December 8, 2011
A Penton Media Publication
Tune in to EngineeringTV.com

MACHINE Design.com

BY ENGINEERS FOR ENGINEERS

Industrial robots see and feel

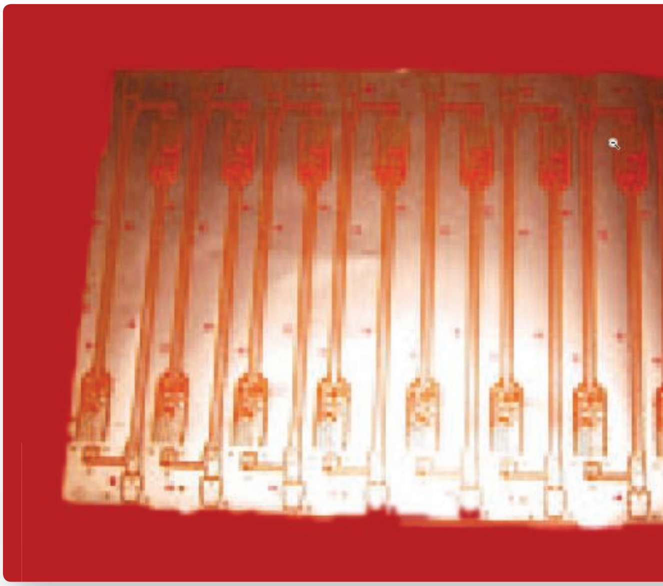
page 44

EXTREME CAR DESIGN, page 22

SMARTPHONE APPS HANDLE CONTROL TASKS, page 50

CABINETS THAT CAN TAKE A BEATING, page 56

BASICS OF DESIGNING WITH FLEX CIRCUITS, page 62



What you should and shouldn't do to keep flexible circuits reliable.

No-vex flex circuits

Authored by:
Mark Finstad
Flexible Circuit Technologies Inc.
Minneapolis, Minn.

Edited by **Robert Repas**
robert.repas@penton.com

Key points:

- A flexible printed circuit is as much a mechanical device as it is an electrical device.
- The neutral bend axis may not be in the exact middle of the material stack.
- The reliability of flexible circuits depends on many design factors.

Resources:

Flexible Circuit Technologies Inc., *www.flexiblecircuit.com*
RS# 621

Staying flexible, *www.machinedesign.com/article/staying-flexible-0712*

Five tips for designing with flexible circuits, *www.medicaldesign.com/electrical-components/tips_designing_flexible/*

A flexible printed circuit is as much a mechanical device as it is an electrical device. Conductors must be laid out such that the circuit functions properly and reliably. Unlike a rigid printed-circuit board (PCB), flexible circuits bend, flex, and otherwise contort to fit the final assembly. These bending and flexing operations can severely strain improperly routed internal conductors.

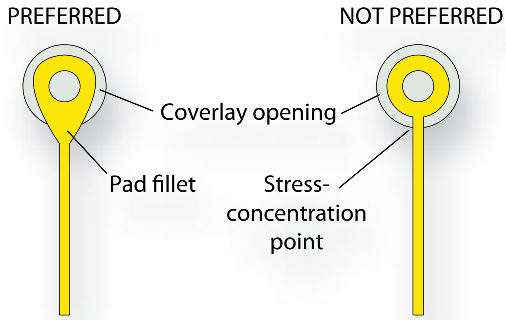
The industry standard IPC-T-50 of the **IPC Association Connecting Electronics Industries** defines a flexible circuit as, "A patterned arrangement of printed wiring utilizing flexible base material with or without flexible cover layers." A typical flexible circuit is formed by stacking four different types of primary layers: the base layer, a metal foil or conductor layer, an adhesive layer that bonds the other layers together, and outer insulating (cover) layers. Multilayer boards stack these four basic layers as needed to complete the circuit design.

The base and cover layers are typically a flexible polymer film that creates the foundation of the flexible circuit and provides most of the physical and electrical properties of the circuit. A number of materials may be used as base films, but most flexible circuits today use polyimide films because of their excellent electrical, mechanical, chemical, and thermal properties.

Normal base-material thickness typically falls between 12 and 125 μm (0.5 to 5 mils), but thinner and thicker bases are possible. It should be obvious that as the base material gets thinner, the circuit becomes more flexible.

