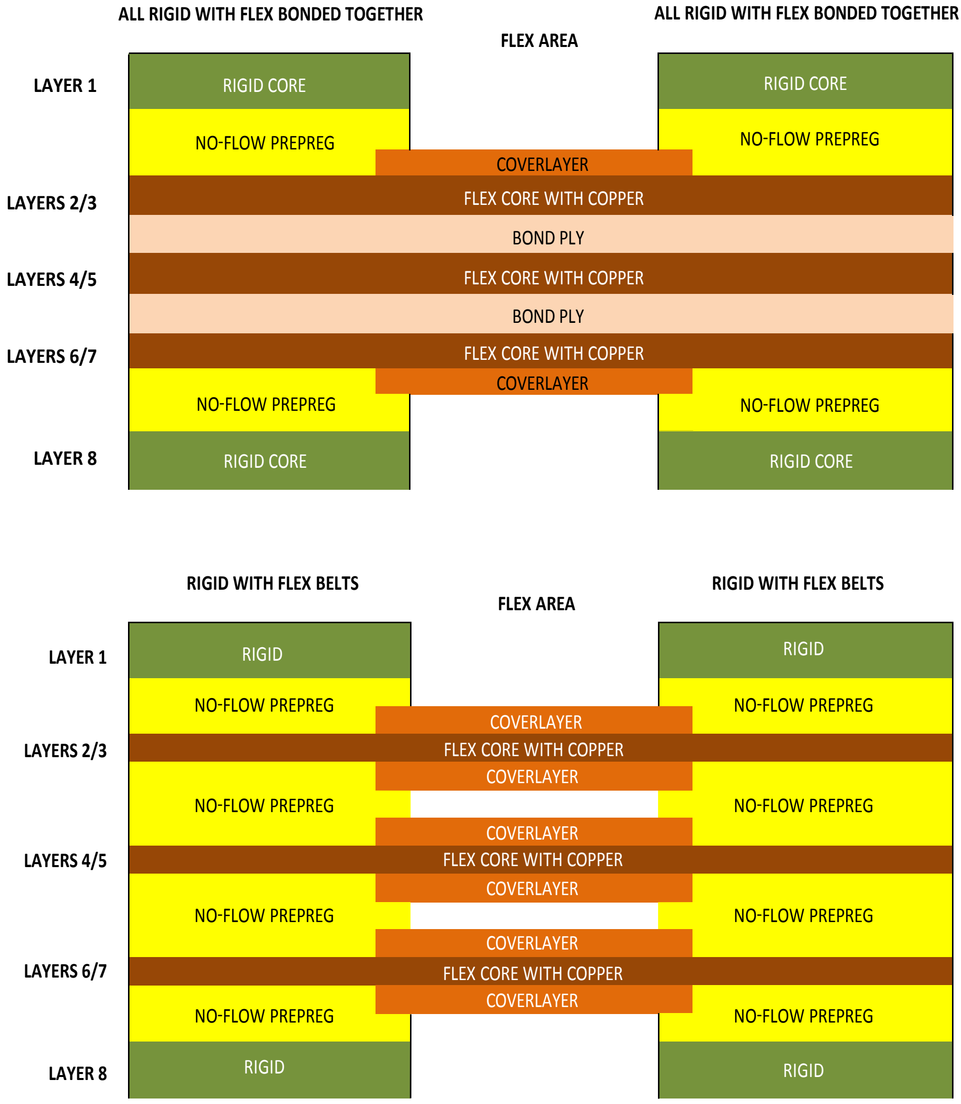


Figure 5: Bonded vs. unbonded flex areas.

If the core layers must be unbonded, this should be noted in the documentation. Those areas where bonding is to be avoided should be identified in the design artwork package. The unbonded areas must have a coverlayer applied to each exposed side (see Figures 5 & 6). In laminated areas, it is not required, and arguably a liability, when considering reliability for plated through-holes in the assembly process. Obviously, in areas where interconnection is required between multiple layers of internal circuits, a dielectric is required as shown in Figure 5.

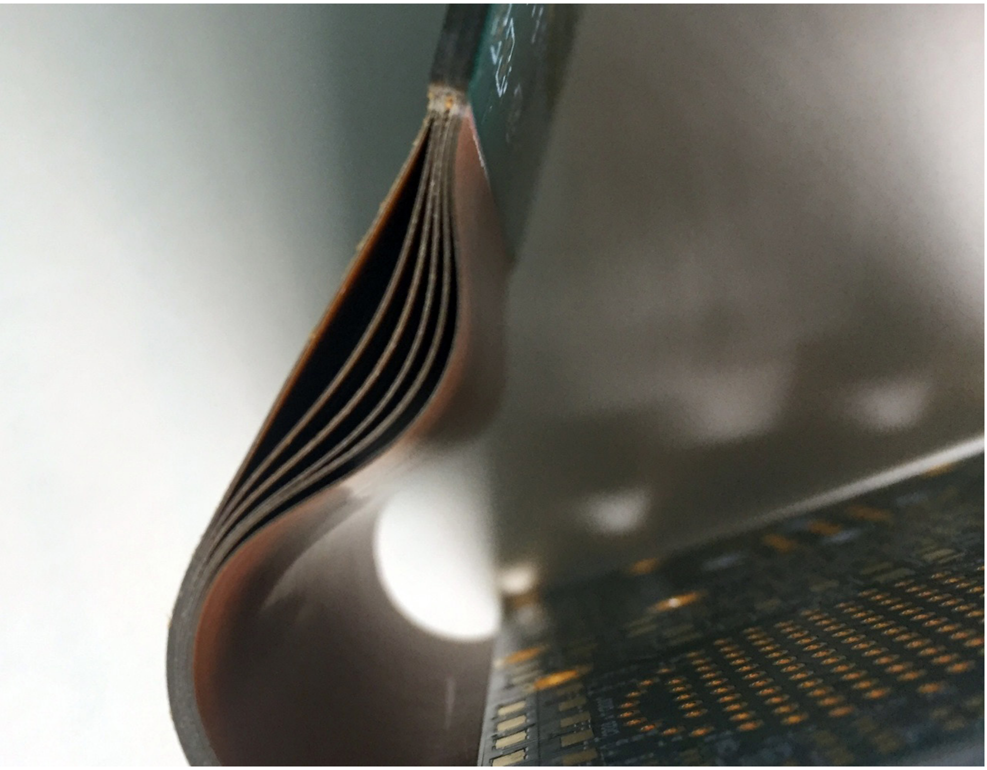


Figure 6: Completed PCB flex belts separated.

Circuit Layup Symmetry

It is a long-standing practice to design multilayer circuit structures with special attention given to the symmetry of the layers in the circuit. That is, the construction should be balanced from center to surface on both sides of the circuit board, whether rigid or flex. Balanced construction is for the rigid section of rigid-flex. It is preferred that flex cores be balanced, too; however, it is not mandatory. This is most easily accomplished by choosing core materials that feature two metal layers. It’s more difficult to control this balance when using core materials with an odd number of metal layers.

The symmetry requirement applies to the layer count as well as the overall copper area of the different layers. In this regard, the retention of maximum copper in the circuit pattern is beneficial to manufacturing, because copper is a dimensionally stable element of the construction. The base polymer is by its very nature flexible, and not intrinsically dimensionally stable.

Designing for Bending - Understanding the Important Issues

Bending and flexing are hallmark functions of a flex circuit, whether the circuit is bent once or flexed millions of times. Understanding specific design rules for flex circuits is crucial for success in the field.

The first thing to keep in mind is perhaps the most obvious: The thicker the cross-section of the material stack in the bend area, the less flexible it will be. It is important to keep the area where bending or flexing is to occur as thin as possible. Ideally it should be a single metal layer, if possible. This is especially true for dynamic flex circuit applications.

In other applications where simple bending to shape is required, thickness is less important, up to a point. There is a minimum optimal bend radius which is largely a function of the properties of the copper foil. The less ductile the copper, the larger the bend radius should be. However, there are some long-established “rules of thumb” to keep in mind, courtesy of Joe Fjelstad’s ***Flexible Circuit Technology***:

- For a single metal layer, the minimum bend radius should be on the order of 3-6 times the thickness of the circuit.
- For two metal layers, the minimum should be 7-10 times the thickness of the circuit.
- For multilayer flex, the minimum bend radius should be 15-20 times the thickness or more. If the multilayer sections are unbonded in the bend area, this rule might be relaxed.
- For dynamic flexing applications where high flex cycle life is required, again the design should be limited to a single metal layer and the bend radius made as large as possible (20-40 times thickness or more).

In short, always choose materials, copper metal foils, and thicknesses that will accommodate the proper bend radius. See Figure 7 for an example of bend radius to circuit thickness.

As a closing thought, finite element modeling can be very useful in determining the bend limits of a particular structure. It is recommended that the designer check with his vendor for recommendations based on their past experiences. It is normally much less expensive and more expedient to "go to school" on the lessons learned over time by others.

