RELIABLE Flexible Circuit Design and Manufacturing

By understanding critical issues related to flexible circuits, engineers and designers can save time and money in their design process.

By Mark Finstad

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

Successful flex circuit designs are soon forgotten as the next challenge crosses the designer's desk. On the other hand, failures stick in a designer's mind for years. In this eBook are tips, tricks, and tidbits learned over 29 years to help ensure a reliable and cost-effective flexible circuit design. It covers what to do and, perhaps more importantly, explains the dynamics and material interactions that occur when a circuit is bent or flexed.

What Is a Flexible Circuit?

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 as their base material due to 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 different 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 electro-deposited (ED) copper.

The bonding adhesive film, as its name implies, can be used to affix the metal foil layer to the base material, bond base layers to each other, and also to adhere 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 in order 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.

The Neutral Bend Axis

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.

A point to remember is that 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, such as a copper ground plane in one layer versus normal copper traces in another layer, can shift the neutral bend axis from the middle of the stack.

Documentation and Specifications

Many circuit designs require the designer to generate a drawing package. While it is important to completely specify critical features, an over-specified drawing typically adds cost to the design but does not necessarily add robustness to the design. Don't over-specify non-critical features on the drawing.

A good drawing will have a flat view of the circuit with critical dimensions only. Remember that the electronic data (Gerber, DXF, OBD++) will define every feature of the circuit. Before a manufacturer can begin the setup, every dimension on the drawing will have to be compared to the electronic data. If there are discrepancies (which routinely happen), they must be worked out between the manufacturer and the customer prior to beginning the circuit fabrication. This can easily add weeks to the lead time and will often incur additional charges.

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

There are several methods for laminating flexible circuits used by flex circuit manufacturers (e.g. hydraulic press, autoclave) that take different processing parameters. As an example, specific combinations often need a different thickness of adhesive to ensure that the conductors are completely encapsulated. A thickness that works well for one flex circuit vendor may be too thick or too thin for another vendor. If adhesive thickness is specified on the drawing, manufacturers will not

Produced by Machine Design Custom Solutions MACHINE Design.com

have the flexibility to use materials that best fit their processing techniques.

The drawing should include any testing needed to ensure that the critical features of the finished circuit are verified. Every test specification on the drawing has a cost associated with it 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. But, while some tests are necessary, many others are not. When selecting test options on the drawing, make sure that test is worth its cost.

Many companies use "boiler plate" notes for every flex circuit application. But in order to make the notes generic and broad enough to work on any flex circuit, they include almost every test that can be done on a flex circuit. While this may save time during drawing generation, it could add significant costs for unnecessary tests.

Every flex circuit manufacturer will lay out as many circuits as possible on a processing panel during the manufacturing process. The processing panel carries the flexible circuits through the manufacturing processes **(Figure 1)**. In many cases, additional assembly such as component mounting will be required after the circuits are received from the flex circuit manufacturer. In these cases, it is desirable for the circuit to be received in a carrier to assist in component placement or for processing by automated pick-and-place equipment. The carrier is called a pallet and may contain one or multiple circuits depending on the circuit and pallet size. The pallet must be sized to accommodate the limitations of the pick-and-place equipment, but at the same time it should have as little impact as possible on the bare circuit manufacturing process.

Most flexible circuits are processed in panel form with different vendors using different processing panel sizes that best fit their processing equipment and techniques. If the designer specifies a pallet size that does not fit well on the manufacturer's processing panel, it can have a significant impact on the number of circuits the manufacturer places on a panel, with direct impact on the final cost of the bare circuit.

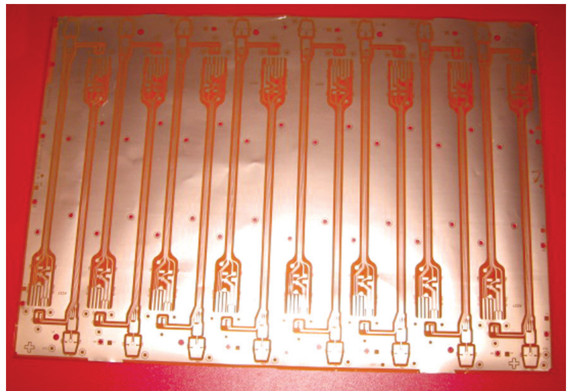

Figure 1: A typical processing panel with circuits nested to optimize space.

