CHAPTER
5

Practical Design Guidelines for Flex

Introduction

Flexible circuits are obviously unique among electronic packaging technologies in that they offer a wide variety of advantages unobtainable using conventional rigid interconnection technologies. Freeform integration of electronic elements through all three dimensions of space is highly liberating to the design process.

Such advantages, however, cannot be garnered without a thorough understanding of basic flex circuit design principles. Proper use of those design principles can provide a path to early success. In contrast, failure to use good design practices can result in early failure. The objective of this chapter is to provide information vital to the successful production of flexible circuit designs—ones that will consistently perform to user expectations.

Design Preliminaries

Before embarking on a flex circuit design, it is important that a holistic overview of the project be taken. In this overview, a circuit designer should attempt to take into account as many of the items discussed in the implementation section as are possible or relevant. This act of taking stock of the project will help the designer appreciate the broader perspective of the task, minimizing the possibility that a gross and avoidable error will be carried through the design process.

It is also important that the designer keep in mind that flexible circuit designs require a balancing of both mechanical and electrical concerns. These two competing concerns, the designer will find, often oppose each other's requirements in a design. It will be the holistic approach that will help the designer thread the needle to make the best possible choice from given alternatives.

Use of Mockups

Let us reassert here the value of using paper doll mockups. This simple practice will help the designer prevent many errors by exposing potential problems early and will save both time and money.

Some modern CAD systems have demonstrated the ability to execute three dimensional layouts required for flex circuit applications, but the physical model will probably always prove of some value in addressing both the ergonomic elements of assembly and the concerns of access should field repair be required.

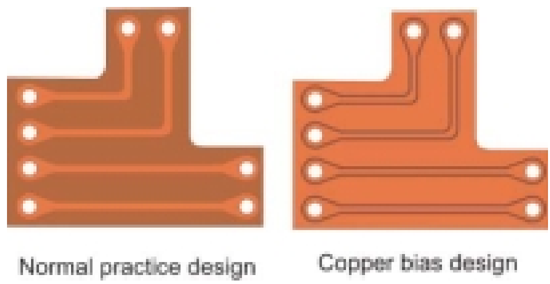

Figure 5-1 Designing with a bias for copper improves the circuit's dimensional stability. It may not be practical for all applications.

DESIGN WITH A BIAS FOR COPPER

Favoring the use of copper in design is good practice for some very solid reasons, assuming that there are not important conflicts created by the practice. If all other design objectives are met, then the primary reason for maintaining extra copper is that it helps to enhance the dimensional stability of the circuit. Designing with a bias for copper is a practice especially well suited to single-layer flex circuit designs. See Figure 5-1.

As indicated before, the decision to add or leave copper should be made in light of the objectives of the circuit's final use. For example, if a reduction in weight of the final product is a key objective, then there would be need to trade away some of the enhanced dimensional stability. Another reason for maintaining the copper is that it reduces the amount of copper etched and is thus more environmentally friendly in terms of chemical usage.

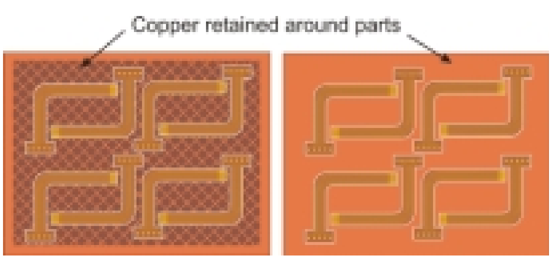

Figure 5-2 Copper foil is maintained by design around circuits to provide better panel dimensional stability. Two examples are shown.

TOLERANCE SETTING FLEX CIRCUIT DESIGNS

Proper application of tolerances of flex circuit design features is a matter that concerns both the flex circuit manufacturer and the flex circuit user.

In general, it is recommended that the largest practical tolerance be given to all features and locations to facilitate manufacture. This is because the base materials are flexible and prone to distortion, making accurate measurement over distances difficult. To compensate, it is recommended that more than one datum be used on larger circuits. Individual datums can be provided locally relative to features deemed important. (See Figure 5-3.) This will result in a more accurate measurement being taken and can preempt potential conflicts in measurement results between inspectors. To avoid confusion in design, one datum should be defined as the primary or master datum and others as secondary or slave datums.

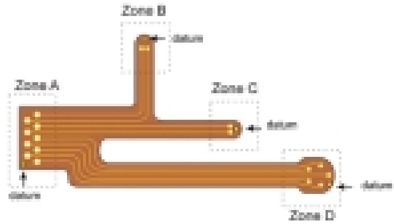

Figure 5-3 The use of multiple datums, with one being the prime and the others secondary, facilitates both accurate measurement of the circuit and device placement during the assembly process.

Tight tolerances can be attained, when required, but to do so requires

General Guidelines for Circuit Feature Sizes and Tolerances

Design Feature	Standard Product	Reduced Producibility	Normal Tolerance	Minimum Tolerance
Hole size	0.020" (0.5mm)	< 0.020" (0.5mm)	±0.003" (0.075mm)	±0.001" (0.025mm)
Trace width	0.010" (0.25mm)	<0.010" (0.25mm)	±0.002" (0.050mm)	±0.001" (0.025mm)
Space width	0.005" (0.125mm)	0.005" (0.125mm)	±0.002" (0.050mm)	±0.001" (0.025mm)
Trace to edge of part				
Steel rule tool	0.020" (0.5mm)	<0.010" (0.25mm)	±0.010" (0.25mm)	±0.005" (0.125mm)
Class A die	0.005" (0.125mm)	<0.002" (0.050mm)	±0.005" (0.125mm)	±0.002" (0.050mm)

Feature to Feature Location (relative true position)	12.0" (≤300mm)	≤ 18.0" (≤450mm)	≤ 24.0" (≤ 600mm)
Preferred (per MIL-STD-2118)	0.028 (0.7mm)	0.034 (0.85mm)	—*
Level A (per IPC-D-249)	0.034 (0.85mm)	0.040 (1.0mm)	0.046 (1.15mm)
Standard (per MIL-STD-2118)	0.020 (0.5mm)	0.024 (0.60mm)	—*
Level B (per IPC-D-249)	0.022 (0.55mm)	0.024 (0.60mm)	0.034 (0.85mm)
Reduced Producibility (per MIL-STD-2118)	0.012 (0.30mm)	0.016 (0.40mm)	—*
Level C (per IPC-D-249)	0.012 (0.30mm)	0.018 (0.45mm)	0.022 (0.55mm)

*Mil-Std-2118 off ers the following statement regarding tolerances: "Drawing tolerances must reflect bend and fold allowances between component mounting rigid areas."

Table 5-1 Tolerance guidelines for standard flex circuit manufacture. (Chart does not apply to leading edge products.)

special attention and good techniques. As a result, the expense of tighter tolerance circuits tends to be greater due to anticipated loss of yield. Table 5-1 provides some general guidelines for tolerancing based on different design standards.

The values offered in Table 5-1 are rather generous by today's standards, however. To provide a global perspective on flex circuit feature capability, a 2004 survey of 20 Japanese flexible circuit manufacturers indicated that more than half of the companies were routinely producing flex circuits with traces of 110μm or less, and five companies were producing circuits with features of 70-90μm routinely. Today are a number of flex circuit manufacturers in Japan and elsewhere are producing circuits having features of 35 to 50μm and some have shown capabilities down to 25μm and even 10μm feature sizes.

General Guidelines for Dimensioning and Tolerancing

Proper dimensioning and tolerancing of flex circuits is vital to achieving good manufacturing yield. While it is not possible to point out every possible situation where dimensions and tolerances can be used in such a way as to confuse the interpretation of a drawing, there are certain general guidelines that, if followed, can do much to minimize the potential for confusion. Following are a few such guidelines:

- Show sufficient dimensions so that the intended sizes and shapes can be determined without requiring the distances between features to be calculated (or assumed).
- Provide individual dimensions only once and check them.
- State all dimensions clearly so they can only have a single possible interpretation.
- Show the dimensions between points, lines or surfaces, which have a necessary and specific relation to each other or which control the location of other components or mating parts.
- Check dimensions to avoid accumulations of tolerances that may permit alternative interpretations.
- Provide dimensions to features, which are shown in profile making certain that the feature's dimensions are not ambiguous.
- Do not show dimensions to lines representing hidden surfaces.
- Do not use "off part" datums.

Special Design Considerations

There are some unique elements of flex circuit design that require early consideration. Mostly they address mechanical issues that could affect usability and/or long term performance. However, they will definitely affect circuit layout and so are given early consideration.

Lay Out Circuit to Conserve Material

Conservation of material in flexible circuit manufacture serves to help keep manufacturing costs down. This is important because flexible circuit materials tend to be expensive in comparison with standard rigid materials such as FR-4.

We suggest accomplishing close spacing of the circuit. The technique of optimizing the number of circuits per panel is called ***nesting***. The term ***optimizing*** is used in place of the seemingly more logical term ***maximizing*** for a reason, that reason being that the layout of a flex circuit should be based on end use, and some uses may demand that portions of the flex circuit be properly oriented relative to the grain direction of the foil (such as is required for dynamic flexing). This may result in a less-than-maximum material use for circuit construction. However, when this is not the case, there is the opportunity to lay the circuit out in various ways to get the most out of the material.

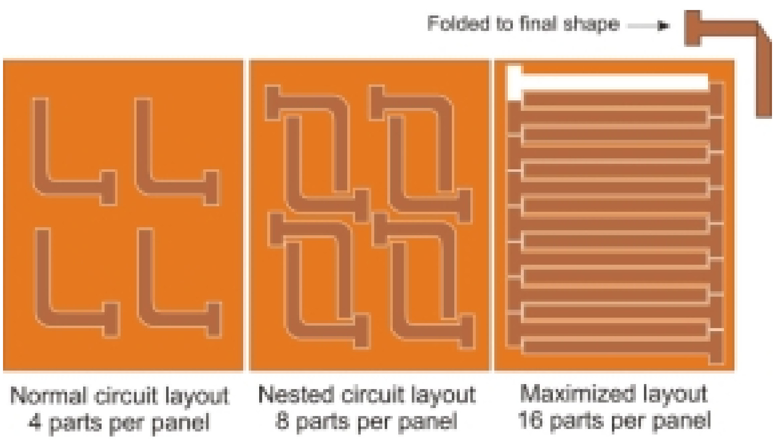

Figure 5-4 Proper circuit nesting can greatly improve panel yield and lower overall cost. If folding can be tolerated as an assembly operation, yield can be maximized. For dynamic flex circuit designs, the grain direction requirement may impact layout.

While nesting is routinely performed by the manufacturer, the designer can aid in this process by taking advantage of the fact that flex circuits can be bent and folded. Thus, adding a small length to a circuit arm can allow a circuit to be produced more economically, provided the user doesn't mind adding a folding operation to his assembly process. (See Figure 5-4.)

Service Loops

The addition of a small amount of length to the flex circuit beyond the design requirement is advisable for most flex circuit applications. This little extra length of material is commonly referred to as the service loop length.

The purpose of the service loop is offer sufficient length to facilitate both assembly of the product and servicing of the product once in the field, if it should ever be required. The extra length also helps to compensate for small, unforeseen variations in both the package and the flex circuit.

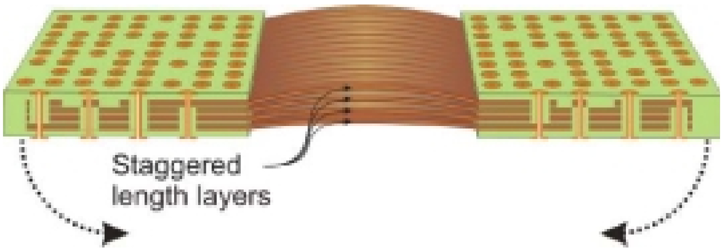

Figure 5-5 Staggered length designs facilitate bending of the flex circuit; the circuit can only be bent in one direction by design.