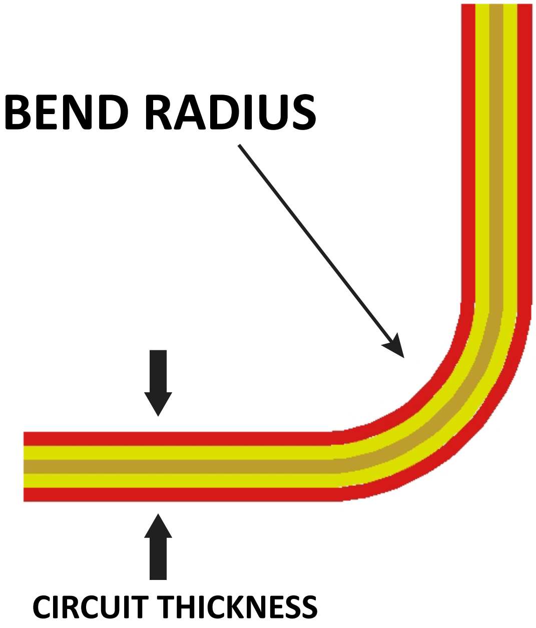
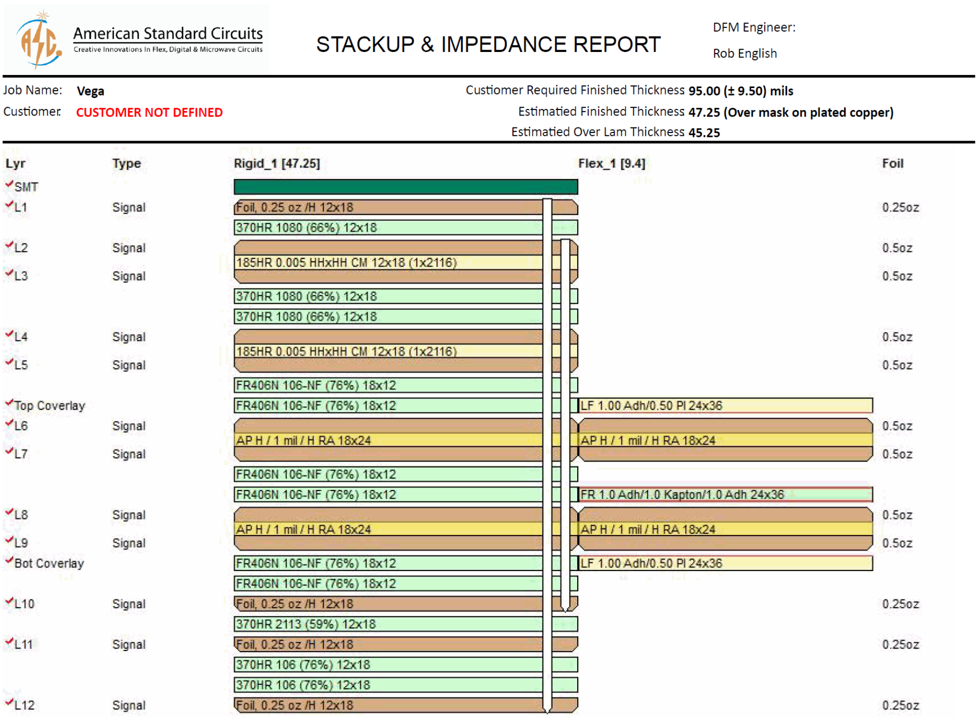


Figure 7: Illustration showing bend radius.

Dealing with Controlled Impedance Designs

With the increase in higher-frequency circuits, the control of electrical properties in key circuit routes in a design is becoming more critical. Flexible circuits are well suited to the challenge due the nature of the base materials, which are nominally homogeneous polymers of relatively low dielectric constant, as opposed to the heterogeneous epoxy glass composite materials used for rigid circuits, which tend toward higher Dk values and less suitable dissipation factors.

Several factors need to be addressed to get the right characteristic impedance values from a design: trace width, dielectric location, and the thickness of the dielectric between ground and traces. The properties of the dielectric are also of importance. In the design process, it is best to provide nominal values for the controlled impedance trace widths and then let the vendor adapt the process to deliver the right product. To determine the feature size requirements, the designer can use one of several impedance solvers available on the market. See Figure 8 stackup with impedance values that ASC generated.



Impedance Table

Layer	Required Impedance (Ohms)	Tolerance (Ohms)	Impedance Type	Upper Ref	Lower Ref	Design L/W (Mil)	Design Space (Mil)	Finished L/W (Mil)	Finished Space (Mil)	Artwork L/W (Mil)	Calculated Impedance (Ohms)	CoPlanar Spacing (Mil)
L3	100.0	±10.0	diff_stiripline	L2	L4	3.00	3.50	3.00	3.50	3.50	97.66	
L5	100.0	±10.0	diff_stiripline	L4	L6	3.00	3.50	3.00	3.50	3.50	97.46	
L8	100.0	±10.0	diff_stiripline	L7	L9	3.00	3.50	3.00	3.50	3.50	62.35	
L10	100.0	±10.0	diff_stiripline	L9	L11	3.00	3.50	3.00	3.50	3.25	100.04	

Figure 8: Stackup and controlled impedance model.

In general, tighter tolerances for etched features are possible with flex circuits because of the lower profile adhesion treatment or “tooth” of the copper. When the design allows, the use of thicker flexible dielectric substrates can ease the etching challenge because thicker substrates allow for wider signal lines. The wider circuit trace can be more easily fabricated to meet the tight tolerances needed for controlled impedance circuits.

To achieve greatest flexibility, the ground plane can be hatched. Improving flexibility by cross-hatching the plane layers in the flex portion instead of increasing thickness is helpful. Remember that hatching will have an effect on the impedance value. Most fabricators have the modeling tools necessary to see if a cross-hatched (preferred) design is feasible and will allow for the needed impedance callout. See Figure 9 showing cross-hatched area on the flex core.

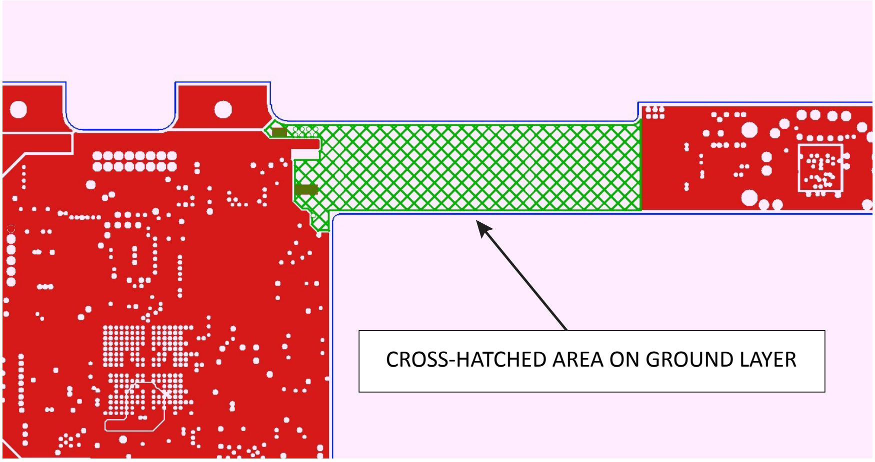


Figure 9: Cross-hatched ground layer.

In terms of impedance structures there are several different constructions possible using flexible circuits.

The first and simplest is the co-planar stripline construction. The circuit is produced with just one metal layer, alternating single and ground features. These constructions are well-suited to higher characteristic impedance designs. A drawback of these designs is that they are susceptible to EMI noise.

The second type is the microstrip circuit, a two-layer flex construction with one metal layer devoted to ground. These circuits, nominally targeted at 50 ohms, have been successfully employed in transmission line applications, and are often used for single-ended interconnections. Higher characteristic impedance designs can be built with microstrip construction, but flexibility normally suffers.

Stripline circuits are the third basic type of controlled impedance construction. These circuits feature three metal layers with ground

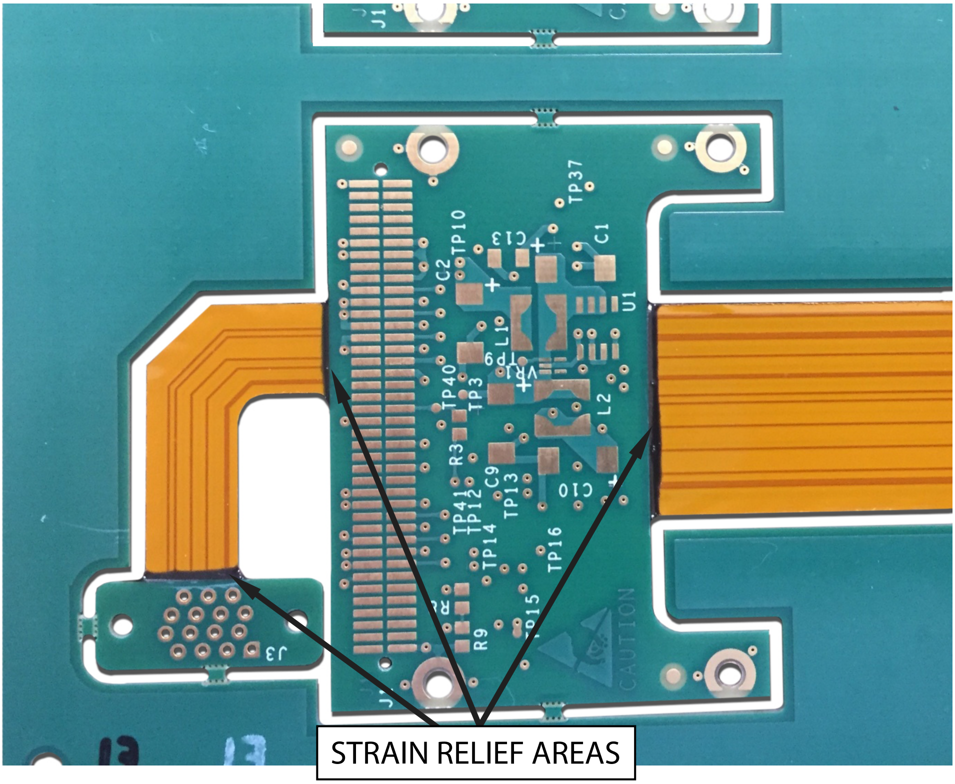


Figure 10: Epoxy strain relief.

layers on both sides. Excellent signal integrity can be achieved with this construction, however they tend to not be very flexible due to the extra dielectric and metal foil used. Stripline circuits are often designed to 100 ohms and are frequently used for differential pair interconnections.