For instance, if the number of circuits per processing panel drops by 50%, the circuit cost will almost double. If a manufacturer processes circuits on a 12 x 24" panel, and the designer specifies a 12 x 14" pallet, almost half of the processing panel will be scrap since only one pallet will fit on a processing panel.

Always keep in mind that two of the biggest cost drivers of flexible circuitry are layer count and circuit size. Specifying an awkward pallet size will have the same impact as making the circuit much larger. Some flex circuit designs allow the circuits to be palletized after they are removed from the processing panel. This lets the flex circuit manufacturer optimize the processing panel density and still ship the pallet size desired by the assembler. For this reason, it is important to discuss palletized designs with the flex circuit manufacturer to accommodate all requirements for the lowest possible cost.

Critical Design Areas

Whether a flexible circuit functions reliably when bent does not just depend on the use of flexible materials. Other design factors include 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 of indicating reliability is the bend radius to board thickness ratio (called the bend ratio). A bend radius of 10X the board thickness for single- and double-sided flex, and 20 x the board thickness for multi-layers, creates high reliability for proper operation. Radii below these ratios reduce reliability and may make the design questionable. However, it is possible to reduce the radius below these ratios and still maintain high reliability for specific applications. The flexible circuit manufacturer can help you determine the minimum radius for a given application. In addition, IPC-2223 lists several equations to calculate minimum allowable bend radii for a number of circuit types.

Flexible circuits that are bent beyond 90° are subject to much higher compression and tension force, which increases the chance of damage. If a circuit must be bent 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, designers should avoid plated metallic coatings in a bend area. Electrolytic plated copper is less ductile than rolled annealed copper, and thus possesses a greater chance of fracturing when bent. In tight bend ratio applications, copper plating should be limited to pads-only use. Likewise, avoid gold and nickel plating in the bend region 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 or flexible area concentrate where there is a discontinuity in materials 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 bending 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. In an optimum flex circuit design, all conductors will be of uniform width and will cross the bend area perpendicular to the bend. This does not mean that there cannot be a 0.010" line and a 0.030" line 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.

Another design consideration is not to stack conductors on multiple layers. 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 increases the overall circuit thickness and creates an "I-beam" effect. If this method of EMI reduction is used, line pairs should be staggered to reduce the effect **(Figure 2)**.

There are times when a more rigid board is needed to hold larger or surface-mount technology (SMT) components. The two most common methods of attaining this rigidity is through rigid-flex construction that combines rigid PCB technology with flex circuit technology, or by using multilayer flex with stiffeners. While there are many applications that need rigid-flex construction, there are just as many equally served by multilayer flex with stiffeners — a lesser-cost option.

Rigid-flex construction can easily cost two or three times as much as a comparable multilayer, so it is important to make sure that it is only used when necessary. The most common use of rigid-flex construction is when there are rigid portions of the circuit that have SMT components on both sides of the board. If all SMT

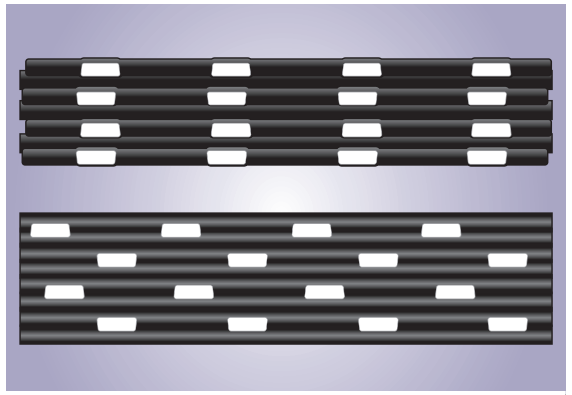

Figure 2: Stacked conductors create an I-beam effect that stiffens the flexible circuit and increases compression and tension stress on the conductors and other materials. For maximum flexibility, conductors through the bend area should be staggered as shown in the lower image.

components are on one side, or there are only through-hole components, such as connectors, a multilayer flex with stiffeners will most likely do the job at a lower cost.