Staggered Length Circuits (Bookbinder Construction)

For ease of flexing multilayer and rigid flex designs, the use of staggered length design is commonly employed. The technique is accomplished by adding slightly to the length of each succeeding flex layer, moving away from the bend radius. (See Figure 5-5.)

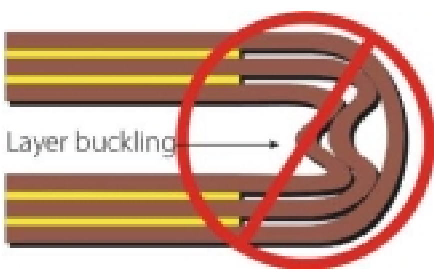

Figure 5-6 Without staggered lengths, layer buckling occurs.

A common rule of thumb is to add length equal to roughly 1.5 times the individual layer thickness. This helps defeat whatever tensor strain might have otherwise been built up in the outer metal layers of the multilayer flex and prevents buckling of the center of bend layers (see Figure 5-6).

Conductor Sizing and Routing

In general, flex circuit conductor width and thickness are determined by a combination of current carrying requirements, the voltage drop allowance and/or characteristic impedance control needs. When designing flex circuits for dynamic applications, the use of the thinnest possible copper is recommended. Thus, it is important that the designer opt for wider rather than thicker traces to accommodate basic electrical needs or requirements. This practice assures maximum circuit flexibility.

Table 5-2 can be used to determine maximum current and line resistance for given trace widths with both 35μm (1-oz) and 70μm (2-oz) copper. These are relatively common foil thicknesses used in much flex circuit manufacture, although 18µm (½-oz) and even lower copper foil thicknesses are becoming increasingly important.

A number of different nomographs for determining other electrical values for copper have been developed to simplify copper trace-requirement specification. The IPC's flex circuit design specification is a good source for such nomographs for those who have interest. There is an effort underway to revise these graphs, which have long been in use, to reflect more practical values.

Conductor Width	Maximum Current for 10° C rise 1 oz copper	Conductor Resistance milliohms/ft 1 oz copper	Maximum Current for 10° C rise 2 oz copper	Conductor Rresistance milliohms/ft 2 oz copper
0.005	.25	1280	NA	NA
0.010	.6	640	1.0	320
0.015	1.1	400	1.8	200
0.020	1.3	320	2.0	160
0.025	1.5	250	2.5	125
0.030	1.8	200	2.9	100
0.050	2.5	120	4.0	60
0.070	3.2	90	5.0	45
0.100	4.0	60	6.9	30
0.150	5.9	40	9.8	20
0.200	6.9	30	12.0	15
0.250	8.6	25	13.5	12.5

Table 5-2 Current Carrying Capacity of Conductors. Conductor width and copper thickness have a direct impact on the current carrying capacity of a flexible circuit. The above table provides a means of determining the conductor width for a 10° C rise

Trace Width Minimums

The minimum practical trace width for a flex circuit varies from vendor to vendor. Flex circuits with traces 250µm (0.010") and greater are fairly easy to obtain; however, line widths 125µm (0.005") and lower are increasingly common. Flex circuits having features in the range of 50µm (0.002") and lower are available in volume production from a limited number of vendors, but the number of such vendors is growing to keep pace with the demand for ever smaller electronic products.

The type of technology used in circuit feature manufacture also heavily influences trace width minimums. For example, plated up copper sputtered polyimide base circuits are basically limited in feature size only by the photolithographic capabilities of the manufacturer. Thus, very small circuit features can be made. For etched circuit traces, however, the trace width and pitch are influenced primarily by the thickness of the base copper foil.

Typically, the trace pitch limit is nearly linear with copper thickness within a narrow range. 18µm (½-oz) copper will yield circuit features at a 125µm (0.005") pitch, while with 35µm (1-oz) copper, etching becomes more difficult under 175µm (0.007") pitch. While some manufacturers can successfully produce 25µm (0.001") features with 18µm copper, vendor capabilities vary widely. It is best to check with the fabricator before attempting to design very fine line features.

Controlled Impedance Lines

Controlled impedance transmission cabling is a popular application for flex circuits, and the value of such product is increasing as digital data signaling speeds continue to rise.

Tighter tolerances for etched features are possible with flex circuits because of the lower profile adhesion treatment or "tooth" of the copper. The use of thicker flexible dielectric substrates, if the design allows, can ease somewhat the etching challenge because thicker substrates allow for wider signal lines, which can be fabricated more easily to meet the tight tolerances needed for controlled impedance circuits. This topic will be discussed in more detail later in this chapter.

Etch Factors

An etch factor is a tool used by the manufacturer to compensate for isotropic etching process effects. It is recommended that the designer check with the vendor to determine if they want inclusion of an etch factor. Usually it is best if the manufacturer makes this adjustment, as they will be most familiar with their process and its capability.

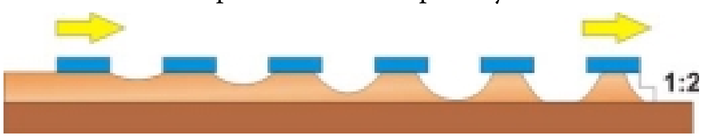

Figure 5-7 The etching process works laterally as well as down, at a ratio of roughly 1:2 laterally to down

The typical line width loss (measured at the top of the trace) due to the etching process is approximately 2x copper foil thickness, although copper type, conductor pitch, etch mask, process chemistry and equipment can all influence the results.

Conductor Routing Concerns

There are a few general issues related to conductor routing of a flex circuit. The first item of concern is keeping to a minimum the number of crossovers in the layout. This will help to keep the layer count down and lower the cost. Newer CAD systems can respond to such a requirement, but the results may need to be massaged or optimized to make certain that the smallest possible area has been consumed in the process.

Routing of conductors on a flexible circuit perpendicular to bend and fold is the recommended design practice. The purpose is to facilitate the bending or folding process and to minimize stress through the area. In addition, circuitry should be routed on a single copper layer through bend and fold areas whenever possible.

Figure 5-8 Routing options for flex circuit trace corners. Avoid sharp corners if possible. A radius is best as it provides a smooth transition and mitigates potential issues related to stress risers.

It is also recommended that designs avoid having right or acute angles (≤ 90°) in circuit routing. This is because they tend to trap solution and may over etch in process. They are also more difficult to clean after processing, so best practice dictates that corners should be provided with a radius if possible. The radius also improves signal propagation, as the reflections at turns are reduced.

With double-sided flex, when and where the conductors must be routed through bend and fold areas and when copper traces are on both sides, the circuit designer should design spaces to be approximately 2-2.5x the trace width.

Preferably, the designer should also stagger traces from side to side. The purpose of this practice is to avoid the I-beam effect. This can be a critical concern in dynamic applications. (See Figure 5-9.)

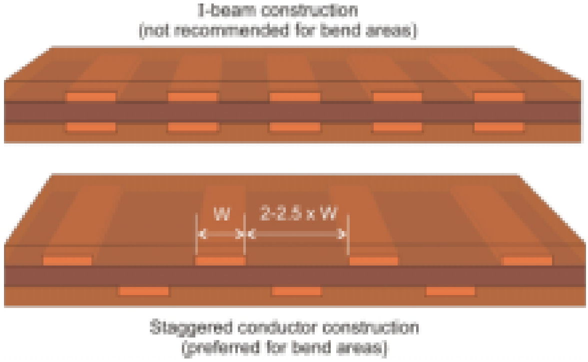

Figure 5-9 I-Beamed vs. Staggered Conductors. I-beamed constructions increase the stiffness of the circuit through bend and fold areas. A better alternative, if space allows, is to stagger conductors for improved flexibility.

Finally, placement of vias within the bend area is highly discouraged as they will adversely affect bend formation and create unwanted points of stress and potential crack propagation.

Ground Plane Design

Ground areas should be crosshatched if electrical consideration of the design will allow for such. The practice helps both to reduce weight and improve circuit flexibility. The size of the openings in the ground plane may be critical depending on the end product requirements for shielding or controlling of characteristic impedance. If openings are too large, some shielding effects may be lost, depending on frequency. Also, ground connections for components should be thermally relieved to reduce heat sinking and assure formation of a good solder joint. This is accomplished by etching a clearance area around the pad while maintaining electrical connection. Figure 5-10 illustrates the technique.

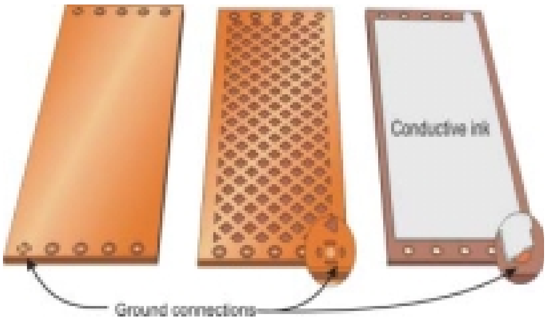

Figure 5-10 Ground planes should be crosshatched if possible to improve flexibility. Clearance holes prevent shorting. Ground connections should be relieved to reduce heat sinking when soldering.

Polymer Thick Film Design Guidelines

Due to their unique nature, polymer thick film (PTF) circuits have their own very specific design rules. As a screen printing based technology, the limits of design are tied to two main factors: (1) the conductivity of the ink chosen and the limits of the screen-printing materials, and (2) the processes used. Much of the latter factor is tied to the former. That is, the particle size of the included conductor material and the polymer carrier will help establish the limits of screen printing. Emerging nanoparticle technologies could boost conductivity significantly, possibly opening the door for broader use of polymer thick film technology. While PTF circuits are not generally considered for dynamic applications, they can actually perform quite well in certain dynamic applications. Some experimenters have reported increases in conductivity with cycling. PTF membrane switches also stand as witnesses to the efficacy of PTF as a flexing technology.

Conductor Width and Spacing for PTF

Generally, minimum conductor width and spacing is considered to be in the range of 375µm (0.015"). It is possible to produce finer lines and spaces using PTF inks, but conductivity can become more of a design performance concern.

Current Carrying Capacity of PTF

Silver-based polymer thick film inks, under normal conditions, can be expected to carry approximately 25% of the current of copper circuits for equivalent line widths and nominal PTF ink thickness. Care should be used, however, in attempting to maximize conductor current-carrying capability under this premise. Hot spots within the conductor matrix can cause rapid degradation of the conductor and possible failure.

Screen-Printed PTF Resistors

Screen-printed resistors are fairly commonly incorporated into PTF circuit designs. If used in a design, the resistors should be kept to a minimum of one or two values to facilitate processing. Generally, the resistors can be printed to ± 20% of value without trimming. Laser or mechanical trimming of the resistor can be used if tighter tolerances are required.

Termination Design Concerns for PTF Circuits

The design rules for circuit pads or lands for PTF circuits are similar to those used for rigid printed wiring boards; however, the termination features should be discussed with the manufacturer. While polymer thick film inks are not directly solderable, conductive adhesives can be used to surface mount components. Again, land design for surface mounting is similar to PCBs.

Interconnection Design Features

This chapter section deals with interconnection design features, including both through holes and lands for making interconnections and the design criteria for making those access points more reliable.

Hole Sizes for Component Leads

While surface mount technology has become the dominant interconnection technology for electronic component assembly, through hole components are still used in many applications. As a result, proper sizing of the hole remains an important design checkpoint.