The metal foil layer provides electrical connectivity for the circuit. While differ-

Multiple nested circuits optimize space on this process panel.



Fillets remove stress-concentration points where a conductor joins with a copper pad. This helps prevent cracking of the copper where the conductor meets the pad.

ent metals may be used, the most common metal found in flexible circuits is copper. Its high malleability, along with good conductivity, makes it an ideal material for flexible applications.

Rolled and annealed (RA) foils are the most common choice, though thinner foils may use electrodeposited (ED) copper.

The bonding-adhesive film, as its name implies, affixes the metal foil layer to the base material, bonds base layers to each other, and also adheres covers to the circuit. As with base films, adhesive films are available in different thicknesses, which are usually determined by the application. For example, different adhesive thicknesses are used in the creation of cover layers to meet the fill-needs demanded by different thickness copper foils. The most-common adhesive films used today are made from a modified acrylic or epoxy base.

When circuits bend or flex, material towards the outside of the bend must stretch to cover the expanded radius, placing that material in tension. Materials inside the bend, however, see the force of compression as the inside-bend circumference shrinks.

At some point in the middle of the material stack is an area that sees little to no tension or compression. This area is called the neutral-bend axis. In a flex circuit, it's loosely defined as an imaginary planar region with no thickness that undergoes neither tension nor compression during bending or flexing. As different layers in the flexible circuit move further away from the neutral-bend axis, the forces of tension and compression become more severe and damaging.

The neutral-bend axis may not be in the exact middle of the material stack. Material composition, thickness, and the amount of area covered by the material (as, say, a copper ground plane in one layer versus normal copper traces in another) can shift the neutral bend axis from the middle of the stack.

The designer must generate a drawing package for many circuit designs. While it is important to completely specify critical features, don't overspecify noncritical features of a drawing. Overspec'ing adds cost but does little to boost robustness.

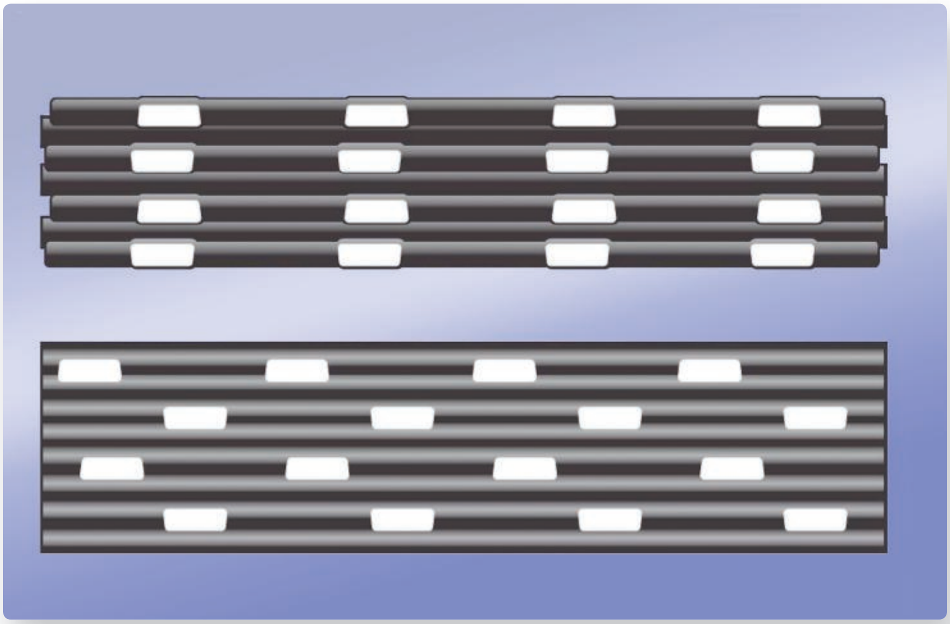
Good drawings have a flat view of the circuit with critical dimensions only. Remember: Electronic data (Gerber, DXF, OBD++) will define every circuit feature. Before a manufacturer can begin the setup, the designer should always compare every dimension on the drawing to the electronic data before beginning setup. The FCB manufacturer and the customer must work out any discrepancies, which routinely happens prior to beginning circuit fabrication. Otherwise, discrepancies found after fabrication begins can easily add weeks to the lead time and often incur additional charges.