Most stiffeners are commonly FR-4 or polyimide materials applied with a modified acrylic or epoxy adhesive. When using multiple stiffeners on a single flexible circuit, it's best to keep all stiffeners an identical thickness to keep costs low.

Stiffeners can be used for a variety of purposes such as to rigidify pin areas, SMT areas, or hole patterns. While some SMT components may not need a stiffener, adding one is recommended and adds very little to the cost or bulk. They also help reinforce solder joints and boost abrasion resistance.

If in doubt as to whether an application should be rigid-flex or multilayer flex, it is advisable to contact a flex circuit manufacturer for guidance prior to starting the design.

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 since it can act as a stress concentration point if the circuit is flexed in that area. For that reason, it is advisable to radius all inside corners a minimum of 0.030" on the circuit outline. If space allows and the inside radius can be increased to a value of 0.060" to 0.075", do it. The larger the radius, the less chance there will be of a tear starting in that location.

Assembly

It is a well-known fact that heat softens a flexible circuit and makes it easier to bend. While heat makes forming easier, it is important to ensure that the heat source does not over-stress the flexible circuit. A heat gun is capable of extreme temperatures that are well above the maximum temperatures tolerated by a flex circuit. It is virtually impossible to regulate the circuit temperature since 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, it is not advisable to use a heat gun to heat form flex circuits.

The recommended method for heat-forming is to first cold-form the circuits with customized tooling and then load the circuits into an oven while still constrained in the forming tooling. Once the tooling and circuits have reached the necessary temperature, they should be removed from the oven and cooled while still constrained in the tooling. Use extreme care when removing the hot circuits from the oven because they will be very soft and can be easily damaged. The best temperature to heat form circuits is the lowest temperature that will yield good results. For most circuits, this temperature will be between 200° F – 275°F. Keep in mind that if the circuits are exposed to elevated temperatures after they are formed but no longer constrained, they will return to a flat state.

As stated previously, most flexible circuits are constructed using layers of polyimide film dielectric and copper foil bonded with a modified acrylic or epoxy adhesive film. These materials provide the properties that let the circuit bend and flex while also providing electrical isolation between layers. The shortcoming of this construction is high moisture-absorbing properties and low tolerance to elevated temperatures. While the polyimide film can tolerate very high temperatures, acrylic and epoxy adhesive have service temperature of only 125° C.

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.

Low tolerance to elevated temperatures can be further aggravated by trapped moisture. Flexible circuits are extremely hygroscopic, meaning they absorb moisture well, and can saturate in well under an hour if exposed to high humidity. To make matters worse, it can take many hours of baking to drive all the moisture from a thick circuit. When flex circuits with high moisture content are exposed to reflow temperatures, the moisture in the circuit turns to gas and rapidly expands. This can easily cause massive delamination and blistering. For this reason, it is imperative to thoroughly bake flex circuits immediately prior to reflow. If the circuits are baked more than 30 min prior to reflow, they should be stored in a sealed desiccant box.

When designing flexible circuits, don't hesitate to call a flex-circuit manufacturer during the design process. Each year, flex circuit manufacturers see hundreds or thousands of new flex designs. This experience gives the company a unique insight into what does and does not work. Tapping this knowledge base gives the designer a better chance of incorporating the right design features by not incorporating the wrong ones.

Routing Conductors

As mentioned earlier, when a circuit is bent or flexed, the materials are subjected to tension and compression forces.

Figure 3 shows a cross section of a typical single-sided circuit depicting the different forces and the neutral bend axis (dashed line). The neutral bend axis will not always fall in the exact center of the material stack. It tends to shift toward materials that offer a greater resistance to stretching and compressing than other materials in the stack. Copper plane layers, heavy copper conductors, and thick (>0.003") polyimide layers are typical materials that offer greater resistance to stretching and compressing. These thicker materials can have a profound impact on the way a circuit behaves during a bending operation.