Finished hole diameter for through hole mounted components in flex circuits for most applications should be nominally 200-250μm (0.008-0.010") larger than component lead to meet best practice design requirements for automated component placement. However, this is not always possible or practical. One key advantage of flex circuits is that, because of the thinness of the circuit, smaller gaps between the component and the through hole can be reliably soldered—but the devices are more difficult to insert.

Best or preferred case flex design practice suggests that all lands or pads should be made 2-2.5x the hole diameter. Holding this value is primarily a concern with single-sided flex, where maximum solderable area is sought to ensure that a reliable connection can be made.

Again, as with drilled through holes, this ratio will not always be practical, as is the case with miniature connectors. In those cases where very small lands are mandated and pin in hole assembly is required, a plated through hole may be required to enhance solder joint reliability.

Figure 5-11 **Through Hole Land or Pad Termination Sizing.** Maintaining a proper drilled hole to pad relationship is most important with single layer flex designs. Plated through holes can get by with smaller lands.

Via Hole Sizing

Vias can be designed as small as is practical for the manufacturer's yield. Small vias offer great advantage for circuit layout, but circuit cost may be affected if they are designed too small, depending on what technologies are available for making holes in the base material. Current generation punching and laser techniques are capable of economically mass producing interconnection vias as small as 25-50μm (0.001-0.002"). In contrast, drilling, because of the higher cost of small drills, becomes more expensive as the holes get smaller. Because flexible circuit base materials are thin, it is fairly easy to plate small through holes reliably. The small plated holes are also highly reliable in flex circuits. This is due in large part to the thinness of the base material, which results in a total material expansion that is low and less of a concern with respect to thermal cycling.

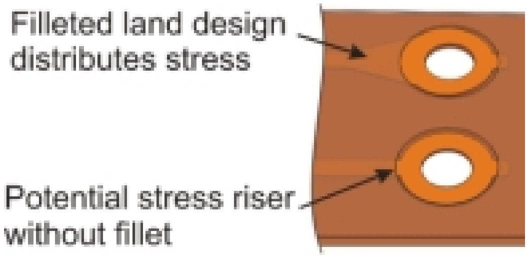

Figure 5-12 The practice of filleting pads helps to improve the reliability of the circuit by more evenly distributing stresses at the junction of the circuit land to the coverlayer opening.

Filleting of Lands and Pads

Termination lands and pads on flexible circuits should be filleted. This process increases pad area and helps to distribute stresses local to the coverlayer openings better, effectively relieving a stress riser condition that commonly causes failure if the fillet is not supplied or ignored. Earlier CAD systems had difficulty in producing these features, but today's more advanced systems can more reliably address the requirement for fillets without difficulty. See Figure 5-12.

Pad or Land Hold downs for Single-Sided Flex

Termination pads on single conductor layer circuits and surface mount lands on flex circuits of any layer count may require special land hold down techniques. With single-sided flex circuits, the use of special features variously referred to as tie down tabs, anchoring spurs, or rabbit ears may be employed to prevent the land from lifting during soldering processes in cases where excessive heat is used. With new lead-free solders, this may become more important.

An important note on this subject is that features such as tie down tabs could well cause problems as the industry moves to higher data rate signaling, and they should be used with caution. The stubs associated with some tie down features are capable of acting like antenna and can broadcast noise within the package when higher frequencies are used. Thus, an evaluation of the approach may be warranted, depending on the nature of the design. Figure 5-13 shows typical hold down tab features and alternative designs.

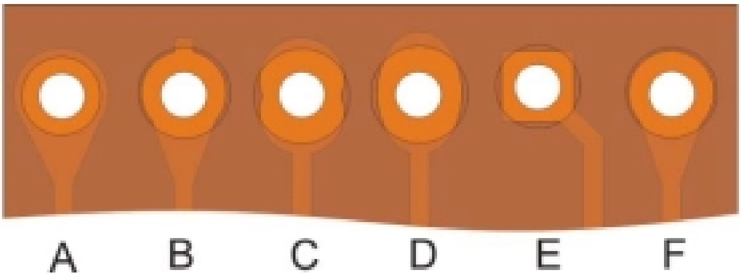

Figure 5-13 Various pad designs to help facilitate their capture by the coverlayer. (A) Standard filleted pad with full pad capture (B) Standard filleted pad with hold down tab (C) Overlapping pad design (D) Oval pad design (E) Corner entry to square pad (F) Plated through holes normally require only filleting.

Surface Mounting Lands for Flex

Surface mount in combination with flex circuit technology is now very popular as the world's flex circuit designers look to the success of Japanese products, which often employ flex circuits with surface mounted components. Surface mounting lands, however, often require a slight modification of standard design rules when applied to flex circuit applications.

The use of holes or slots drilled or routed into the coverlayer before lamination is a common way for flex circuit manufacturers to access solder lands. However, if traces are routed straight into the land, misregistration of the coverlayer could result in the creation of a stress riser, as shown in Figure 5-12. The same concerns regarding through hole components hold true for surface mount land features. In Figure 5-14 (A), the potential stress riser condition is again shown. Side or corner entry to the land is more tolerant to misregistration (Fig 5-14 [B]). Laser-cut or mechanically punched or routed coverlayer openings can be made rectangular (Fig 5-14 [C]).

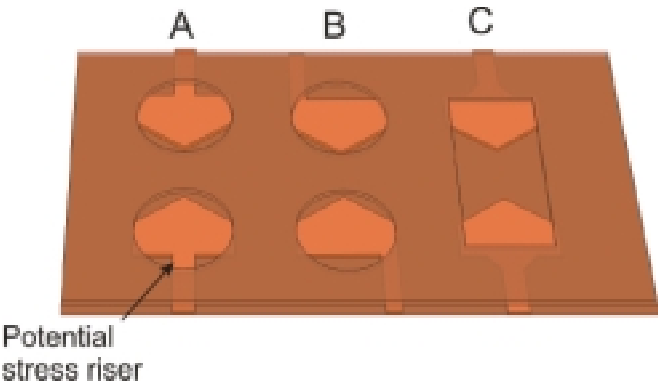

Figure 5-14 Coverlayer openings for discrete SMT components create special design concerns.

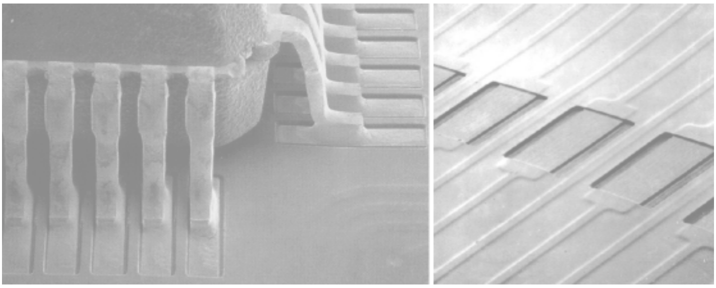

Figure 5-15 Examples of precisely photoimaged cover films for surface mount lands on flexible circuits (Photo courtesy DuPont)

Rectangular openings can also be achieved by using photoimageable cover films in place of a coverlayer. When accessing component lands for device assembly, it is recommended that solder lands extend beneath the coverlayer, as shown in Figure 5-15.

SMT device lands for both discrete and leaded devices should be extended to allow capture by the coverlayer. Normally, lands should be 250-375µm (0.010 to 0.015") larger to facilitate land capture and prevent undesired lifting of the land during assembly or repair.

As with single-sided through hole lands for flex circuits, the objective is to prevent land lift during soldering operations and to provide extra strength against component pull-away in operation. This is of greater importance with components of greater mass. Figures 5-15 and 5-16 provide examples of common approaches used to access surface mount features in flexible

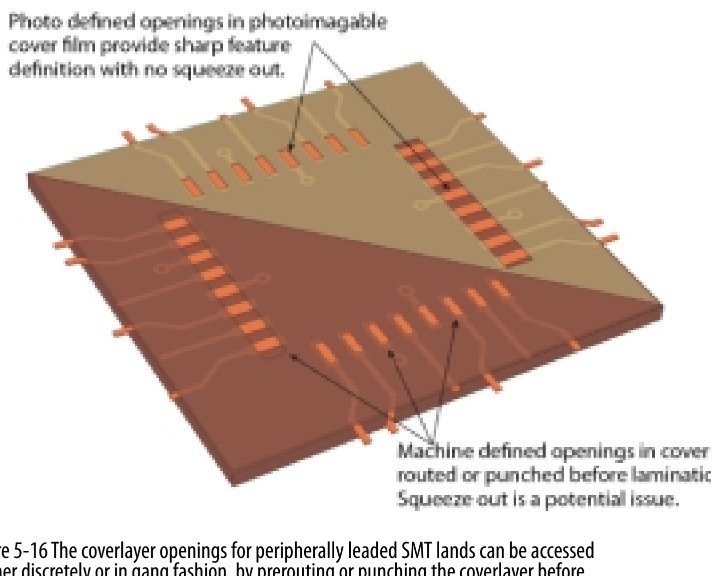

Figure 5-16 The coverlayer openings for peripherally leaded SMT lands can be accessed either discretely or in gang fashion, by prerouting or punching the coverlayer before lamination. Use of photoimageable coverfilms makes this task much easier.

circuit design applications while maintaining hold down capability.

Lands for Plated Through Holes

Except for the shared need for filleting, double-sided flex with plated through holes does not require tie-downs, due to the rivet effect from the plated through hole. This inherent feature of the plated through hole serves effectively in preventing the pad from lifting away from the surface of the flexible circuit during soldering processes, should excessive temperatures be used in the assembly operation. Plated through holes are especially advisable if very small pads are required by the design to ensure formation of a reliable solder joint. This may require the addition of a second layer of copper, thus making a single-sided design a double-sided one. But, ease of processing and increases in reliability should, hopefully, offset any increases in cost.

Button Plating

An alternative plated through hole construction can be created using a process called button plating. The process can best be characterized as one where through holes and vias are selectively plated with copper. A finished structure can be seen in Figure 5-17.

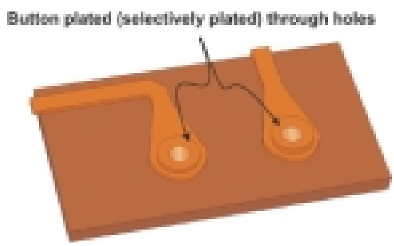

Figure 5-17 An example of button plating

The basic idea is relatively simple, however, success requires reasonable care in the manufacturing process. In practice, the manufacturer first drills and makes conductive the hole walls of the flex circuit using a suitable technology (electroless copper or graphite coating). The manufacturer next takes the panel in for imaging, where the panel is coated with a photoimageable plating resist, and the holes and vias to be plated are exposed and developed. A copper pattern plating step follows, and the hole walls and annuli of the holes are plated to the specified thickness. That resist is stripped away, and a second plating resist is applied and exposed to create a positive circuit image, which can then be etched to create a metal circuit pattern. The holes and vias are tented over in this process. This action prevents the metal etching chemistry from entering the vias and etching out the holes, thus creating electrical opens.

Coverlayer and Covercoat Concerns

As mentioned earlier in the section on flex circuit materials, there are several types of flexible covercoating systems available. Each has its own special applications and advantages. Included among them are the following:

Adhesive-backed films

Adhesive-backed polymer films are the type of coverlayer most frequently specified and used by flex circuit designers and manufacturers. It is also the flex circuit covering method best suited to dynamic flex circuit applications because of the balanced material properties from side to side.

Screen-printable liquid covercoats

Applied and cured by simple means, screen-printable liquid covercoats are the least expensive covercoat type and the one most often used with polymer thick film and simple single-sided copper constructions.

Photoimageable liquid and film polymers

Newer methods for covercoating flex circuits involve the use of photoimageable polymers. The results have been very promising. In process, the flex circuit is coated with a polymer film, which can then be imaged and developed to access termination features. This method, which looks quite good for many applications, could help put an end to many of the coverlayer misregistration problems manufacturers have had with small features and, in addition, quell concerns over adhesive squeeze-out onto lands.