There is also a stripline construction in which ground lines surround the signal lines to approximate coaxial cable constructions. These structures are of greatest interest where crosstalk is a concern and where maximum signal integrity is required. To improve performance, some designers stitch the top and bottom grounds together through the ground traces with plated through-holes, but this can be a costly process.

Important Mechanical Design Concerns

Flexible circuits are much thinner and generally less mechanically robust than their rigid counterparts, therefore there are some important issues related to the mechanical requirements in flex circuit design.

First is the matter of the plated though-hole. Through-hole reliability is critical to the electrical performance of a flexible circuit. While there are no hard rules relative to plating thickness in the though-hole, a higher nominal value is generally going to provide better long-term performance and greater resistance to barrel cracking and other thermal and stress-induced failures.

With respect to positioning of holes in the body of the flex circuit, there should ideally be a minimum of 50 mils of clearance between the edge of any such hole and the edge of the circuit or transition areas between rigid and flex. Placing vias closer to the flex and rigid interface can cause issues with possible chemistry leaking under the rigid portion and causing shorts or contamination.

In rigid-flex circuits, it is recommended that some strain relief be provided at the interface between the rigid and flexible areas of the design. The transition area is highly vulnerable to failure at the interface due to the naturally occurring stress concentrations which manifest there. The simplest solution is to provide a bead of polymer at the interface to move the stress point back from the edge. See Figure 10.

Other Specific Issues of General Concern

This book is not intended to cover all the different issues related to flexible circuits, but instead provide a short list of key issues that often fall through the cracks based on our experience. The following examples are offered to help readers avoid certain traps that have ensnared flex circuit designers in the past.

1.) High Layer-Count Designs:
There are a variety of challenges related to high layer-count flex designs (e.g., a 20-layer circuit with two rigid layers and 18 flex circuit layers). High layer-count designs are difficult for rigid circuit manufacturers and they are even more difficult for flexible circuit manufacturers, owing in part to the inherent dimensional instability of flexible base materials. Because of this, it is highly recommended that the designer engage early on with the fabricator to understand these issues and to work through any concerns with the supplier. Every design is unique, but experience can be a great teacher and the manufacturing engineer can be a great source of help in steering clear of the major pitfalls.

2.) Bookbinder Constructions:

The term "bookbinder" comes from the days when book pages were bound in sections by sewing them down a center line. In the final construction, the center pages were more prominent and the outer pages withdrawn on the edges of the sections. Possibly it was noticed that all of the pages could be made flush if they were cut to different widths before the stitching, though it was generally much easier to cut the pages after stitching and folding. In flex circuits, the term is applied to the staggering of the length of circuit layers through the bend within the design to provide greater ease of flexing in multilayer and rigid-flex designs. The technique is accomplished by adding slightly to the length of each succeeding flex layer, moving away from the bend radius. A common rule of thumb is to add length equal to roughly 1.5x the individual layer thickness, but the value can vary based on the tightness of the bend and the number of layers. Therefore, it is recommended that some modeling be carried out in advance of committing to manufacture.

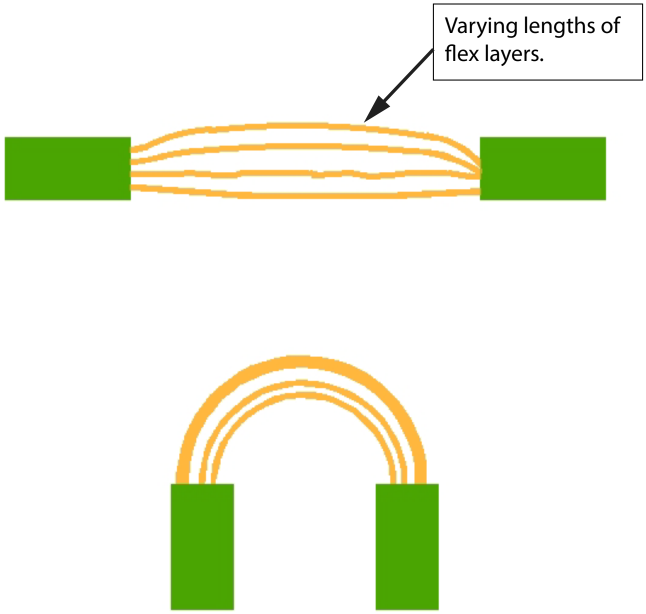


Figure 11: Bookbinder construction.

A paper doll mock-up can be very instructive as a quick check. The extra length with each succeeding layer helps defeat whatever tensor strain might have otherwise built up in the outer metal layers of the multilayer flex, and it prevents buckling of the center of bend layers. If there are questions about the practice, the designer is advised to check with the vendor in the early stages for guidance and assure better first-pass yield. It is important to note that bookbinder requirements are extremely costly to manufacture. It requires a lot of additional up front tooling and fixturing costs. See Figure 11.

3.) Copper Layers Greater than 75µm (2 oz. per square foot):
Thicker copper foils are useful in many flex circuit applications where they can serve to address high-current, high-power requirements for both discrete traces and power and ground planes. They can also help hold the shape of a flex circuit which is formed to fit an application. The challenge of thicker copper is that it is more difficult to process the thicker copper and at the same time hold feature size accurately. This is because etching, which is the most common method of defining circuit features, is an isotropic process which works in all directions while at the same time undercutting etch resist and leaving traces narrower on top and wider on the bottom.

A potentially effective alternative is to begin processing with a thinner copper base and then pattern-plate additional copper in the areas where it is needed. That said, thick copper circuits are not necessarily to be avoided, but they should be approached with a full understanding of the issues. It is recommended that the designer consult with the fabricator about any issues and options before committing to thick copper.

4.) Anchoring Pads:
On flex circuits and rigid-flex circuits when the flex layer is an outer layer, there is a weak point at the conductor/pad interface. Any stress to this area can cause a fracture of the conductor at this point. The best way to prevent this is to provide specific reinforcements to the isolated pads on flex layers. Anchoring the pads will assist with this type of design. There are a number of ways to potentially anchor pads. Some of the popular ones are illustrated in Figure 12.

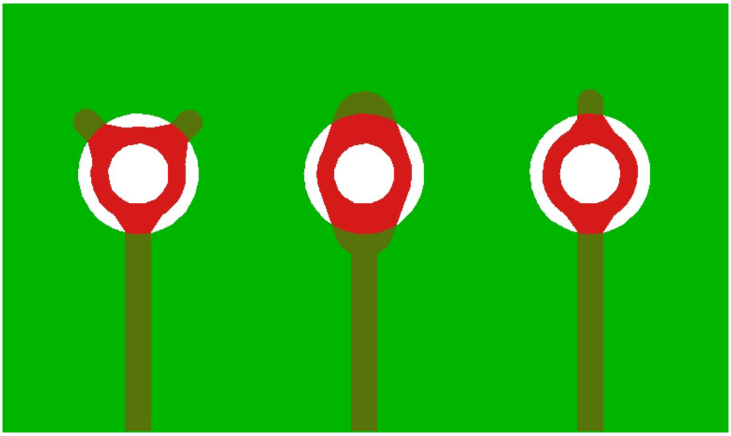


Figure 12: Anchoring pads.

5.) Creating Holes Located Only in Flexible Sections of Rigid-Flex Boards:

In rigid-flex circuits, it is generally desirable to limit the drilling of through-holes to those areas where the rigid and flex areas are laminated together. However, owing to the diversity of design needs, there are occasions when holes are needed in the flexible areas. This requirement can have a significant impact on both the processing and the physical performance of the design. Figure 13 illustrates an example of this.