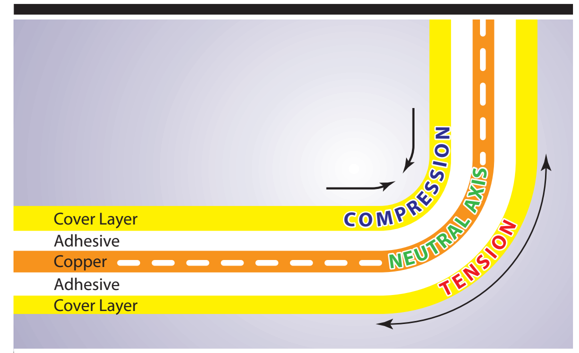

Figure 3: When bent, a flexible circuit is subjected to compression forces on the inside of the bend, while tension stretches the materials on the outside of the bend. At a point between the two extremes lies the neutral bend axis, where compression and tension forces are minimal.

For example, consider two double-sided flex circuits of identical size and shape. One of the flex circuits has 0.007" conductor traces on both layers and 0.001" polyimide film for the base dielectric and cover material. The other circuit has 0.007" lines on one layer and a heavy solid ground plane on the other layer. The first circuit has a balanced or symmetrical construction, so the neutral bend axis will land very close to the middle of the stack. This circuit behaves the same way regardless of its bend direction.

However, on the second circuit the neutral bend axis shifts toward the heavy copper plane because it offers greater resistance to stretching and compressing than the other materials in the stack. This results in the 0.007" conductors being subjected to exaggerated stretching or compressing forces when they are bent. Also, the second circuit can tolerate a tighter bend radius when the copper plane is to the outside of the bend than it could if the plane were to the inside of the bend.

Wide conductors (>0.010") 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" before entering and after exiting the bend area.

Conductor routing guidelines have been created to take into account the unique mechanical characteristics of flexible circuitry. Deviating from these guidelines, and those of IPC-2223, may produce circuit designs of questionable reliability.

Vias and Fillets

As previously stated, plated through-holes or vias should be placed in areas of the circuit that will not be bent or flexed. There are several reasons to keep vias away from bend areas. First, plated through-holes could experience shear forces due to the differential tension/compression between the inside and outside of the bend depending on where it fell in relation to the center 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 cracking 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 will be thicker and the plated barrels will be deeper. The combination of deeper plated barrels with higher stretching and compressing forces due to the added thickness only 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 non-flexing areas for all plated holes, place the holes in areas that experience the least amount of bending.

It is good practice to place fillets on all termination and via pads. **(Figure 4)** While not every design requires fillets to function reliably, they almost never cause a detrimental effect on the circuit. When fillets are used around termination pads, they eliminate stress concentration points that could otherwise cause cracks where the conductor enters the pad.

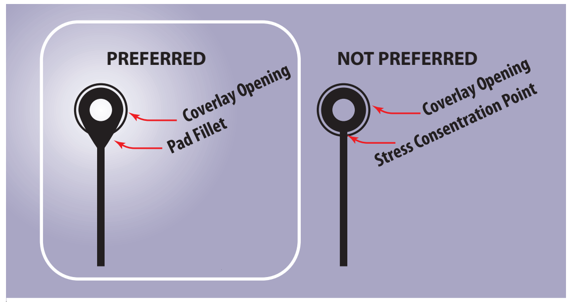

Figure 4: Fillets remove stress concentration points where a conductor joins with the copper pad. This helps prevent cracking of the copper where the conductor meets the pad.

Copper Plane Placement

When possible, place copper planes on or near the center layer of the circuit. As noted earlier, copper planes resist stretching and compressing and can cause a positional shift in the neutral bend axis. By placing the copper plane near the center layer, the associated shifting effects on the neutral bend axis are minimized. In some instances, the design does not allow the planes to be positioned near the center of the stack. When this is the case, plane layers should be balanced as much as possible on opposing sides of the neutral axis. The effects from the planes will tend to cancel each other out, resulting in the least amount of neutral axis shift.