In most flex circuit designs, the coverlayer or covercoat serves more than one purpose. For example, covercoats commonly function as a solder mask, helping to prevent solder from shorting circuit traces together, and serve to isolate electrically and protect physically the circuit from damage.

In addition, as described earlier, coverlayers serve to help restrain the pads physically and hold them in place during soldering, preventing pad lift. The coverlayer (or possibly a covercoat) also allows conductors to be placed in the neutral axis for improved flex and bending performance. This subject will be covered in more detail later in the chapter.

Given the diversity of the roles flex circuits play in electronics packaging, it is understandable that it has been difficult for suppliers to come up with a universal solution that is at once low cost, high performing and easy to apply. Nevertheless, steady progress is being made by material suppliers, and new solutions are regularly being developed and offered to the industry.

Sizing Coverlayer Openings

As was seen in the discussions on surface mount land design, the sizing of coverlayer openings varies according to pad design features and with the number of metal layers. Again, the key area of concern is with single-metal-layer flex circuits, where potential pad lift demands special care in design. Table 5-3 provides general guidelines for sizing of coverlayer openings.

Trace-to-Cut Line Concerns

Best current practice for flex circuit design generally recommends that the edge of the part to the edge of conductor spacing be >1.25mm (0.050"). It has been shown, however, that circuits can be made reliably with edge-to-conductor spacing of ≤ 250µm (0.010"), although this normally comes at increased cost, which will vary depending on the tooling system used. Refer to Table 5-1 on page 77.

Flex Circuit Type	Coverlayer Opening
Single metal layer flex circuit with land hold down features	Coverlayer opening can be roughly equal to pad diameter.
Single metal layer flex without land hold down features or filleting	Openings in coverlayer should be 250µm (0.010") less than pad diameter.
Double-sided flex PCBs and multilayer flex with plated through holes and filleted lands	Coverlayer opening can be equal to or slightly larger than pad. This minimizes squeeze out.
Non-component plated through hole vias	No opening unless needed for electrical test purposes

Table 5-3 Coverlayer opening guidelines vary with the nature of the design.

Tear-Resistance Features in Flex Design

All flex circuit designs should be made as tear-resistant as possible. While the material may not be intrinsically tear-resistant, tear-resistance can be improved by employing certain features in the design. There are several possible methods, described below and illustrated in Figure 5-18.

All of the following techniques have been successfully used to help prevent tearing. One or more of the following techniques can be used:

Radius All Internal Corners

The first line of defense against tearing is to make certain that all internal corners are provided with as generous a radius as possible. This design practice is the most important and simplest of all methods used to prevent tearing of the flex circuit material.

Leave Metal in Corners

The circuit design should, if possible, have small areas of copper provided for at internal corners to serve as tear stops at the inside of corner radii. This serves to prevent further or imminent propagation of a tear through the polymer, should a tear in the material start.

Laminate Glass Fabric in Corners

Glass cloths can be laminated into corners during the fabrication process. Though not flexible, this method has been shown to provide a very robust corner construction and has been favored in the past by military product designers. It is an expensive solution, however, because of the type of preparation required and should be used only after careful consideration of the alternatives. (See Figure 5-19.)

Use Fluoropolymer Coverlayer

The use of fluoroplastics such as Teflon® as coverlayers helps to improve tear resistance by virtue of the high tear resistance of the polymer itself. This is due to the fact that fluoropolymer tends to stretch rather than tear, adding toughness to the substrate. An additional benefit of using fluoropolymer coverlayers for those involved in high frequency design is that the dielectric constant of the coverlayer is much lower.

Use of Radiused Slots

The use of slots with ends that have a radius to access relieved circuit features also can serve to provide tear resistance. Normally, such features can easily be provided for during the punching operation or other circuit fabrication process.

Drilled Holes at Corners or Ends of Slits

Drilled or punched holes in corners or at the ends of access slits have been used with success when flexible appendages must be spaced close together. This method allows the greatest use of material, but the hole size chosen will impact tear resistance. If the hole is very small, the overall robustness will be reduced.

Aramid Fibers Inside Cut Line

As an alternative to glass cloth, the use of aramid fibers routed through corners or along the entire outline of the flex circuit is a unique method to stop tearing of flex circuits. The thin polymer fibers have very high strength and are very pliable, minimally affecting flexibility. However, this is a labor intensive method and should only be specified with the knowledge of cost impact.

Ultimately, the choice of which method to use to restrain or prevent tearing of the flex circuit is not of overriding importance. What is very important is that the designer makes certain that some suitable method to protect against tearing is used.

To summarize, all internal corners should be provided with radii, and it

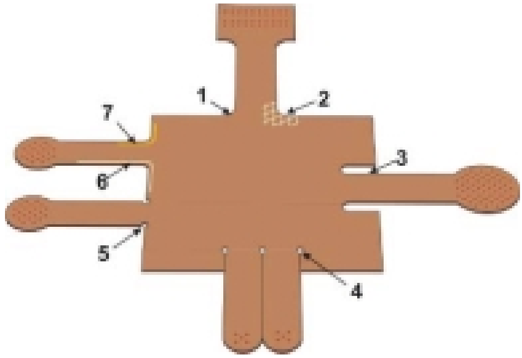

Figure 5-18 Tear-resistance features are important. A number of methods for tear restraint work well for flex. Shown above are: (1)Large radius in corner (2)Embedded glass cloth (3)Recessed slot (4)Hole in slit (5)Drilled hole at corner (6)Embedded aramid fiber (7)Extra copper in corner

is here reemphasized that square or sharp internal corners are an invitation to trouble and should be studiously avoided. If the area of the corner is to be permanently bonded to a rigid base, then it is less important but still recommended that a radius be used.

Stiffeners and Reinforcements for Flex

Stiffeners or reinforcements are commonly used to support components

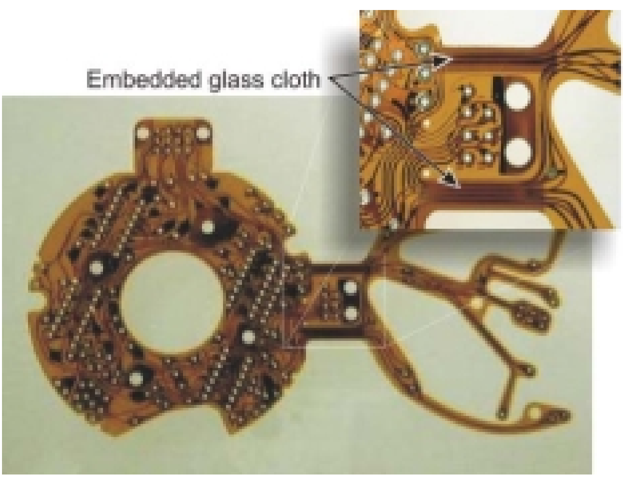

Figure 5-19 Example of circuit with embedded glass-cloth tear restraint

on flex circuits. These important “add-ons” can be fabricated from a wide range of materials, depending on design need. The choice of material is predicated on what objectives are sought (low weight, best heat sinking, lowest cost, best spring qualities, etc.). The materials referenced in Table 5-4 have all been successfully employed to reinforce flexible circuits.

In addition to materials mentioned in the table, the package or box into which the circuit is to be placed can also be used as the stiffener if the design allows. Beyond simple component support, this technique allows the package or box itself to be used for heat dissipation. While a potentially attractive solution for a number of applications, the difficulty of this method comes to light if repair is required, because removal can damage the circuit.

Resin-glass laminates	Extra layer of coverlayer
Thermoplastic sheet	Beryllium copper
Stainless steel	Anodized aluminum
Injection-molded bases	Quartz glass

Table 5-4 Stiffener Material Choice for Flex Circuits. A wide range of materials, both conductive and insulating, can be used to provide stiffness to a flex circuit where required. The table above notes some of the materials that have been used for different applications and needs.

Special Techniques for Stiffeners

Special design techniques allow stiffeners to serve more than the function of component support. For example, while the primary purpose is to support components, stiffeners can be so designed as to aid assembly by enabling the flex to be assembled as virtually a rigid board. This can be accomplished by using one of the following techniques.

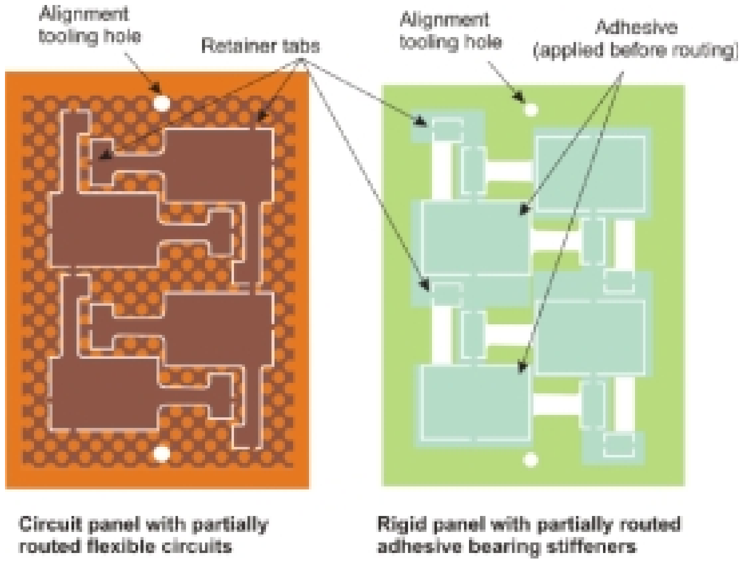

Figure 5-20 Gang assembly or mass application of stiffeners in panelized form facilitates both the application of the flex circuit to the stiffener and subsequent component assembly. When the flex circuit and stiffener are bonded together, the resulting flex circuit panel can be processed much like a rigid board. Note that the adhesive is applied oversized and cut to dimension before the routing step is performed.

Rout-and-Retain Stiffeners

Rout-and-retain stiffeners are produced by CNC routing of the substrate so as to leave it attached in certain locations for easy removal later. Such constructions allow the stiffener to be snapped or cut off after assembly. (See Figure 5-20.)

While routers are pervasively used in circuit manufacturing, lasers and water-jet cutters are other potential manufacturing choices for preparing or pre-cutting stiffeners

Return to Web Punching

Return to web punching (also referred to as "punch out, punch in") of the stiffener requires special punch tooling wherein the rigid material is punched out of the panel and then immediately pushed or punched back into its original position in the panel. The method is commonly used for inexpensive rigid boards and allows mass assembly with relatively simple assembly fixture requirements.

Scoring or Dicing of Stiffeners

If features of the flex circuit design allow the use of scoring or dicing tools to prepare the stiffener panel is potentially possible. With respect to the scoring process, the circuit and/or the stiffener is cut partially through, using special tools which cut a controlled-depth straight path through the rigid circuit material. The cut can be made through the rigid material alone or through both flex circuit and rigid base. After component mounting and assembly, the circuits can be snapped apart along the score lines.

In contrast to the routing concept shown, the other alternative—dicing—requires cutting completely through the circuit and stiffener. Because of the nature of the tools used, all material cuts must be made in a straight line and orthogonal to the major (X&Y) dimensions of the panel.

Adhesives for Bonding of Stiffeners

All of the bonding adhesives used in the creation of flex circuit laminates are candidates for attaching a flex circuit to a stiffener. The choice of which adhesive to use is most often a function of performance requirements.

It is worth checking with the flex circuit vendor for his recommendations. Beyond those adhesives used in normal flex circuit construction, there are other types of adhesives that can be used as well. Following are some of the more commonly used adhesives for stiffener attachment.