Don't specify adhesive thickness on the drawing. The drawing should only specify overall thickness of the circuit and the overall thickness of critical dielectrics that impact impedance.

Every test specification on the drawing has an associated cost that adds to the final circuit cost. Many tests that verify critical electrical features of the circuit, such as continuity and insulation resistance, are standard. When selecting optional tests, however, make sure each test is worth its cost.

WANT MORE?
Focus on this code image using your smartphone and free software from *www.neoreader.com*, and you will be connected to related content on *machinedesign.com*.
http://tinyurl.com/7av5ar6

Stacked conductors, as seen in the upper cutaway view, create an I-beam effect that stiffens the flexible circuit and places additional tension and compression on the conductors and other materials. For maximum flexibility, conductors should be staggered as shown in the lower view.



Whether a flexible circuit functions reliably when bent depends on many design factors, including the bend radii, dielectric type and thickness, foil weight, copper plating, number of layers, circuit thickness, and whether the application is static (bent once for a permanent installation) or dynamic (follows a hinged joint or other moving part).

Tight bend radii boost the probability of failure when the board is flexed. One method used to indicate bend-radius reliability is the bend-radius-to-board-thickness ratio, called the bend ratio. Best practice for reliable operation dictates a bend radius of 10× the board thickness for single and double-sided flex, and 20× the board thickness for multilayers. Radii below these ratios reduce reliability and may make the design questionable.

Flexible circuits bent over 90° are subject to much higher compression and tension force, which boosts the chance of damage. If a circuit must bend beyond 90°, it should be bent once only and then mounted in such a way as to prevent the bend from opening again.

When possible, avoid plated metallic coatings in a bend area. Electrolytic-plated copper is less ductile than rolled annealed copper, and thus has a greater chance of fracturing when bent. Likewise, avoid gold and nickel plating in the bend area for the same reason.

A rigid PCB can incorporate many features that should be avoided on a flexible circuit. Most of these "don'ts" concern features that cause discontinuities in the bending area. The stretching and compressing forces present in the bend area concentrate at discontinuities in material or construction. These higher concentrated forces can lead to fractures in the conductors or insulation.

A good example of a feature that would cause a discontinuity is a via. A via is basically a plated through-hole in the circuit board. If placed in a bend zone, vias weaken the surrounding area when the circuit is bent. This weakness tends to draw the bend towards the center of the hole. The insulation material over the outside of the via is subjected to extreme stretching forces, which can easily result in cover film cracks. If cracks form in the cover material over a via, they will almost always propagate outward from the via and eventually through surrounding conductors.

In addition, it's important not to change conductor width or direction in bend areas. An optimum flex-circuit design maintains a uniform width for all conductors that cross the bend area perpendicular to the bend.

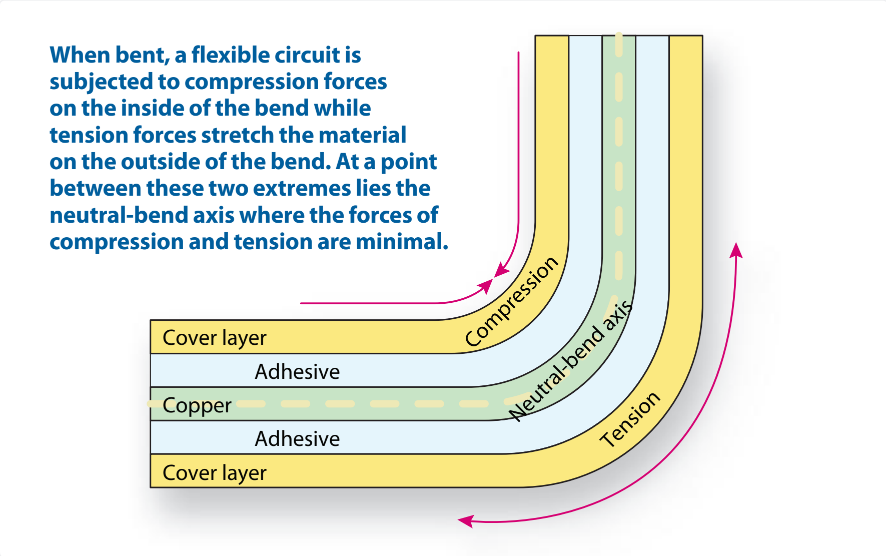
This does not mean that there cannot be 0.010 and 0.030-in. lines side by side. It just means that each line should not change width in the bend area. Also, a change of direction of a conductor in a bend area can potentially cause a stress concentration point and should be avoided.