A single copper plane that cannot be placed near the center layer should be positioned such that it is to the outside of the neutral bend axis on the majority of the bends. Copper conductors can tolerate compressing better than stretching. By placing the plane to the outside of all or a majority of the bends, the conductors in those areas will more likely be compressed rather than stretched.

Another trick that reduces stress caused by copper planes is to either cross-hatch the copper plane or use a conductive coating such as silver epoxy instead of copper for the plane. Cross-hatching the copper plane reduces the amount the plane resists stretching and compressing and makes it act similar to a standard conductor layer. Silver epoxy is the most common conductive coating used in the flex circuit industry. The electrical performance of a silver epoxy plane is similar to a solid copper plane, but it is more flexible. In addition, there are some relatively new flexible conductive films that can be utilized to improve overall flexibility while maintaining the necessary electrical properties.

Each of these stress-reducing options has drawbacks. Cross-hatching a plane has significant impact on the impedance of any conductor using it as a return path. Conductor widths and dielectric thickness also need adjustment to maintain the needed impedance.

The drawback to using a silver epoxy plane layer is limited to cost. Incorporating a silver epoxy layer into a flex design adds extra steps to the manufacturer, and those costs will be passed on to the end user.

Drawing Generation

The final segment of this eBook covers the steps needed to obtain prototypes of production circuits. This should help designers have a better understanding of the process, beginning with generating a complex drawing through the procurement stage.

Drawings should include a flat view of the flex circuit with at least a few reference dimensions. Keep in mind that because a flex circuit is not a rigid structure, it can be moved slightly to fit into the available space. For this reason, very few dimensions are truly critical to form and fit.

In addition to the flat view, good drawings include a drill chart indicating the different hole locations, sizes, and whether the holes are plated or unplated. Different symbols are usually assigned to each hole size on both the flat view and the drill charts.

A cross section of the circuit **(Figure 5)** is also a critical part of a good flex circuit drawing. All material types should be either clearly labeled in the cross section or referenced to a drawing note. Material specifications should use standard IPC designations. It is usually a good idea to thoroughly specify material type but stop short of specifying adhesive thickness. The overall circuit thickness should also be specified in the cross section portion of the drawing. It is important to ensure that the maximum circuit thickness specified can be achieved before placing a thickness dimension on the drawing.

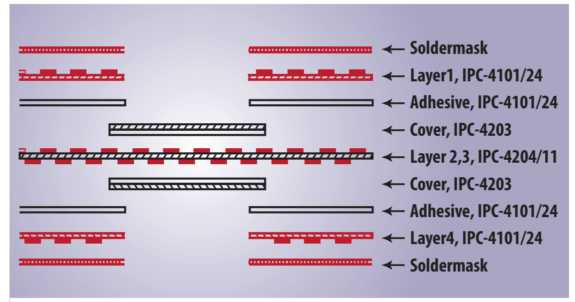

Figure 5: A well-defined cross section similar to the one above should be included on the fabrication drawing.

The drawing should also contain a complete but not over-specified set of notes. Examples of features covered in the notes section would be final surface finish (e.g., solder, ENIG, OSP), specifications to which the circuit must conform (e.g., IPC-6013), testing requirements, and material specifications. It is important to not over-specify the circuit with the notes section. By stating in the notes section that the circuit must comply with the requirements of IPC-6013 and the performance class, most of the testing requirements and other performance-related features are covered.

The notes section should only cover requirements that are either unique to that circuit or deviate from IPC-6013. If a feature is critical to the performance or reliability of the circuit, it should by all means be specified on the drawing. But placing unnecessary requirements on a drawing that does not enhance the function or reliability of the circuit only adds cost.

Electronic Data

It is extremely important to supply a complete electronic data package — in the proper format — to the potential vendor at the quoting stage to ensure you receive the best price. If the electronic data package is missing, incomplete, or is sent in an incompatible format, the vendor has to assume worst-case conditions for all unknowns and will quote accordingly.