Pressure-Sensitive Adhesives

Pressure-sensitive adhesives are very commonly used to attach stiffeners. They are perhaps the most versatile and easiest to use. They exhibit very good bond strength, which in some cases actually improves with age. These adhesives are not generally designed for extended use at high temperatures but are for the most part limited to enduring only short excursions at high temperatures (soldering temperatures). Again, with lead-free solder technology moving ahead, there is need to check capabilities when using higher-temperature lead-free solders.

One particular advantage PSAs offer over other adhesive choices is that, when applied directly to the flex circuit, they allow for the flex circuit to be bonded to virtually any surface, thus effectively making anything in the package a potential stiffener.

Thermosetting Adhesive Films

Thermosetting adhesive bonding films (cast-acrylic films or flex circuit bondplies) can also be used to bond flex circuits to stiffeners, but they require the time and expense of an additional lamination step. Even so, thermosetting film adhesives can offer very high bond strength of the flex to the stiffener.

Liquid Adhesives

One and two part liquid epoxy type adhesives have been used for bonding stiffeners to flex circuits. They are difficult to apply uniformly and thus do not enjoy wide popularity. Such adhesive materials are well suited for—and can be well applied in—the creation of strain relief at the transition edge of the flex and stiffener by creating a bead of epoxy along the entire edge of the transition.

Thermoplastic Adhesive Films

The use of thermoplastic-based adhesive films for bonding flex circuits to stiffeners is another common option. Thermoplastic films have some unique advantages among adhesives in that they are low-stress, fully-polymerized polymer resins that require no cure. With properties that include adhesion to a wide variety of surfaces and materials, and the reported ability to be reworked easily, these adhesive materials may see expanded service in the future.

UV Curable Adhesives

Ultraviolet curable adhesives are another potential adhesive choice for stiffener attachment. With some screen-printable formulations, the UV "activates" the polymer, creating a tacky adhesive with PSA qualities. In addition, because they can be rapidly cured, these adhesives are also an attractive choice for relieving strain on the flex circuit at the transition point from rigid to flex.

Holes for Stiffeners

The diameter and relative sizing of mounting holes for components, of the circuit or breaking of the copper at the transition point can more easily occur if the procedure is omitted from the design or manufacturing process. One or both of the following techniques should be used.

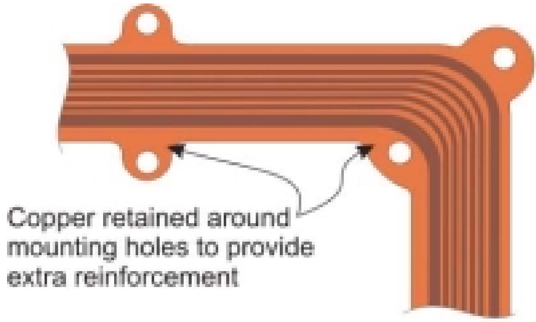

Figure 5-22 Mounting holes should retain copper when flex circuit is to be mounted without mechanical support.

Rounded Stiffener Edges at Transition

The stiffener edges in areas where the flex circuit egresses from the perimeter of the stiffener should be rounded or provided with a radius at the edge to prevent a point of focused stress. Alternatively, breaking the rigid stiffener with a file or sandpaper at the transition edge before assembly can provide a similar benefit.

Fillet Transition Edge of Stiffener

Filleting of the transition edge of a stiffener with a resilient adhesive or epoxy is another common method of strain relieving circuits. The small bead of a suitable polymer will provide a simple means of transitioning strain from the stiffener to the flex circuit. Figure 5-23 illustrates the two approaches.

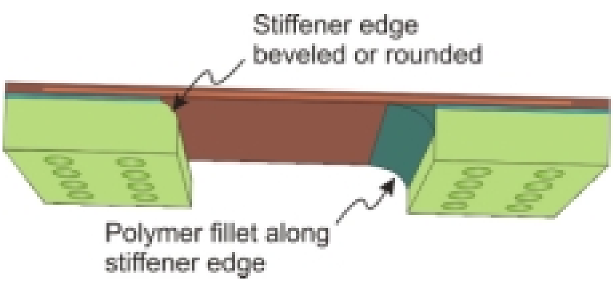

Figure 5-23 Strain relief at the flex to rigid transition helps minimize the potential for focus stress.

Strain Relief for Unsupported Flex Circuits

Strain relief should also be provided when mounting the finished circuit or assembly. The following methods can be used for this purpose:

1) Break or radius sharp edges of any retaining bars or clamps may be used to hold the flexible circuit in place.
2) Use a low modulus, elastomeric material between restraining bars and the flex circuit.
3) Bond the circuit to the assembly housing, using a double backed adhesive foam material or simple pressure-sensitive adhesive.

and those for final flex circuit assembly mounting, have different purposes and often somewhat oppositional requirements. The result is that the design rules can vary considerably, depending on the application. Explanations as to how to determine the appropriate size follow. Figure 5-21 illustrates the concept.

Component Holes in Stiffener

Holes in the stiffener for through hole mounting of electronic components, such as dual in-line packages (DIPS), should be 250µm-375µm (0.010-0.015") larger than the through holes in the flex circuit (which, in turn, are by design rule 250µm [0.010"] larger than the lead). This is to allow for any movement and misregistration between flex and stiffener that might occur during the stiffener lamination or bonding process. This technique also helps to assure the greatest opportunity of accessing the through hole with the component lead without interference from the stiffener.

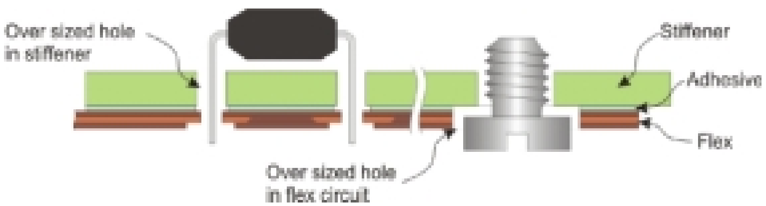

Figure 5-21 Access holes through the stiffener fill different needs. Holes in the stiffener for leaded components are made slightly larger than holes in the flex circuit. This allows for any movement during the stiffener lamination process. It also assures maximum opportunity for accessing the through hole with the component lead.

Assembly Mounting Holes

Holes in the stiffener for mounting the assembly should be equal to or slightly smaller in diameter than the holes in the flex. This assures that the stresses are placed on the rigid portion of the assembly and not on the flex circuit. This is not an ironclad rule, as it is possible to mount the flex circuit directly to a carrier without a stiffener, using common mounting hardware if necessary.

Unsupported Mounting Holes

Mounting holes that are not supported by a stiffener should be designed to maintain copper around the hole for added strength. (See Figure 5-22.) This practice is of value with regular mounting holes, as well, if the design will permit. Such features are also a convenient means of making a connection to ground.

Strain Relief for Flex Circuits

The provision of strain relief at the edges of stiffeners helps to prevent stress risers from occurring at transition areas from flex to rigid. Tearing

Methods of Connecting Flex Circuits

Connecting a flexible circuit to other elements of an electronic system is a vitally important element in system design, manufacture and assembly. There are numerous methods of making connection to flexible circuits. Virtually all connector manufacturers have connectors either specially built for, or readily adaptable to, flex circuit designs.

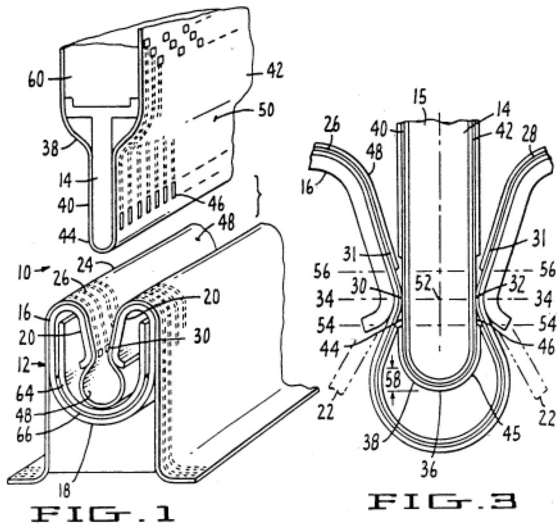

Figure 5-24 Patent drawings of a novel high density flexible circuit based connector invented by John Krumme and Gary Yasumura while they were at Beta Phase. A shape memory alloy was used to open the unique zero insertion force (ZIF) connector.

One innovative connector manufacturer of record, Beta Phase, Inc. (Menlo Park, Calif.), actually made its connectors out of flexible circuits, producing high-performance connectors with very high pin count equivalents. The concept was much ahead of its time and did not get much use outside of extreme performance applications, such as the Cray supercomputer, but the technology was purchased by Molex, and some elements of the earlier concepts are now available. As well, other connector manufacturers now have comparable product in the market.

Flexible Circuit Connector Types

Some basic connectors are relatively simple devices. Examples include insulation displacement and crimp-type connectors. These have proven popular in applications where cost is important. They are not generally considered suitable for high-reliability applications, however.

To make both male and female pin in socket type connectors, swaged or brazed pins can be attached directly to the flex circuits. These have also brought some success in certain low-end product areas where performance is not a key concern.

The Sculptured® flex circuit technology described earlier has the ability to integrate the connector directly into the flex circuit itself. The method is suitable for a number of different electronic applications, due to the fact that no discrete joining of pins to the flexible circuit is required. By using this technology approach, it is possible to have edge contacts that extend, unsupported, beyond the edge of the flex circuit. It is then possible to simply post-form the leads as required to create a viable male pin connector for mating with a compatible socket.

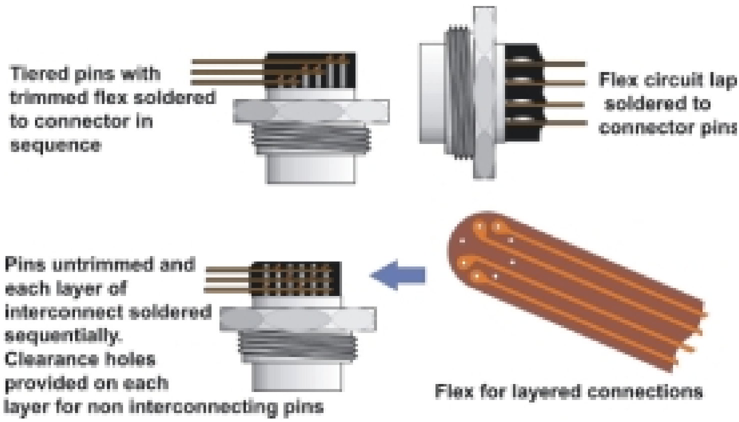

Figure 5-25 Historical approaches for soldering military style two part connectors to flex circuits. A key concern is inspection, as solder joints are hidden by each successive layer.

In addition, or as an alternative to the sculpturing method described, edge card contact constructions can be created by folding the contact area of a flexible circuit around a stiffener. This is a simple and relatively inexpensive way to interconnect a flex circuit. It is directly analogous to edge card contacts on rigid boards, for which there are numerous types of mating connector solutions available. Because the flex circuit itself is thin, it is possible to accommodate a wide range of connector designs simply by altering the thickness of the stiffener (see Figure 5-26).