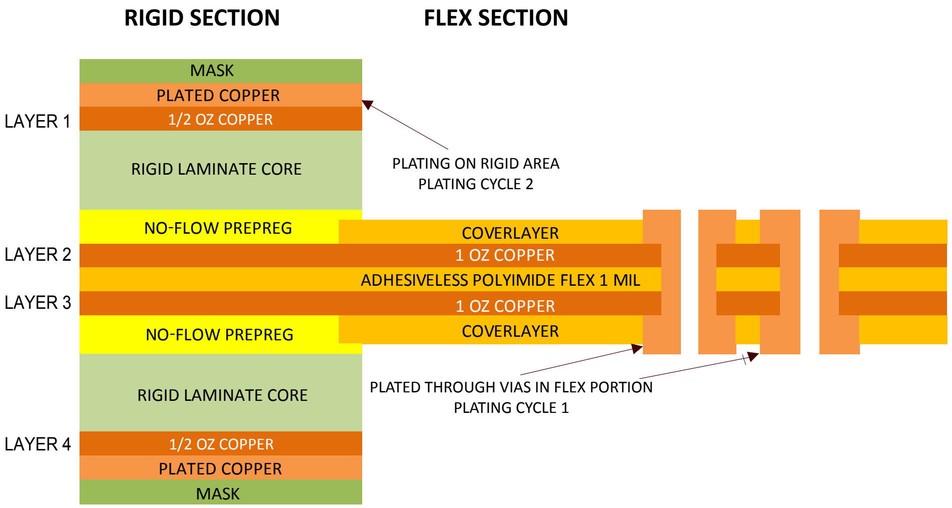


Figure 13: Rigid-flex with PTH in both the flex and rigid areas.

The processing aspect is related to the fact that at a minimum, a separate drilling step is required and if the hole requires plating, all process steps associated with plating must be carried out. The prospective impact on cost is readily apparent. It is well worth discussing the design with the fabricator's engineers before locking in such approaches to see if there might be a better solution. If not, at least the manufacturing engineer might be able to help suggest ways to make the circuit easier to manufacture and more reliable. With regard to reliability, any holes in the flexible sections should be kept out of the area which will experience the most flexing as these can cause stress risers and non-uniform performance.

6.) Rigid-Flex Circuits with More than Ten Layers or Three Flexible Cores:

Hopefully, it can be appreciated by the reader that the greater the number of layers in a rigid-flex circuit, the greater the opportunity for errors, especially the compounding of small errors which are insignificant in and of themselves, but which grow with the increase in handling and processing. It is not possible to make blanket design recommendations on this topic, because the potential variety of designs is virtually infinite. Thus, it is highly recommended that the designer check with the fabricator's manufacturing engineers to get their thoughts on what issues might come up and what problem preemptive design choices might be made.

7.) Flex Section as an Outer Layer:

One of the simplest forms of rigid-flex are those with flexible sections on the outer surfaces rather than on the inside. It is possible to make a two-layer rigid-flex circuit with a flex circuit on one side and none on the other, with the two sides interconnected by plated through-holes. To assure that the flex areas remain flexible after processing, there are special processing requirements, such as making sure that the rigid and flexible sections are not inadvertently bonded together. Additionally, when utilizing a flex core as an outer layer in a multilayer construction it is important to note that all copper features are typically plated on the outer layers. If it is your intention to flex this device to any extreme or more than once it would be advisable to have only the holes and pads plated on the flex circuit side. See Figure 14.

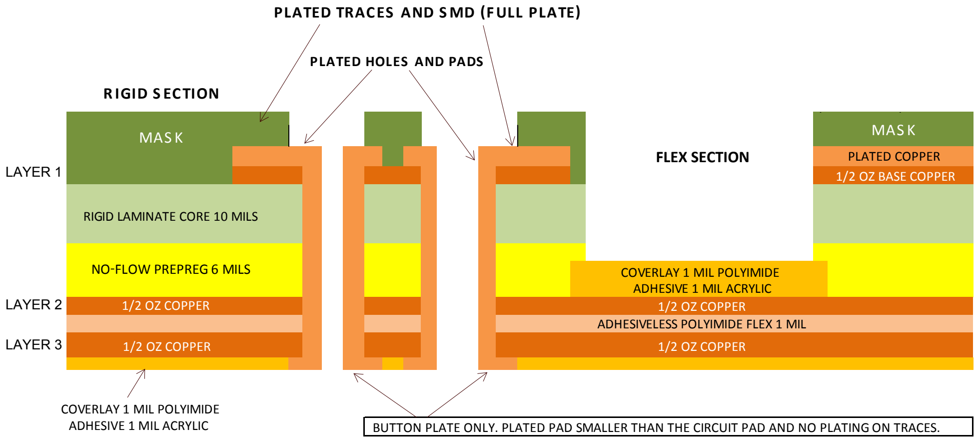


Figure 14: For rigid-flex circuits that must flex during operation, plate only the through-holes and pads on the flex circuit side as shown here.

Electrodeposited copper is brittle and plating the traces on the flex side may crack when flexed. These structures are relatively straightforward in terms of processing but it is once again worthwhile to check with the fabricator's engineers for review and comment before submitting the design for quote.

8.) Flexible Circuit Structure with Multiple Discrete Breakout Sections:

One of the most attractive features of flexible circuits is their ability to serve as miniature wiring harnesses. Flex circuits can interconnect—in three-dimensional space—the various elements (e.g., modules, switches, and power sources) of an electronic product. In many cases, such as with one and two metal layer circuits, this is a primary objective that can be easily accomplished by simply designing the circuit with discrete routes to desired terminations in panel form and then routing, punching or cutting the circuit from the panel. In the case of multilayer and rigid-flex circuits, this activity is not so simple and much more attention to design and process detail is required. Because of the infinite variety of design possibilities, it is not possible to give a generic recommendation beyond consulting with the fabricator's manufacturing engineer to pre-determine a best approach.

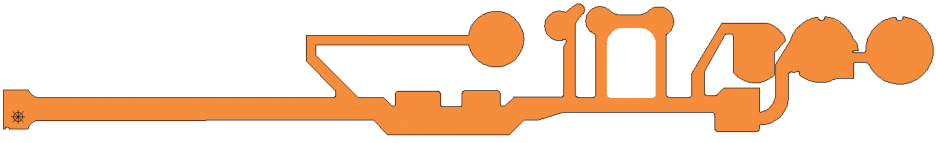


Figure 15: Flex with multiple discrete breakout sections.

9.) Flex Layers with Multiple Layer Separations and Discrete Routes:

As with the previous discussion, it is possible to prefabricate internal layers with the same multiple discrete routes and, moreover, to route them from separate flexible layers of the design. It is a fundamental compounding of the challenge described earlier and as such it requires even more attention to detail and even more consultations with engineering staff because the process planning can be quite complex. See Figure 16.