The most common data formats used by circuit vendors today are Gerber and DXF. These formats can be easily converted to virtually all CAD and CAM software used by circuit manufacturers. OBD++ is also gaining popularity with a majority of flex circuit vendors. Formats with extensions such as PDF, BMP, JPG, and TIF are often mistakenly sent as "CAD data." These files can show some features of a circuit but are not true CAD data. The aforementioned files only offer a picture of the circuit and are difficult, if not impossible, for the manufacturer to use in the fabrication of the circuit. It is a good idea to check with prospective vendors prior to sending data to make sure they can read what you are going to send.

In a complete electronic data package, the information should be spread out over multiple CAD layers. At a minimum, there should be one CAD layer for each of the following features:

- Each individual circuit layer;
- Drill layer;
- Circuit border;
- Outer layer cover access definition (flex) or SMOBC (rigid flex);
- Marking and legends;
- Stiffener outlines; and
- Fiducials.

If the circuits will have further processing performed on them, such as component mounting, they will most likely be received from the circuit vendor in pallet form.

The definition of the pallet **(Figure 6)**, and the position of each circuit within the pallet, should be included in both the electronic data package and on the drawing. As mentioned earlier, you should contact your circuit vendor prior to defining the pallet size to ensure that the size specified does not significantly reduce the number of parts the vendor can place on one panel. If the manufacturing panel density is reduced, circuit cost goes up proportionately.

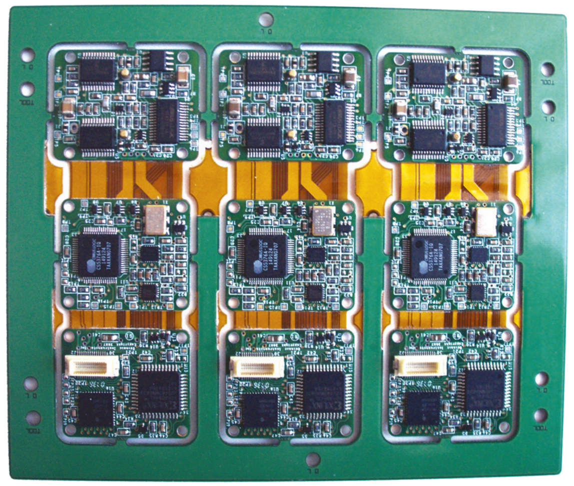

Figure 6: A typical pallet holding multiple circuits as they are processed through pick and place and reflow.

Circuit Class Ratings

Flexible circuits are assigned class ratings based upon their end use. Class 1 circuits are the lowest cost and also have the least stringent quality requirements. They can be found in products such as musical greeting cards, where a failure is mostly an inconvenience.

Class 2 circuits are a step up from Class 1 in price and quality requirements. Class 2 flex circuits would most likely be found in laptop computers, medical diagnostic equipment, and automotive applications.

Class 3 circuits are by far the most expensive flex circuits and have the most stringent quality requirements. Class 3 circuits are used where a circuit failure could very well mean the difference between life and death. Class 3 circuits would be found on military and commercial aircraft, guided munitions, and implantable medical devices.

The flex circuit designer and buyer have to determine the performance class required for the specific application, and then choose a flex vendor that best fits the need.

Vendor Quotes

Most flex circuit vendors have a niche market to which they cater, and matching the vendor's niche market with your application will most likely get you the best part at the best price. With a quick phone call to the vendor, you can usually determine if a vendor's niche fits your application. You will want to inquire about the company's volume capability. Many flex vendors are strictly prototype shops, whereas others do only medium- to high-volume jobs. It's also a good idea to get the vendor's preferred minimum and maximum volume and see if your design fits into the vendor's "sweet spot" from a capacity viewpoint.

The vendor should be asked about maximum and preferred layer count, minimum through-hole size, and plating aspect ratio capabilities. While you have the vendor on the phone, ask what industry the company serves most. For instance, a flex circuit vendor that builds millions of dollars worth of flex circuits for the military or medical industries will have a much better understanding of the unique quality requirements encountered in those industries. Although a flex vendor that caters to the military/medical (high reliability) industry will be able to easily manufacture a commercial circuit, it will almost always be much more expensive than a flex vendor that caters to the commercial market.