Surface mounted connectors are an increasingly important and common connector choice for use with flexible circuits, for obvious reasons. With size reduction a common objective

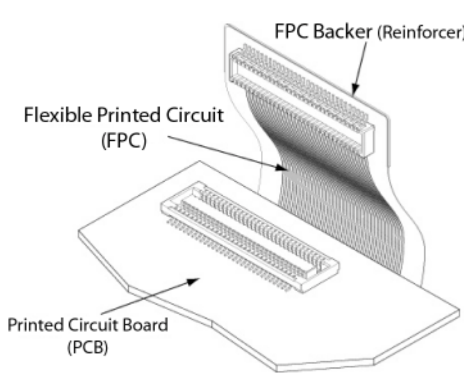

Figure 5-26 Surface mount connectors for flexible circuits facilitate high density interconnection. (Graphic courtesy Molex)

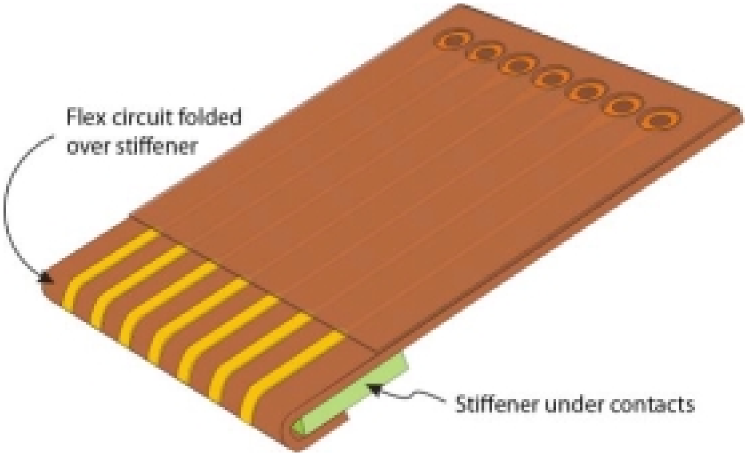

Figure 5-26a Ends of the flexible circuit can be converted to an edge card contact by placing a stiffener behind the flex and folding and bonding the flex to the stiffener.

of flexible circuit technologies, it comes without surprise that low-profile connectors are well-suited to the needs and abilities of flex circuits.

A number of low-profile connectors are presently in the market. These miniature connectors are very nicely suited to many space-constrained flexible circuit applications. Low-profile connectors known as low-insertion-force (LIF) and zero-insertion-force (ZIF) connectors that can handle contact pitches down to 0.30mm (.012") have been produced by commercial manufacturers. The profile height for such connectors can be as low as 0.60mm (.24").

Another option for low profile interconnection of flexible circuits is lapped connections. Solder, conductive polymers and adhesives have all

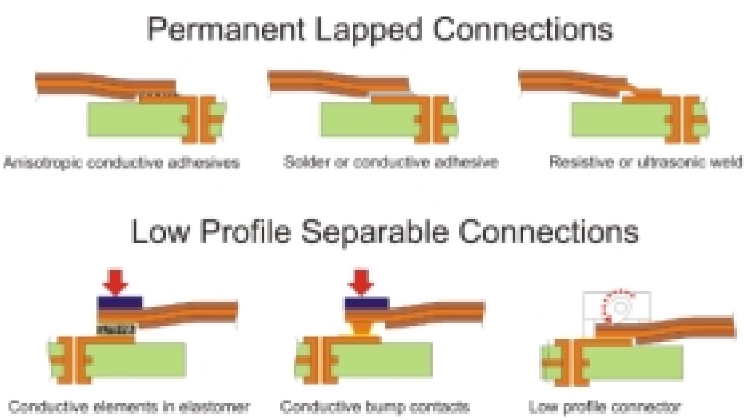

Figure 5-27 Low profile connections of flex circuits can be accomplished in many ways. Both permanent and separable connections are possible. Shown above are but a few examples.

been used to make lapped connections between a flex circuit and a mating interconnection structure. It is a reasonably common method; for example, a large percentage of flat panel displays are connected using anisotropic adhesives.

Another unusual design approach to making flex circuit interconnections is one wherein the connections are made directly between chips, using anisotropic adhesives or lap soldered connections. This approach has been proven, both by modeling and manufactured prototypes, to be capable of providing very high speed and low power.

To summarize the topic, connecting the flexible circuit to a next level or associated interconnection device or system can be accomplished using one of many options. The choice is predicated on the cost and performance requirements of the end product. The examples provided are not exhaustive in terms of options, but they are representative.

Bending and Flexing Design Concerns

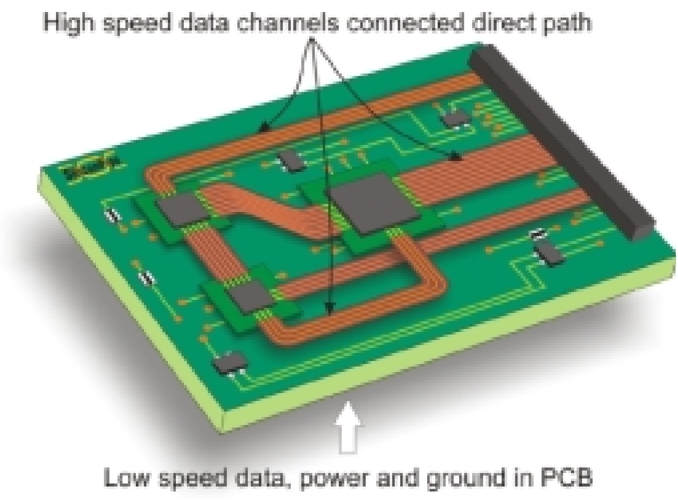

Figure 5–28 Making flex circuit connections directly between chip packages can provide significant performance improvements. (Courtesy SiliconPipe, San Jose, CA)

While flex circuits typically are employed simply to allow the user to form the circuit to fit the shape of the package (flex to install applications), there are still many applications that require some dynamic flexing. In fact, in most applications, the very act of placing the flex circuit into the assembly requires that the circuit be bent or folded. In some applications this can occur several times. Flexible circuits are capable of enduring many millions or even billions of flexural cycles, provided the design is properly matched to the task.

Those not involved in dynamic flex design should also take to heart the lessons of this process. For example, it is important to remember that even static flex circuits can be dynamically cycled by virtue of their application and design. Such events are common occurrences in circuits designed for any type of mobile equipment, such as automobiles and planes.

For example, shock and vibration encountered by a vehicle can cause a flex circuit to endure millions of low amplitude, high frequency flex cycles. If dynamic flex design rules are not taken into account or are simply ignored, the potential for unexpected cyclic fatigue failure of an application subjected to shock and vibration exists. Attention to the few simple rules for dynamic flex provided here can benefit many flex circuit applications. They are, arguably, good practice for all flex circuit designs.

Bending and Flexing Techniques

A number of clever approaches and techniques have been developed by engineers over the years to achieve the desired bending or flexing motion in a flexible circuit. The types of motions employed range from linear extension and contraction to rotational flexing through various small angles of 5° or 10° to more than 360°. Figure 5-29 provides conceptual examples.

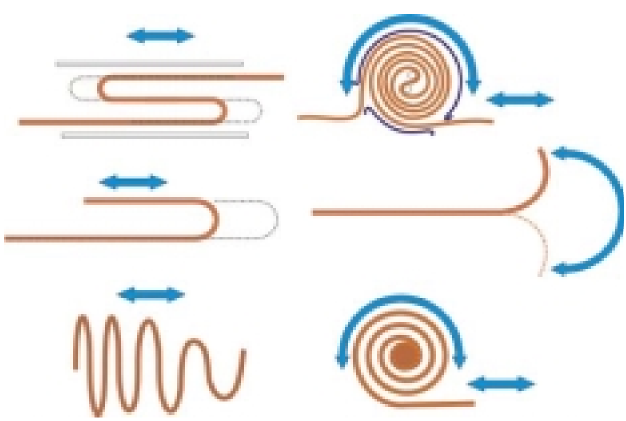

Figure 5-29 Various flexing and bending methods are illustrated. Clockwise from the bottom left: accordion flex, rolling flex, counter rolling flex (must be vertically space constrained), "window shade" flex, large radius or hinge type flex and coiled flex.

Avoid Placement of Through Holes in Bend Areas

An important design practice that is sometimes overlooked or ignored is the avoidance of placing plated through holes in the bend areas. This is of particularly great importance in dynamic applications. For static applications, it may be possible to place vias through a bend successfully if they have a coverlayer and if the bend radius is large enough. That said, it is still a practice that should be avoided.

Route Traces at 90° Through Bend and Fold Areas

Conductor traces should be routed through bending and flexing areas at 90° (perpendicular) to the bend line. This is an intuitively natural routing scheme and serves the purpose of bending well. However, the rule is somewhat fungible and seems to be violated regularly for matters of convenience. For example, in some hinge circuits (see Figure 5-33), the traces may be bent in more than one direction to achieve the design purpose.

Route Conductors on a Single Layer Through Bend

Whenever possible, conductors should be routed on a single metal layer through bend and fold areas to enhance flexibility. When not possible, the conductor should be staggered from side to side to avoid the I-beam effect discussed earlier.

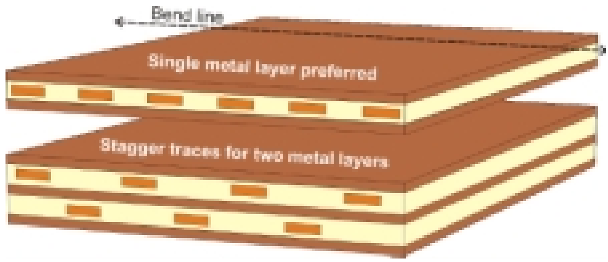

Figure 5-30 It is preferred practice to route traces through bend areas in a single metal layer. If two metal layers are required, the traces should be offset or staggered.

Design to Keep Copper in Neutral Axis

The concept of neutral axis is very important to flexible circuits. In theory, the center of any item being bent is nearly immobile, with stress being absorbed by the outer layers of material. Therefore, if the copper (or other metal) foil is kept to the center of the design, the flexing life should be enhanced. (See Figure 5-31.)

Many experiments have verified this theory. Data provided in the graph in Figure 5-32 dramatically illustrate the effect that neutral axis placement can have on the flexing life of a flex circuit.

Flex Dynamic Areas with the Copper Grain Direction

The orientation of the grain of the copper foil has a definite effect on flexural life of a design. Grain direction is of greatest importance with flex circuit designs fabricated using rolled and annealed (RA) or traditional

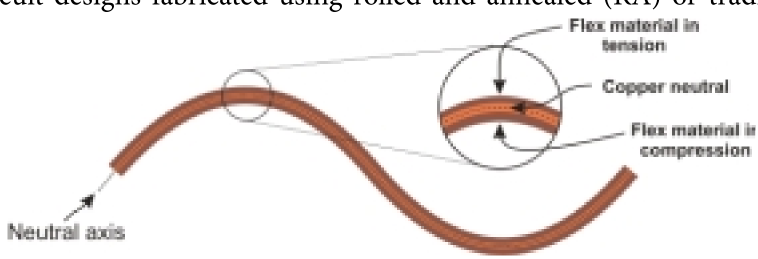

Figure 5-31 By keeping copper in neutral axis, it is possible to minimize cyclic strain and vastly extend flex endurance.

electro-deposited (ED) copper foil. With vendor-electroplated copper on sputtered film, orientation is not as critical, as there is no specific grain direction. The effects of grain direction on flexural life can be very significant, as the data found in the graph in Figure 5-32 indicate.

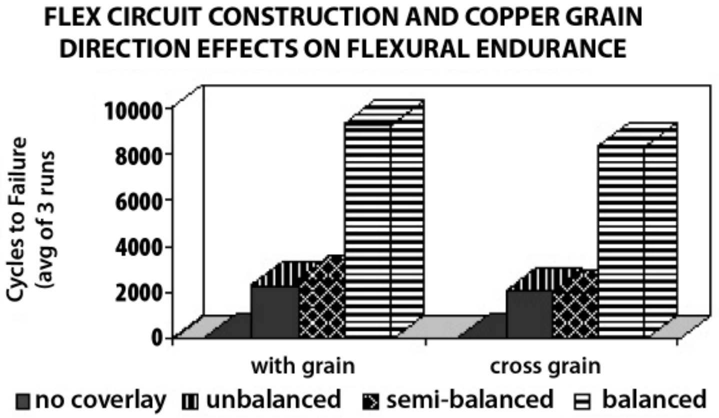

Figure 5-32 Data show construction influence on flexural endurance.