Don't stack conductors on multiple layer boards. Many board designers like to run signal and return lines on top of each other on adjacent layers to minimize electromagnetic interference (EMI). Doing so makes the overall circuit thicker and creates an "I-beam" effect. If this method of EMI reduction is used, line pairs should be staggered to reduce the effect.

Flexible-circuit materials are reasonably tear resistant. But once a tear starts, it tends to propagate. A sharp inside corner on the circuit outline is a prime location for a tear to start. It can act as a stress concentration point if the circuit flexes in that area.

For that reason, radius all inside corners a minimum of 0.030 in. on the circuit outline. If space allows and the inside radius can be relaxed to a value of 0.060 to 0.075 in., do it. The larger the radius, the lower the possibility of a tear starting in that location.

It's well known that heat softens a flexible circuit and makes it easier to bend. While heat makes forming easier, one must ensure that the heat source does not overstress



the flexible circuit.

A heat gun is capable of extreme temperatures that are well above the maximum temperatures a flex circuit tolerates. It is virtually impossible to regulate the circuit temperature. It is a function of the heat gun setting and the distance the flex circuit is from the nozzle.

Even at a low setting, a heat gun can produce temperatures high enough to blister and delaminate a circuit if it is placed too close to the nozzle. For this reason, one should not use a heat gun to heat form flex circuits.

The recommended method for heat forming is to first cold form the circuits using a customized forming tool and then load the circuits into an oven while still constrained in the tool. Once the tooling and circuits have warmed sufficiently, remove them from the oven and let them cool while still constrained in the tooling.

Use extreme care when removing heated circuits from the oven because they are quite soft and easily damaged. The best temperature to heat-form circuits is the lowest temperature that yields good results. For most circuits, this temperature ranges from 200 to 275°F. Keep in mind that if the circuits see elevated temperatures after they are formed but no longer constrained, they will return to a flat state.

Don't use rigid PCB temperature profiles to reflow flex or rigid-flex PCBs. A reflow oven temperature profile for a rigid PCB will most likely be far beyond what a flex circuit can tolerate. The most obvious signs of excessive temperature exposure will be blistering and delamination. It is advisable to contact a flex-circuit manufacturer that also does assembly work for guidance on proper reflow temperature profiles.

Wide conductors, those greater than 0.010 in., are more robust and tolerate bending better than small conductors. If a bend is pushing the minimum bend ratio limits, it is a good idea to widen small conductors in the bend area.

Because forces from a bend can radiate out beyond the bend zone, the widening should be gradual and the conductor should reach its maximum width at least 0.10 in. before entering and after exiting the bend area.

As previously stated, keep plated through-holes or vias out of areas that bend or flex for several reasons. First, plated through-holes could experience shear forces because of the differential tension/compression between the inside and outside of the bend. This may cause the through-plating of the via to fracture, breaking the electrical circuit.

Second, any hole represents a mechanical discontinuity in the circuit that is prone to cause cracking of the outer cover material. If cracks form in the outer cover, they will almost surely propagate with time and may cause the plated hole to crack and fail.

Flex circuits with multiple layers are thicker and the plated barrels are deeper. The combination of deep-plated barrels with higher stretching and compressing forces from the added thickness aggravates the problem. It is best to place vias in areas that see limited or no bending during installation or service. When the design does not allow sufficient space in nonflexing areas for all plated holes, try to place the holes in areas that experience the least amount of bending.

It is a good practice to place fillets on all termination and via pads. While not every design needs fillets to function reliably, they almost never cause a detrimental effect on the circuit. Fillets used around termination pads eliminate stress concentration points that could otherwise cause cracks where the conductor enters the pad.

As a final word of advice, consult your flex-circuit manufacturer early and often. Every year flexible-circuit manufacturers see hundreds, even thousands, of design options. Start a dialog with a flex-circuit maker during the design process, before the design and drawing are locked in and signed off. Their experience can dramatically improve the odds of finding the most efficient solution to even the most unique design challenges. **MD**