10-LAYER RIGID-FLEX

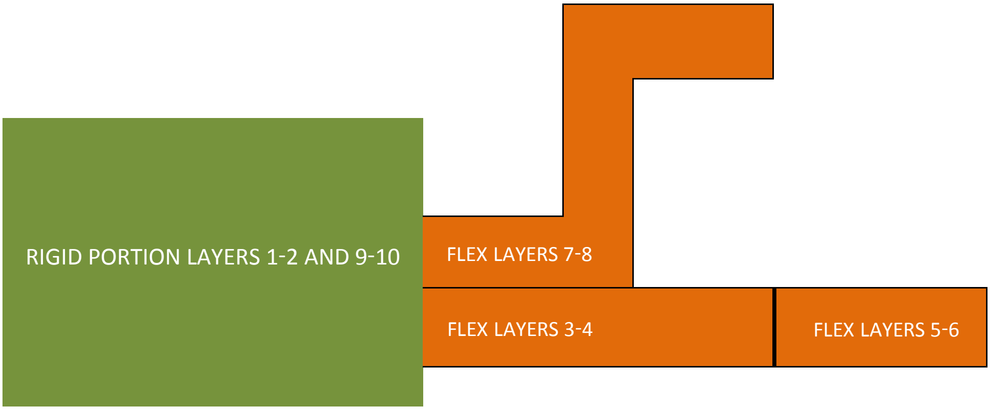
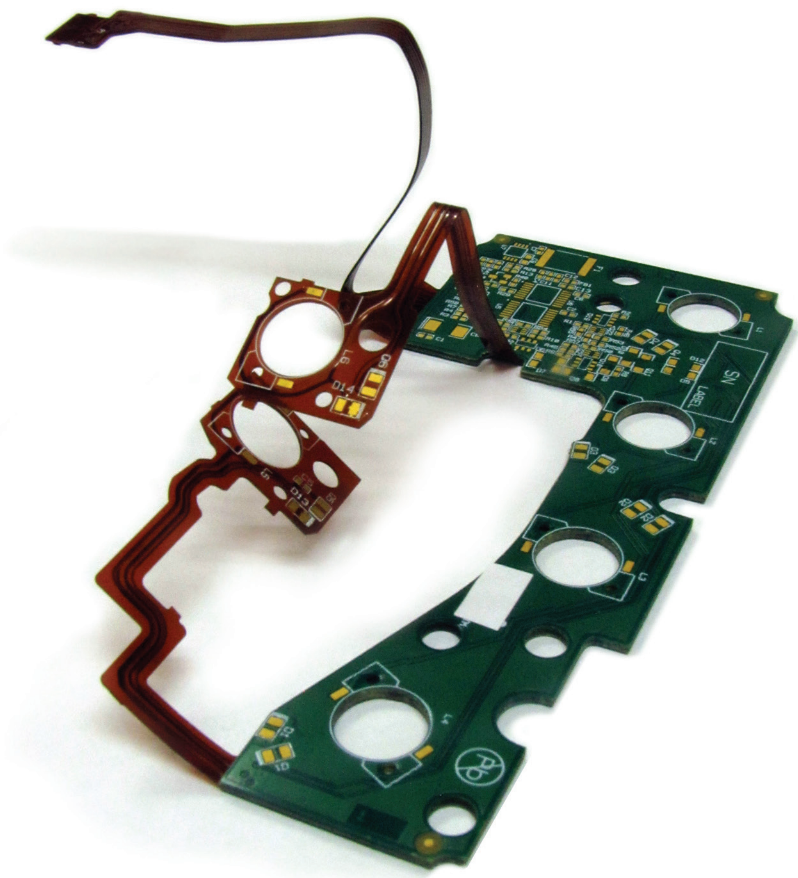
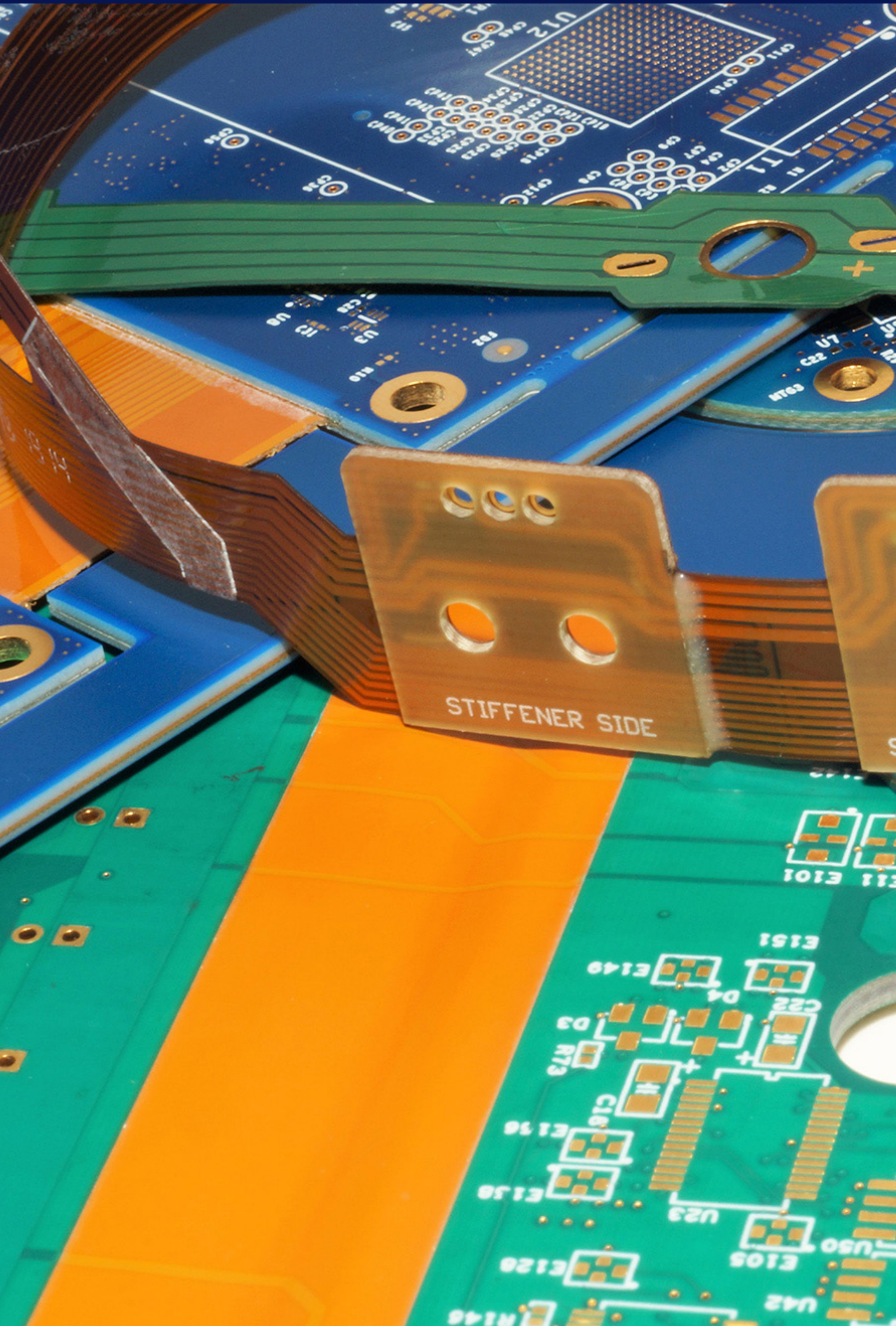


Figure 16: A 10-layer rigid-flex with multiple layer separation and discrete routes.




STIFFENER SIDE

CHAPTER 3

REVIEW OF GENERAL REQUIREMENTS FOR FLEXIBLE CIRCUIT DATA PACKAGES

The fundamental purpose of this book is to provide in abbreviated form a discussion of key matters related to the design and manufacturing of flexible circuits to help designers avoid delays during manufacture. Here is a summary of the key bits of information designers should understand to assure a desired outcome on first pass. The points of discussion here are general and not exhaustive. As has been said in earlier sections, the diversity of design makes every product unique. With that caveat, the following is a list of items deemed generally important when designing flex and rigid-flex circuits. Appendix B also has a sample set of notes for a rigid-flex part.

Product Class Definitions

Having an upfront statement of the intended use of the flex circuit will help to frame the expectations in terms of reliability and cost. Defining the class of the product lets the manufacturer know which added controls to apply to the manufacturing process and the level of care required in the inspection process to ensure that the customer gets the right product.

Specified Materials for Circuit Construction

While there are only a limited number of flexible circuit materials in wide use, there are many potential candidates and, therefore, a need to constrain the options to meet expectations. When specifying materials, the designer should inform the manufacturer of the type of base polymer to be used and the type of adhesive desired (if an adhesiveless laminate is not to be used). In a multilayer construction, it is highly recommended to use only adhesiveless flex laminates. The designer also needs to define the type and nature of the copper weight to be used. Normally the copper for flex is a foil that is rolled and annealed; however, some of the thinner copper weights (sub 1/2 ounce) are typically only available in electrodeposited copper and not recommended in a dynamic design.

It is also important to specify the nature and type of coverlayer material and stiffeners to be used. In short, any material that is to be used in the construction and stay with the final part must be identified.

Hole Pattern with Sizes and Plating Requirements

Holes are not always required for flex circuits, but usually they are required for vias or for making connections to leaded components or connectors. Thus, documentation must specify the size, number and location of any holes required for the circuit design (see Figure 17). This data is easily extracted from digital design files and will be critical to the vendor to determine the cost of the manufactured circuit.

DRILL CHART			
ALL UNITS ARE IN MILLIMETERS			
FIGURE	SIZE	PLATED	QTY.
+	0.2	PLATED	480
D	0.6	PLATED	30
Ⓟⓖ	0.55	PLATED	15
⑦	0.5	NON-PLATED	60

Figure 17: Hole size table.

Conductor Layer Count

The majority of flexible designs have only a single metal layer, but a substantial number of flex circuits feature two metal layers. Multilayer and rigid-flex circuits are less common and there is need to provide layer counts for the circuit design. Layer count is perhaps the most obvious indicator of overall circuit complexity and is important for quoting and production planning purposes.

Circuit Design Artwork

Proofing circuit artwork and checking the design against various design rule requirements and objectives (e.g., design for manufacture, design for reliability, etc.) are important and necessary conditions for release to manufacturing. Ideally, the information is supplied in digital form in one of the major CAD formats. The best format is ODB++. The CAD information provides definition of what the circuit will look like, where terminations will be located and how they will be shaped. It is also important in case the design needs to be modified or adjusted by the flex manufacturer to compensate for processing. The final design contains information needed to provide an accurate and realistic quote.

Cross-Sectional View of Circuit Construction

A cross-sectional or side view of the finished circuit provides the manufacturer with a scaled view of construction materials. The cross-section also lets the quality control department know what to inspect in terms of circuit layout since it offers a means of predicting overall thickness. See Figure 18.