Notes:

1) The unbalanced construction consisted of 25µm polyimide with 25µm adhesive on a base of 50µm polyimide with 25µm adhesive and one-ounce copper.
2) The semi-balanced construction used adhesive to achieve the desired balance (25µm polyimide with 50µm adhesive). The base was the same as above.
3) The balanced construction had a coverlay makeup that matched the base material exactly (50µm polyimide with 25µm adhesive).

Keep Flexural Arc Small

For maximum flex life, it is best to keep the range of the flexural arc or total angle of flexure of the circuit for dynamic designs as small as possible (that is, flex the circuit over the smallest possible distance). This is a key technique used in later-model disk drive applications to allow them to achieve their present high-flex-life cycling.

Provide the Largest Bend Radius Possible

The designer is advised always to provide the largest practical radius through bend areas. This design approach is especially important for dynamic flex, but, as has been previously noted, it can also be

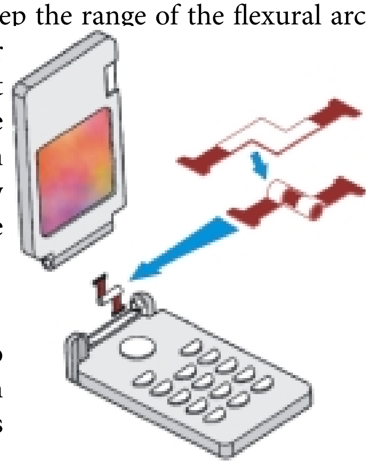

Figure 5-33 Example of a hinge flex design concept for a cellphone (Source: US Patent No. 6948240)

important in flex applications that are apparently static in nature.

The graphic and simple equation in Figure 5-34 illustrate the effect of bend-radius diameter on the copper foil. As can be concluded by calculation, the elongation requirements for the copper foil rise significantly as bend radii decrease.

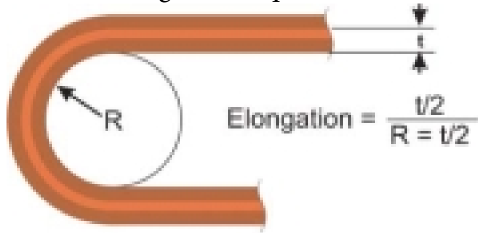

Figure 5-34 Small-diameter bend radii demand greater elongation from materials used in flex circuit construction, especially copper foil.

Guidelines for Minimum Bend Radii

While finite element modeling can provide excellent predictive data for suggesting bend limits, there are some common guidelines that have long served to keep the design inside the limits. For normal bending of flexible circuits, those guidelines can be found in Table 5-5. For very high flex life dynamic flex circuit designs, fabrication and testing of prototypes commonly remains the preferred method of design verification.

Flex Circuit Type	Minimum Bend Radius
Single Metal Layer	**3-6 x circuit thickness**
Double-sided Flex	**6-10x circuit thickness**
Multilayer Flex	**10-15x circuit thickness (or more)**
Dynamic Application (Only single-sided recommended)	**20-40x circuit thickness** (increase in bend radius increases life)

Table 5-5 Minimum bend radii design guidelines for flex circuits

Creasing and Folding Flex Circuits

Creasing and hard folding of flex, while not a preferred practice, can be successfully accomplished with some attention to certain details. When required, circuit should be bonded to prevent it from bending back at the crease or fold line. Strain relief is also recommended. As noted earlier, it is important to keep the construction balanced for best flexural endurance life. The ideal copper for such strain-bending applications will be a low strength, high-elongation copper. Fully annealed soft copper is normally a good choice for applications requiring a small radius bend.

Bending Flex Circuits to Hold Shape

When bending flex circuit products for static, form-to-fit applications, holding shape is a desirable condition. However, flex circuits sometimes have some elastic memory. The following principles for shaping flexible circuits to fit permanently in their application will help to overcome the condition. The first principle is maximizing the metal area. Copper, or any other metal one might use for the conductors, will permanently deform plastically when bent beyond its elastic limit.

Many polymers (elastomers are generally excluded, although they can take a set over time) will also permanently deform if their elastic limit is exceeded; their limit, however, is many times greater than that of metal. Thus, when the composite structure that we now call a flex circuit is bent, the metal has plastically deformed, while the polymer is still likely to be in its elastic range.

Provide for Metal Dominance in Bend Area

In order for the copper (or other metal) to prevent the polymer from snapping back, it must overwhelm the elastic memory of the polymer. Copper is stronger and higher in elastic modulus, but if the traces are small or the copper is a low percentage of the local area, the remnant elastic strain in the polymer may cause the flex circuit to regress to its original flat shape. This method is in keeping with the practices used by flexible circuit manufacturers to help maintain dimensional stability.

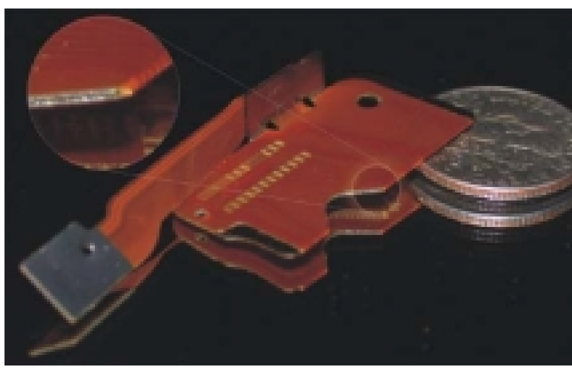

Figure 5-35 Very small bends in the flex circuit are possible, as demonstrated in this figure (from a disc drive application).

Widened Circuit Traces Through Bending Zone

If circuit weight is a concern, the area of extra copper can be localized. In such cases, the circuit features are widened in the area of the bend and then reduced in width again as they enter and exit (see Figure 5-36). Circuit traces should taper to the new width in both directions.

Determining Bend Area Length

As to the potential question, Through what length of the bend area should the traces be widened? A simple method to get a first order approximation is to determine the circumference of an imaginary circle having the desired bend radius, and multiply that result by the bend angle divided by 360 (the degrees in a circle). This should ensure that a sufficient

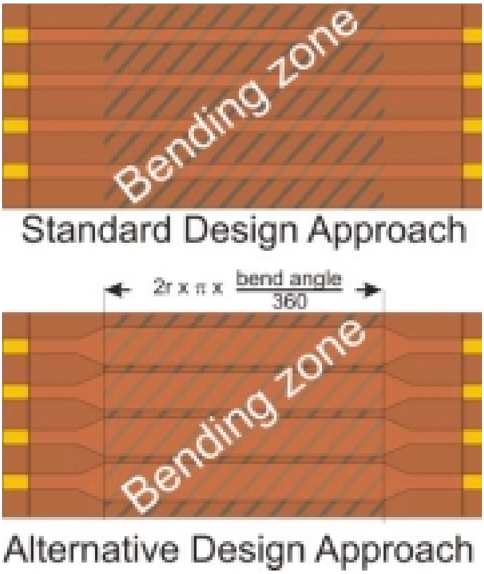

Figure 5–36 To create a permanent bend, copper traces can be made wider through the bend area. An alternative is to use thicker copper.

amount of the bend area is filled with the wider copper traces. However, a little extra length may be required, depending on the construction.

Use Thicker Copper in Bend Area

If widening the traces alone does not help sufficiently, then one of two analogous methods can be used. One can either use a thicker metal foil or use a thinner flexible base material. The objective remains the same: Make certain that the metal can overwhelm the polymer in order to hold the final shape. There are advantages and disadvantages to both paths. Making the copper thicker may make etching a bit more difficult; it will also take longer to etch and will use more chemistry. On the other hand, making the polymer thinner could make handling a bit more difficult, and the strength of the final assembly will not be as great as with the alternative method.

Permanent Shaping Alternatives

Slight over-driving of the bend to create a stable, more permanent shape can be used to advantage, with the caveat that the minimum bend radius for the flex construction not be violated. Note that when permanently bent and plastically deformed, the copper is normally thinned in the area of the bend, and is thus weakened. If a truly accurate predictive solution is desired, one can use finite element modeling methods.

Heat Forming

Another way to get the flex circuit to hold shape is to form it (usually using a mandrel of some sort), bend the circuit into shape, and then apply heat to the fixture, allowing it to cool in place. The objective is to relieve all of the remnant elastic strain in the polymer by allowing it to deform plastically to the final shape with the addition of heat. This approach works easiest with polymers and/or adhesives that have relatively low melting points. It is a very common method for making dome switches in polymer thick film circuits using polyester base materials.

In contrast, polyimide has a very high melting temperature, making it a less attractive candidate. In this method, one must normally rely on the ability to get above the glass transition temperature of the adhesive and allow the circuit to cool back below that temperature before releasing it from the mandrel.

Use of Low-Modulus Polymers

A final choice for permanent shaping is to use nontraditional base materials. These would be materials of low strength and having little if any elastic strength. Unreinforced FEP or PTFE [Teflon], for example, falls into this category. This combination will allow the user to deform the circuit permanently into the desired shape. There are other options that are variations on one or more of these themes, but these are the basics.

Summarizing this topic, keeping and holding flex circuits in shape is not that difficult, but it does take some attention. The method of choice relative to those mentioned above will, obviously, depend on the demands of the design and its application.

Finite Element Modeling of Flex

Over the years, with increases in computing power and increased availability of memory, there has been a significant improvement in finite element modeling tools, in terms of both cost and performance. Thus, it is not surprising that—given the importance of the mechanical requirements to the long-term performance of a flexible circuit assembly—many companies are beginning to perform finite-element modeling of the circuits to validate the design before committing the product to manufacture. This can be much more cost effective than iterating through a number of prototype runs, provided the modeling parameters are properly selected.

FEA tools are widely available and many can accept data directly from many types of design software and the user need only input

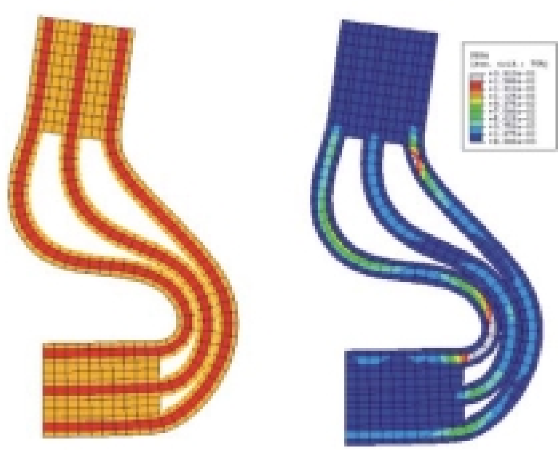

Figure 5-37 Finite element modeling can significantly improve the chances of first-pass success in design for application by providing valuable information about the location and magnitude

material properties. Meshing is automatic. Figures 5-37 and 5-38 illustrate where the strain is located and dislocation in a simple bend and also re-emphasize the importance of coverlayer in reducing strain on the copper foil.

