



Flexible Circuit, Rigid Flex and Printed Circuit Boards

Design Tips for Flexible Circuits 01.10

Flex Circuits vs. Traditional PCBs

Flex Circuit design vs. PCB

Flex circuits still only account for about 15% of the overall printed circuit market. So it's understandable that there are still questions about the design of a flex circuit vs. traditional printed circuits boards (PCBs). PCBs have been in existence since the early 1960s and there are many experienced PCB designers working confidently around the world. However we frequently are queried about the nuances in design for flex.

Designers are warned by management, project engineers etc... That the design of a flex circuit is different and is asked to consult with the fabricator or in some cases simply want to offload the design responsibilities to the supplier.

It's not that difficult and adding this knowledge to your experience tool box isn't as daunting a task as it may seem.

Similarities and differences

PCB design rules are well known. There are minimum hole sizes, minimum trace width, minimum space between traces and pads. We know to keep copper geometries a certain distance away from the routed edges. We know about hole size and outline tolerances, copper and board thicknesses and myriad other specifications that we don't even think about any more because it has become an inherent part of our thought process.

Flex circuit design rules are *very similar!* We must pay attention to all of the same issues, minimum hole sizes, minimum trace/space specifications, distance to edge and tolerances.

First let's discuss the fabrication process. Traditional PCBs and flex are fabricated in much the same way for the first several steps. The flex material, typically copper clad polyimide, is allocated, drilled, plated, imaged, developed and etched just as printed circuit boards are. The next step however is where the changes occur. The panels must be baked as are PCBs to remove moisture from the wet processes, then however where a PCB would go to a solder mask station, flex circuits go to a *cover layer* station.

The insulating layer of a flex circuit is made of polyimide the majority of the time. This is not a screen process as it is in PCB fabrication, it is a lamination process. Therefore the rules for oversize of openings are a little different.

The *cover layer* material is made of 1 mil thick polyimide with a 1 mil thick adhesive attached. The material is drilled to create the openings and so the tolerances of drill location and size apply.

Also during the lamination process the adhesive is heated to a temperature that allows the adhesive to flow easily. It must fill in all of the gaps between traces and pads so that there is no air trapped between layers. What this means to a designer is that the *cover layer* openings for a typical 1 oz copper design needs to be oversized by .010" (10 mils). This is significantly larger than the typical PCB solder mask which oversize's 2-3 mils. The reason is first the tolerance of drill size and location, but also to account for the adhesive which squeezes out into the openings. We want to design such that the adhesive flows out to the pad, but not on top of the pad. That would affect the size of the annular ring. We call this phenomenon squeeze out and we want it to dam up on the thickness of the copper pad and not flow over the *dam*.

Another area we must worry about that isn't common with PCBs is the "flex area". Some reasons for using flexible circuits are simply size and weight, but many of the applications take advantage of the flexibility and use the circuit in this manner. There are two typical ways this is done. Flex to fit: The circuit is flexed once only to fit into the assembly and Dynamic flex: This circuit will not only flex to fit into the assembly, but will be dynamic during operation. Either scenario requires a bit of thought be put into the trace routing and pad placement in that area. First it's best not to have any components (solder pads) in this area. Whether the flex is formed once for fit or dynamic, the solder joint still will be the weakest part of the circuit. Solder, RoHS or leaded, is rigid and not intended to bend, flex or twist. Therefore if those joints are in the flex area you will likely see fractured solder joint at some point. It's best to keep all solder points at least .100" (100 mils) away from flex areas. Further is better if real estate allows.

Traces should route through the flex areas perpendicularly. This allows us to take advantage of the malleability of the copper. Rolled annealed copper has grain and if run horizontally in the flex area, may split or fracture leaving the engineers an intermittent issue to try to find which is very frustrating and difficult to identify. It's best to route traces at a 90 degree angle through the flex area and then make the necessary direction changes to allow completed design.

The only other big change to keep in mind is that angles in flex are not preferred. So the miter tool on your design software should be set to round instead of angular when designing a flexible circuit. The natural flexibility affects the transition from horizontal to vertical trace routing so we like to make that transition as seamless as possible. Rounded corners give us that luxury.

Conclusions

So as you can see the design tactics for Flexible Circuits don't differ much from traditional PCBs. All of the typical specifications still apply and we add a few more things that require special attention. Cover layers require bigger openings than traditional solder mask, trace directions matter in the "flex areas" and miters should be round instead of angular. You may use all of the same CAD tools for design and output Gerbers as you would for any PCB. The documentation is all the same also with fabrication and assembly drawings being the norm.

So go ahead and design your next flex circuit with the same confidence you have with PCBs. Of course if questions still arise you may always count on your supplier for quick and accurate support.