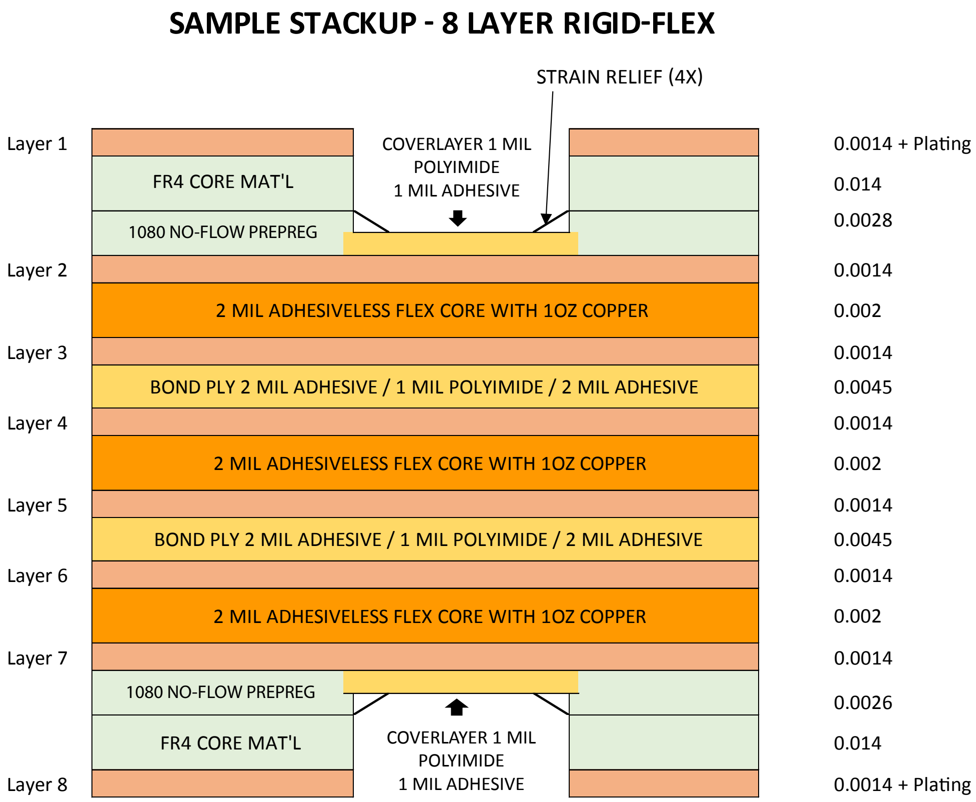


Figure 18: An example of an eight-layer rigid-flex.

Coverlayer or Covercoat Opening Locations

Coverlayer and covercoat openings physically define the features that need to be accessed to attach components, interconnect or testing probes (see Figure 19). It is key that the documentation package defines all such locations accurately. It is different than designing a solder mask layer since mechanical tolerances, coverlayer adhesive squeeze-out and location tolerances have to be considered. It is important to note that any surface mount or openings with 90 degree corners usually means the coverlayer must be laser cut and cannot be done on conventional CNC equipment. This would be a cost adder to the design; however, in many cases it cannot be avoided.

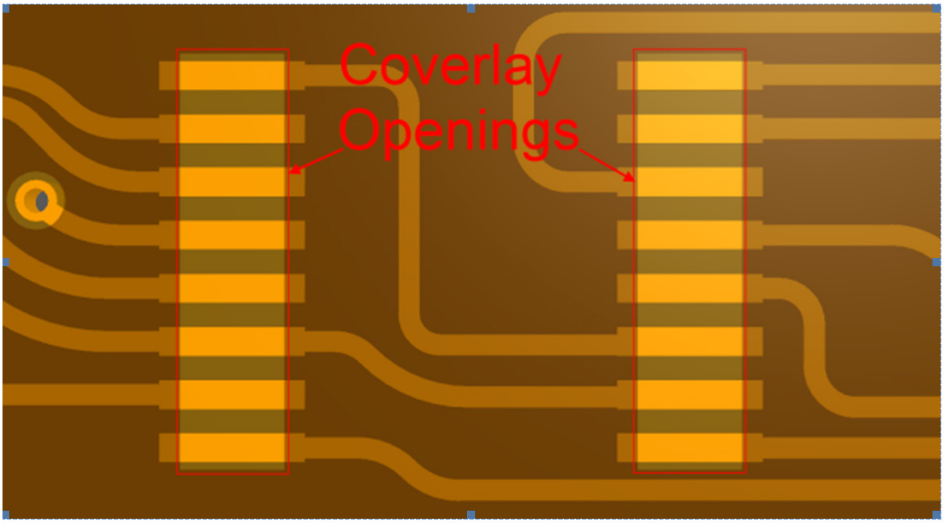


Figure 19: Coverlayer openings.

Circuit Outline with Dimensions and Datum Features

The physical outline of the finished circuit needs to be accurately documented to define the periphery of the flex circuit material relative to the circuitry itself. The outline data is used to:

- Directly singulate the circuits from a panel by soft-tooling via numerically-controlled routing machines, cutting blades, lasers or water jet cutters.
- Create hard-tooling, including steel rule die punch tools and/or Class A tool-and-die punch set technology.

The datum features are important reference points to facilitate measurement of the circuit features, especially the outline. It is best if datum features are called out based on the part's internal physical features rather than its external features. This allows inspectors to base their measurements on real features rather than imaginary points outside the part. Multiple datum features are recommended if space allows.

Multiple datum features are also important because flexible materials are dimensionally less stable than rigid materials used for circuits. Features on flexible circuits, especially those of larger size, often move relative to each other and the circuit may shrink or grow during processing making predicting the precise location of features increasingly difficult over long distances. On a localized basis, and/or with smaller flexible circuits, the effects are not so great and the part can more easily meet requirements. This approach does not sacrifice any tolerances. It is simply a means of recognizing and accounting for flex circuit manufacturing dimensional variances. Figure 20 illustrates the benefits of multiple datums.

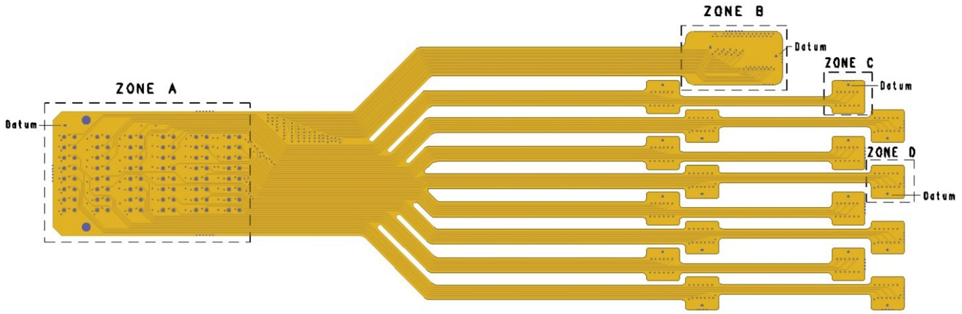


Figure 20: Flex circuit with multiple datums.

Marking Requirements

The increased use of automation in manufacturing minimizes the need for marking in many applications. However, some designs can still benefit from marking despite the added cost. When marking is used, the documentation package must accurately define where to locate markings on the circuit. The choice of ink type and color must also be defined.

Bend, Flex and Crease Locations

Flexible circuits are typically bent or flexed in some manner. Knowing where the bending is to occur is important, especially when the flex circuit is to be used in dynamic applications. In dynamic flex applications, the grain direction of the copper foil should be accounted for in manufacturing. It has been shown that there is a difference in the dynamic flex life of circuits fabricated with the bend going with the machine direction and those where bends are made transverse to machine direction.

Accurate bend or flex information also informs the engineer or technician where and in which direction to bend the circuit, resulting in easier installation. It is also beneficial to define the location of permanent bends and/or creases that may be needed, which can be accomplished in a couple of ways. If a marking step is required, special indicating features in the marking circuit artwork can be printed onto the coverlayer or while marking is applied.

This facilitates assembly by providing information as to which direction a bend must take. For example, dotted lines can be used to indicate bends in one direction and solid lines can indicate bends in the opposite direction. Similar features can also be made in the metal as part of the circuit. Because there are no common standards for this task, the fabricator's manufacturing engineers should be able to provide some guidance based on their experiences.

Stiffener Locations and Bonding Requirements

Stiffeners are commonly used to support components mounted onto flexible circuits. When they are required by a design the documentation package must include size, construction, location and the preferred material to be used to bond them to the flexible base material of the circuit. If any special strain relief techniques are required, such as an epoxy bead along flex-to-rigid transition, this should also be noted. The cross-sectional view should also show any strain relief as discussed earlier. See Figure 18.

Below is a diagram of a flex circuit with stiffeners. Stiffeners are used for assistance with connector insertion or component placement rigidity. The stiffener and coverlay termination points should overlap to avoid stress point helping to reduce the chance of traces breaking and cracking. See Figure 21.

Contact and Termination Finish Requirements

The finishing material on exposed flex circuit conductors can vary depending on their use and application. For example, gold is commonly used as a finish when wire bonding is to be used for direct chip attach, IC packaging, and when the exposed contacts are to be used for making separable connections to electrical connectors. More often, a solderable surface is all that is required and a number of options are available.

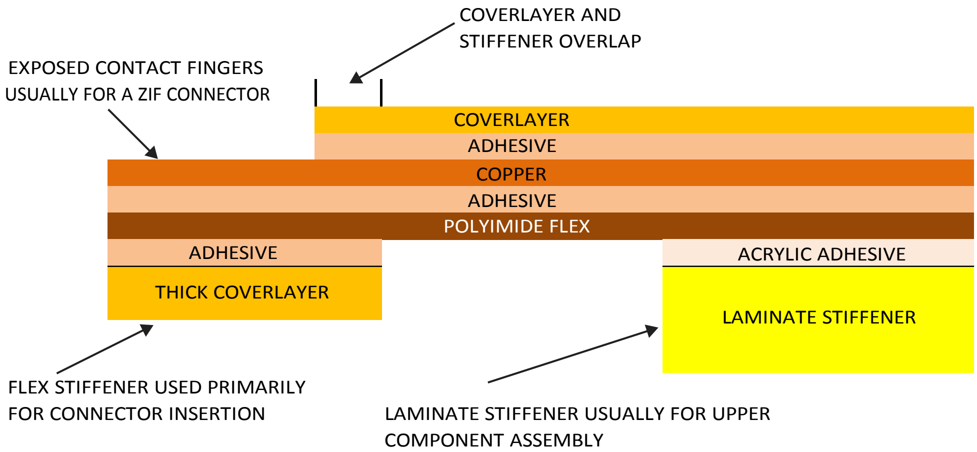


Figure 21: Different types of stiffeners.