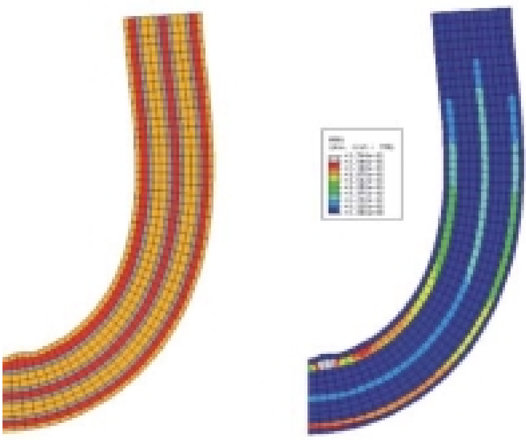

Figure 5-38 Finite element modeling illustrating stress in a three-metal-layer flex circuit during bending. Note buckling of material in bend area in lower image. (FEA models courtesy Michael Perry)

Shielding Flex Circuits

With the proliferation of wirelessly operated electronic products, there is increasing concern around the world about electromagnetic interference or EMI. Shielding of flex circuits may be required to block out unwanted electronic interference or noise. There is, in addition, the converse need to minimize emissions emanating from the circuit as well. Shielding can be accomplished by using the electronic system in a shielded room, but this obviously is not practical for most of today's electronics. Alternative methods, therefore, are necessary.

Following are some techniques developed for flexible circuits:

Integral Foil Shields

The use of laminated copper (or other metal) foil on the outer surfaces of the circuit can provide excellent shielding. Such approaches should be carefully weighed to assess their cost-effectiveness, however. If only simple shielding is required, lower-cost screened-on coating may serve. All metal foils also tend to be stiffer and heavier than the alternatives.

Thin Metal Shielding

Vacuum sputtering of metal, and other dry metallization processes such as vapor deposition, have been used successfully to metallize the outer surface of the circuit, providing the required shielding. Such shielding is very lightweight and has been successfully employed in satellite applications. The process requires the use of expensive equipment and may not be suitable for cost-sensitive applications.

Screen-Printed Conductive Polymer Shielding

Screen-printing the surface with conductive polymers is a technique that has been used with great success in many applications. To use this technique, access to ground must be provided through the coverlayer. This allows the conductive ink to be screen-printed down into the opening and make contact. No openings are required when a floating ground is satisfactory for the application. (See ground plane section.)

Graphite Coatings

Depending on the level of signal attenuation sought or required by the application, lower-conductivity coatings such as graphite or carbon films may also serve the user's needs. Graphite coatings can be easily applied by spraying.

Membrane Switches

Membrane switches are ubiquitous. Any time one physically interfaces with the world of electronics, the odds are that he is doing so through a membrane switch of some type. As common as they are, membrane switches are also perhaps one of the more unsung and least visible members of the flex circuit family. As switches are fundamental elements of the electronic interconnection hierarchy, it is worthwhile to look more deeply into this important component.

Basically, a membrane switch is—as its name implies—an electrical switch created on a thin film or membrane. They are typically of low power, with maximum current ratings of around one-tenth of an amp. The circuitry for these devices is often somewhat elaborate since they frequently provide connections for a host of different input functions. Perhaps the most common application for membrane switches is in a keyboard of some type. While not all keyboards are made of flexible materials, a great many are. The most common layouts are matrix type (rows and columns) and common line connections (a common trace plus some number of switches). Other structures are possible depending on the needs of the user, such as integration of electronic circuits (including passive devices such as resistors) and land patterns for component mounting.

Conductor Materials for Membrane Switches

The conductor material used for membrane switches varies by application. Copper and polymer thick film (PTF) inks are the most common choices. Cost is usually a key factor when making the choice. Because of this, a substantial number of membrane switches have screen-printed PTF conductors consisting of metal-filled ink. Obviously, the typically lower conductivity of printed inks limits the conductivity, but they are not normally meant to carry current. Rather, they are designed to send a simple signal pulse. Copper is employed when there is need to solder devices to the membrane or when higher conductivity is needed; however, conductive adhesives have proven quite acceptable in most applications.

The switch-life of a membrane contact can vary significantly, from several thousand to many millions. The life-determining factors include such matters as materials of construction, contact design, switch travel, and operating conditions, among many others.

Tactile feedback

While some membrane switches do not provide for tactile feedback, which some suppliers call a Type 1 structure, it is arguable that one of the key elements of membrane-switch design involves providing for tactile feedback. This is commonly a small snap or click that can be felt when a switch is pressed and released. Determining the right amount of force to be applied (the actuation pressure) is both an art and a science. Some customers are very adamant about "feel," which, unfortunately, can be subjective.

There are basically two approaches to getting tactile feedback: polymer dome contacts (sometimes called a Type 2 structure) and metal dome contacts (sometimes referred to as a Type 3 structure). Metal-dome tactile switches have spring metal dome over the contact area. When pressed, it snaps down to complete a circuit and snaps back when released. The shape and thickness of the metal (commonly spring stainless steel) will determine actuation force. They offer a long life but are not well suited for use with flex circuits.

In contrast, polymer dome switches are embossed into the plastic film overlaying the circuit. It is possible to get a good tactile feel from such contact, and though their life expectation is heavily influenced by their use environment, they can endure millions of cycles if designed right. Furthermore, they have the advantage when it comes to cost, since they reduce the number of parts—thus reduced assembly time and complexity. Of course, one can opt to not use tactile feedback. To this end, an auditory response method is employed, such as a small beep. Because of their extreme simplicity, these tend to be the lowest-cost contacts of all.

Contact Design

The contact area design is another important and interesting element of a membrane switch. Contact finish can vary. Gold, nickel, silver and even graphite have been used. The layout will vary with the type of contact used. For example, for a shorting contact, interdigitated fingers are often used. However, when a metal dome contact is employed, a central contact with a surrounding ring is frequently seen (Figure 5-39). Much time and effort has been expended over the years to define the ideal contact.

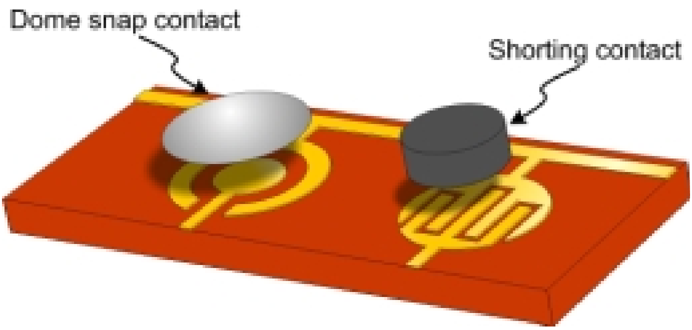

Figure 5-39 Basic membrane-switch contact designs are shown without an overlayer. The shorting contact on the right normally is attached to a resilient material that holds it off the surface when it is not pressed down.

Connecting to Membrane Switches

Perhaps the element most recognizable as a flex circuit in a membrane switch is the tail element. commonly made to serve as half of a mated pair connector. In such constructions, graphite is commonly applied over the circuits as the contact finish. In such cases, the circuit traces are simply plugged into a ZIF-type connector of a sort described earlier in this chapter.

This brief review of membrane switches is by no means complete. It serves only as an introduction to the technology and was meant to provide some appreciation for this important use of flex circuit technology.

Controlled-Impedance Constructions

Controlled-impedance electronic signal-transmission cable applications are one of the applications best suited to the capabilities of flexible circuits. Because of the rapid increase in the growth of high-speed, high-performance electronic products, the use of controlled-impedance interconnections is expected to grow. Following are some of the construction types available using flex circuits. Figure 5-40 shows examples of each type.

Coplanar Stripline

In this very simple method of creating a controlled impedance cable, the circuit is produced with one metal layer by alternating ground and power. Such constructions are well suited to higher-characteristic impedance designs. A drawback of these designs is that they are susceptible to EMI noise.

Microstrip Circuits

Microstrip circuit designs are two-layer flex constructions, of which one metal layer is devoted to ground. Such circuits have been successfully employed in transmission line applications, are normally targeted for a 50Ω characteristic impedance, and are often used for single-ended interconnections. Higher-characteristic impedance designs can be built, but flexibility usually suffers.

Stripline Circuits

Stripline circuits and transmission line cables are also excellent applications for flex circuits. With ground layers on both sides, great signal integrity can be achieved. However, such constructions tend not to 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.

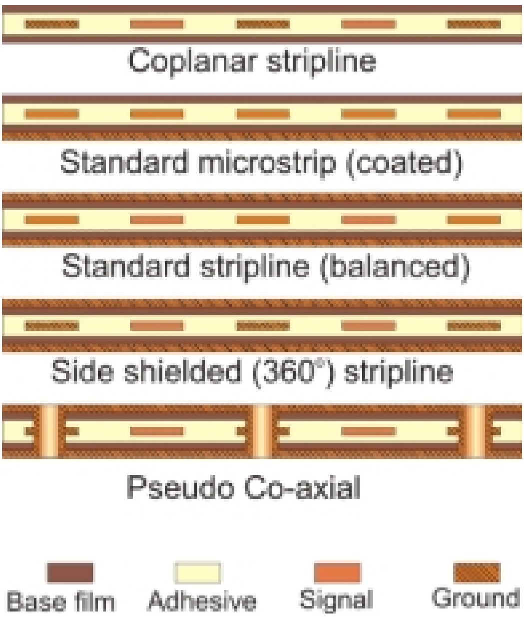

Figure 5-40 Various controlled-impedance constructions

360° Shielded Stripline

360° shielded stripline constructions attempt to replicate coaxial cable constructions by virtue of the fact that the signal line is surrounded on four sides by ground. Such applications are of interest where crosstalk is a concern and where maximum signal integrity is required. Like stripline flexible circuit, these constructions tend to be rather stiff.

Pseudo Coaxial Cable

Some researchers have taken 360° stripline constructions a bit further and used either plated through holes at points along the length of the flex circuit or plated trenches along the length of the copper ground to improve the shielding between the signal lines.

CAD Tools

Designing flexible circuits is, clearly, no mean task. There are many special design elements that must be attended to in order for a design to move easily through the manufacturing process. One important family of enabling technologies can be found in computer-aided design (CAD) tools. Newer CAD solutions are being adapted specially to meet the needs of flexible circuit designers.

In fact, with the increasing emphasis on the use of flexible circuits in all manner of electronic products, CAD tool suppliers, such as Mentor Graphics, are creating new tools to address the growing need for rapid learning in this important technology sector.

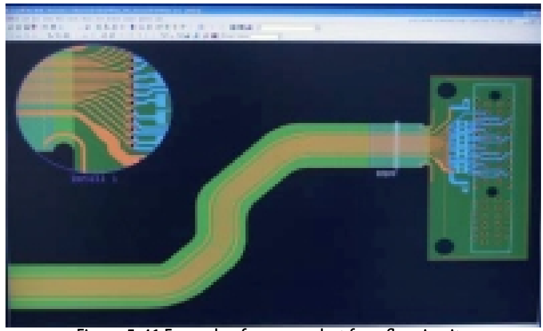

Figure 5-41 Example of a screen shot for a flex circuit design executed using CAD software developed for flexible circuits (Photo courtesy Mentor Graphics)

These new tools not only help to make certain that the appropriate connections are made but also now address the various mechanical needs for curved trace routes to prevent stress risers. An example of a computer aided flex circuit design having one such solution applied can be seen in Figure 5-41.

Summary

This chapter has provided an overview of some basic, yet very important, design practices required for successful implementation of flex circuit technology. As stated earlier, there are many other design requirements for PCBs in general, many of which are common to both flex and rigid circuit designs. Many concerns of flex circuits are created by the mechanical demands placed upon them. The reason is simple: What was formerly a simple replacement for a standard PCB has become a much more complex and highly mechanical, multifunctional interconnection device.

This simple reality forces the electronics designer to give proper consideration to mechanical concerns that could normally be ignored in rigid board design, but which are vital in the designing of flex circuits.