Vendor Quote Feedback

Often after reviewing a customer drawing and data package, the circuit vendor has questions or suggestions to improve either reliability or reduce cost. Designs that are bound for a military or medical end use are very difficult and time-consuming to change once the design is qualified and signed off. To incorporate cost saving and reliability suggestions from a vendor into your design, you will need that advice prior to the design qualification.

Most flex circuit vendors provide a cursory or detailed review of your application at no cost. It is important to also give the vendor details of the end use such as bend locations and bend radii, operating temperatures, and chemicals the circuit will contact in service. With this information, your flex vendor can provide valuable insight to help make your flex circuit project successful.

If you take one item away from this eBook, it should be to consult your flex circuit manufacturer early and often. Every year vendors may see hundreds, even thousands, of design options. It will help you immensely to start a dialog with a vendor during the design process, before the design and drawing are locked in and signed off. When you rely on the vendor's experience, you can dramatically improve your odds of finding the most efficient solution to even the most unique design challenges. ■

For more information on flexible circuits, check out additional links at www.flexiblecircuit.com/products/value-added/links.php

Glossary

Access hole
A hole in the cover layer that provides access to the conductor pads and through-holes.

Annular ring
The exposed copper ring that surrounds a flex-circuit through-hole.

Conductor
The electrical path that carries current from one component to another.

Conductor spacing
The width of the insulating gap between conductors. A minimum conductor spacing prevents electrical shorts between conductors.

Conductor width
The width of the conductor measured across its base.

Cover
An insulating material that covers the etched conductors on inner or outer layers. Internal covers are typically used in unbonded circuit areas for greater flexibility.

Dynamic application
Flex circuits used where flexing is a requirement of the application.

Flex circuit
Flexible printed circuits made from etched foil conductors laminated between flexible insulating layers.

Hold-down tabs
An extended foil tab that helps hold the pad against the substrate insulation. Hold down tabs are also called “anchor spurs.”

I-beam effect
Conductor strands layered over one another create a stiffening effect reducing flexibility. Strands should be staggered between layers.

Major access hole
An access hole that exposes a major portion of a conductor pad, typically solder coated for component mounting.

Minor access hole
An access hole that exposes a small portion of the conductor pad where solder is not needed nor wanted.

Nesting
A method of creating the greatest number of circuits on a panel for production that minimizes production cost.

Pad
The portion of a conductor that surrounds a through-hole used to mount components or as an electrical connection. Sometimes referred to as terminals or lands.

Profile tolerance
The tolerance of the dimension between two parallel lines separated by a specified tolerance. For example, a steel rule die must trim the circuit between two parallel lines spaced 0.015” apart. The true cut could occur anywhere within the zone marked by the parallel lines.

Rigid-flex
A circuit that contains both flexible and rigid circuit board components.

Selective plating
Plating of the flex circuit such that only selective areas receive the plating process. By keeping the plating off bent portions, flexibility and durability of the bend are improved.

Static application
A flex circuit used in a situation where the bending of the flex circuit occurs only during installation and maintenance, and then it remains fixed in place during service.

Stiffener
Rigid material (typically polyimide or FR-4) added to a flex circuit to reinforce areas for mounting components. Unlike rigid-flex, stiffeners do not carry any conductors.

Substrate
An insulating layer upon which foil is bonded to one or both sides.

Tangency
A misalignment of the cover access hole with a through-hole such that the through-hole approaches the edge (tangent to) of the cover hole.

Tear stops
Small relief holes of copper, polyimide, or Teflon guards usually in inside corners that help prevent propagation of tears in flexible circuits.

Trim line
The final cut-out area around a flex circuit.

Unbonded area
An area without adhesive bond between conductive layers of a flex circuit used to improve flexibility.

Via
A plated-through-hole with no cover access hole that connects internal layers.