Tolerances for Manufacturing

Manufacturing tolerances should be provided in the documentation package. Traditionally the manufacturing tolerances are covered in design documentation which provides a complete description of the finished product. Most documentation formats provide a tolerance block near the title block for the drawing. The tolerances called out in the tolerance block should accurately reflect the application of the finished flex product. Flex and rigid-flex designs are subject to meet criteria listed in IPC-6013, which is specific to flex circuits.

Special Electrical Testing Requirements

The electronics industry is continuously moving to ever-higher operating speeds and special tests are needed to ensure performance. Electrical testing requirements such as characteristic impedance validation, high-voltage potential withstanding or other non-standard tests should be identified and defined in the documentation. Conditions of testing and test point locations should also be clearly identified.

Special Processing Requirements

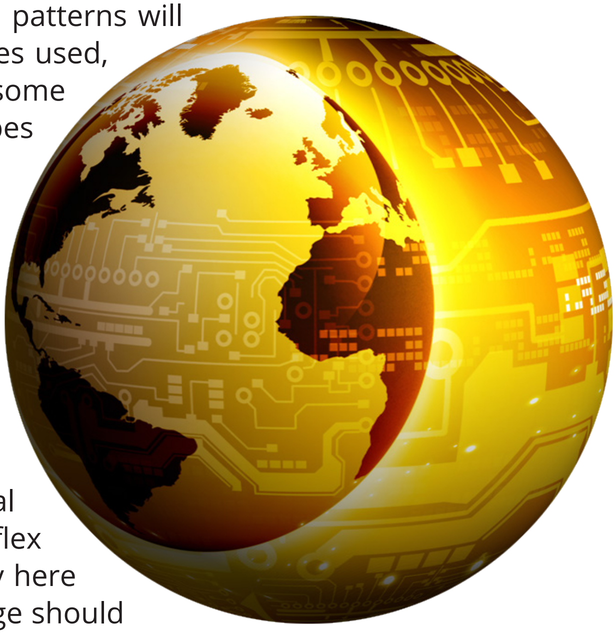
Special and/or non-standard processing requirements are occasionally required for flexible circuit designs and should be clearly outlined in the documentation.

List of Applicable Standards and Specifications for Reference

Standards and specifications are a unique type of glue that provide a consistency that holds the industry together. All relevant specifications that will be used to test, evaluate and accept or reject the product should be cited in the documentation package. For flex products, the acceptability criteria is listed in IPC-6013.

Bill of Materials for the Circuit

A bill of materials (BOM) relative to the final assembly should be included in the documentation package. The BOM may not be necessary for production, except by a turnkey manufacturer. Although the various hole and surface mount device (SMD) patterns will give clues to the component types used, they are not fully informative and some components or component types may require special attention.



This list of documented items is comprehensive, but not exhaustive. Since each design is unique, it is inevitable that there will be some information that varies depending on the product. Designers and product engineers will often place special requirements on individual flex circuit designs. The key takeaway here is that the documentation package should be as complete as possible to ensure, to the highest degree possible, that the product will be right the first time. Once again repeated here for emphasis, the documentation package provided to the fabricator is the primary means of communication and its clarity and completeness will prevent confusion, loss of irrecoverable time and unnecessary expense.

CONCLUSION

Flex and rigid-flex circuits have proven their utility and value in countless products since their introduction. It is likely that they will continue to be chosen and used to help facilitate and improve the ever-expanding range of electronic products in use around the globe today. However, as has been shown, there are many issues related to their design and manufacture that require proper consideration and expertise to fully address. We recommend that those about to embark on a flex circuit project seek expert consultation to ensure a first-pass success.

REFERENCES:

Sources & Additional Reading

Joseph Fjelstad, ***Flexible Circuit Technology***, p. 186, Fourth Edition, 2011 BR Publishing, Inc.

Robert Tarzwell, ***Flexible Printed Circuit Design & Manufacturing***, 2010, DMR Ltd.

IPC-2223, *Sectional Design Standard for Flexible/Rigid-Flexible Printed Boards*

IPC-6013, *Qualification and Performance Specification for Flexible Printed Boards*

APPENDIX A

Advantages of Flex and Rigid-Flex Circuits

Since their introduction, flexible and rigid-flex circuits have been steadily moving from the fringe of electronic interconnection towards its center. Today flex and rigid-flex circuits are found in countless products from the very simple to the highly complex. The reasons for this shift to the center are numerous and most of them are related to the advantages they offer. An examination of some of the benefits and advantages will make this clear.

They are a remedy to natural product packaging problems:

Flexible circuits are often chosen because they help to solve problems related to getting electronics inside the product they serve. They are a true three-dimensional solution that allows electronic components and functional/operation elements (i.e., switches, displays, connectors and the like) to be placed in optimal locations within the product assuring ease of use by the consumer. They can be folded and formed around edges to fit the space allowed without breaking the assembly into discrete pieces.

They help reduce assembly costs:

Prior to the broad use of flexible circuits, assemblies were commonly a collection of different circuits and connections. This situation resulted in the purchasing, kitting and assembly of many different parts. By using a flex circuit design, the number of part numbers required for making circuit related interconnections is reduced to one.

They eliminate potential for human error:

Because they are designed as an integrated circuit assembly with all interconnections controlled by the design artwork, the potential for human error in making interconnections is eliminated. This is especially true in the cases where discrete wires are used for interconnection.

They can reduce both weight and volume requirements for a product:

Flexible circuits are appreciably lighter than their rigid circuit counterparts. Depending on the components used and the exact structure of the assembly and final products, they can save perhaps as much as 60% of the weight and space for the end-product compared to a rigid circuit solution. Additionally, their lower profile can help a designer create a lower profile product than is possible with a nominal 1.5mm thick rigid board.

They facilitate dynamic flexing:

Nearly all flexible circuits are designed to be flexed or folded. In some unusual cases, even very thin rigid circuits have been able to serve to a limited degree. However, in the case where dynamic flexing of a circuit is required to meet the objectives of the design, flexible circuits have proven best. Modern disc drives, for example, require that the flexible circuit endure anywhere from tens to hundreds of millions of flexural cycles over the life of the product. Other products (laptop hinge circuits, for example) may only require thousands of cycles, but it is the dynamic actuation capability enabled by the flex circuit that is key to its operation.

They improve thermal management and are well suited to high temp applications:

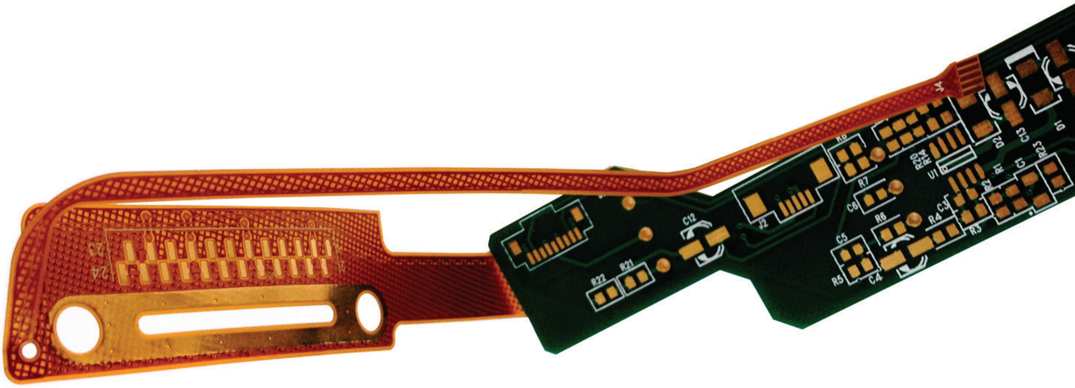
High temperatures are experienced both in assembly with lead-free solder and in the operation of higher power and higher frequency digital circuits. Polyimide materials are well suited to the management of high-heat applications. Not only can they handle the heat, their thinness allows them to dissipate heat better than other thicker and less thermally conductive dielectrics.

They help improve product aesthetics:

While aesthetics may seem a low order advantage, people are commonly influenced by visual impressions and frequently make judgements based on those impressions. Flexible circuit materials and structures look impressive both to the seasoned engineer and the layperson. It can make a difference in the decisions made in some applications, especially those where the user gets exposure to the functional elements of the product.

They are intrinsically more reliable:
Flexible circuits help to reduce the complexity of the assembly and can reduce the number of interconnections that might be otherwise required by the use of solder. Reductions in complexity is a key objective of a reliable design. With respect to the minimization of the number of solder interconnections, reliability engineers know all too well that the vast majority of failures in electronic systems occur at solder interconnections. It follows naturally that a reduction in the number of opportunities for failure should result in a corresponding increase in product reliability.

In summary, flex and rigid-flex circuits have significant advantages. There are many additional advantages which go beyond the short list provided here. What is important to remember is that most of the advantages stem from the versatility and unique integrative abilities these important members of the electronic interconnection family can offer.



APPENDIX B

SAMPLE NOTE FOR A 4-LAYER RIGID-FLEX PCB

1. RIGID-FLEX TO BE FABRICATED USING IPC-6013, CLASS 2 STANDARDS.
2.THIS FLEX CIRCUIT TO CONTAIN 4 LAYERS IN RIGID SECTIONS AND 2 LAYERS IN FLEXIBLE SECTION.
3. MATERIALS:

  a. THE RIGID MATERIAL SHALL BE EPOXY GLASS LAMINATE PER IPC-4101 / 24 / 26 / 99 / 101 / 126
  b. THE FLEX MATERIAL SHALL BE ADHESIVELESS FLEXIBLE COPPER CLAD LAMINATE
  c. COVERLAYER TO BE .001" POLYIMIDE WITH .001" ADHESIVE

4. COPPER STARTING WEIGHT TO BE 1/2 OZ. ON ALL LAYERS WITH AN ADDITIONAL PLATING OF .001" MIN. OF COPPER ON LAYER 1 AND LAYER 4.
5. THIS FLEX CIRCUIT SHOULD BE OF MULTIPLE BEND TYPE.
6. MINIMUM BEND RADII TO BE 6X THICKNESS OF FLEX CIRCUIT.
7. MINIMUM LINE WIDTH TO BE 3 MILS AND MINIMUM LINE SPACE TO BE 5 MILS.
8. MINIMUM ANNULAR RING REQUIREMENTS IN ACCORDANCE WITH IPC-6012, CLASS 2, TANGENCY WITH NO BREAKOUT.
9. UNLESS OTHERWISE SPECIFIED HOLE TOLERANCES ARE +/-.003.
10. FINISH: AFTER COPPER PLATING PLATE ENIG, PER IPC-4552.
11. SOLDER MASK RIGID SECTION BOTH SIDES LPISM GREEN (SOLDER MASK OVER BARE COPPER)
12. RoHS MATERIALS REQUIRED.
13. OVERALL THICKNESS IN FLEX AREA SHALL BE NO MORE THAN 9.0 MILS.
14. SILKSCREEN COLOR WHITE ON TOP SIDE OF BOARD.
15. ALL BOARD DIMENSIONS SPECIFIED BY DWG IN ATTACHED FILE TEST.PDF. (DIMENSIONS IN GERBERS FOR REFERENCE ONLY.)
16. FOR ANY DIMENSIONS NOT IN DWG TEST.PDF USE GERBER DATA.
17. VENDOR TO PRIMARY DRILL ALL HOLES (NON-PLATED HOLES SHALL BE TENTED.)
18. MAXIMUM OF 1 X-OUTS ALLOWED IN ARRAY.

ABOUT AMERICAN STANDARD CIRCUITS

Founded in 1988, American Standard Circuits is a leading manufacturer of advanced circuit board solutions in the U.S.A. As a total solutions provider for the worldwide PCB industry, we are capable of delivering advanced technology to virtually every industry sector in quantities ranging from quick prototypes to large-volume production.

There isn't anything we won't do for our customers.

Our goal is to provide our customers with the best quality and value for all their PCB requirements.

First and foremost, American Standard wrote this book for our customers. Our intention in providing this guidebook to our customers is to make sure that they know everything possible to design the very best, most efficient and economical flex and rigid-flex printed circuit boards possible.

We believe that we are not only a flex and rigid-flex PCB fabricator, but that we are our customers' own personal PCB experts. It is our role to look after all their PCB needs to ensure they are getting the best products on the market today. Our experts are your experts and they are ready to help. You can access them at www.asc-i.com/about-us/experts.

Our ongoing commitment to leading-edge, high-level interconnect technology, cost-effective manufacturing and unparalleled customer service has put us at the forefront of advanced technology circuit board fabrication.

We manufacture quality rigid, metal-backed, flex and rigid-flex printed circuit boards on numerous substrates for a variety of applications, including:

- Military/aerospace
- Industrial
- Commercial
- Medical
- Telecommunications
- Consumer electronics
- RF/microwave
- Transportation

Advanced Technology

Our ongoing R&D work in both thermal management and embedded passive components allows us to keep up with ever-increasing complexity, various clock speeds, power consumption, and heat output computing devices. Our specialties include hybrid circuits (made from multiple materials), exotic materials, blind and buried vias, controlled impedance, and thermal management solutions, for which we hold several patents.

Quality

ASC has state-of-the-art process control systems and our certifications include AS9100 Rev. C, ISO 9001:2008 and MIL PRF 31032. We have rigorous SPC controls, automated optical inspection, metallographic cross-section and electrical test facilities. These provide the quality assurance needed to meet the high standards of our customers.

Quick Turns

We have developed the ability to provide quick turns for all of the technologies that we manufacture. Our world-class front-end engineering CAM systems and processes, coupled with our rapid-response manufacturing processes, enable us to provide quick-turn options for our customers.

For more information, visit www.asc-i.com

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

GLOSSARY

Adhesive – Modified acrylic or epoxy that is used to coat polyimide film.

Adhesiveless – Typically used for a flex laminate (copper-polyimide material) with no adhesive. The two options are that copper is sputtered on the polyimide dielectric and then plated up to the desired thickness or alternatively cast solvent-laden polyimide directly onto copper and then cured to polyimide to drive out the solvent.

Base Material – Insulating material upon which a conductive pattern may be formed.

Bend Radius – The minimum radius one can bend a flex circuit without damaging it, or shortening its life. The smaller the bend radius, the greater the material flexibility.

Bond Ply – (AKA: Bondply or Bond-Ply) Polyimide film coated on both sides with a B-staged adhesive. Bond ply is used to laminate two pieces of base material together after etching.

Button Plating – A method of plating flex circuits so that only the circuit's through-holes and surrounding pads are plated. This greatly adds to a circuit's flexibility.

Coverlayer – Polyimide film with B-staged adhesive used to cover and protect the copper traces of the flex circuit.

Dielectric Constant (Dk) – Dk is the measure of the capacitance or energy between a pair of conductors in the vicinity of the laminate compared to that pair of conductors in a vacuum. The Dk for a vacuum is 1.0, flex is 2.7-3.0, and FR-4 is 4.3.

Dielectric Material – A material with a high resistance to the flow of current.

Dynamic Flex – A flex circuit application that requires repeated flexing while in use.

ED Copper – Electro deposited copper is less flexible than RA copper and is typically used to manufacture rigid boards.

Flame Retardant Material – A material that is capable of achieving a UL rating of 94-V0.

Flex Circuits – Circuits that are manufactured from materials that can bend, fold and twist.

Flexible Solder Mask – Type of solder mask when cured over flexible circuits that will not crack, separate or delaminate from the surface of the base material, conductors and lands of the coated flexible circuit.

Grain Direction – In rolled annealed copper, the grain direction results from the rolling and annealing process. The raw laminate supplier always sells the raw material with the grain direction indicated.

Impedance – The measure of the opposition that a circuit presents to a current when voltage is applied.

Laminates – The plastic material, usually reinforced by glass or paper, that supports the copper cladding from which circuit traces are created.

No-flow Prepreg – Prepregs that are uniquely formulated to have limited and controlled flow.

Polyimide – A polyimide is a polymer of imide monomers. With their high heat resistance, polyimides enjoy diverse applications in roles demanding rugged organic materials, e.g., high-temperature fuel cells, displays, and various military roles.

Prepreg – Glass cloth which has been pre-impregnated with a thermosetting resin used for making rigid circuits.

RA Copper – Rolled annealed copper is much more flexible than ED copper and is required for dynamic flexing applications.

Registration – The degree of conformity of the true position of a pattern with its intended position or with that of any other conductor layer of a board.

Rigid-Flex Circuits – A printed circuit board with both rigid and flex materials.

Signal Integrity – A set of measures of the quality of an electrical signal.

Solder Mask – A thin lacquer-like layer of polymer that is usually applied to the copper traces of a printed circuit board for protection against oxidation and to prevent solder bridges from forming between closely spaced solder pads.

Stiffener – Material that is used to reinforce flex circuits when and where required. Most commonly, stiffeners are under areas where electronic components are to be attached. Stiffeners can also help increase wear resistance.

Strain Relief – Rigid to flex Interfaces are critical to the successful manufacture of rigid-flex product. This interface area is also subject to movement of materials as the edges are defined “in process” and not by a router bit at final outline. These interfaces can be reinforced with a bead of epoxy material (or several similar products). This epoxy material is referred to as “fillet material” or “strain relief” material.

Stripline Circuit – A stripline circuit has a trace layer which is sandwiched between two parallel ground planes with a dielectric in between.

Thermosetting Resin – Thermosetting resin is a material that hardens when heated and cannot be remolded, softened or reshaped by subsequent heating.