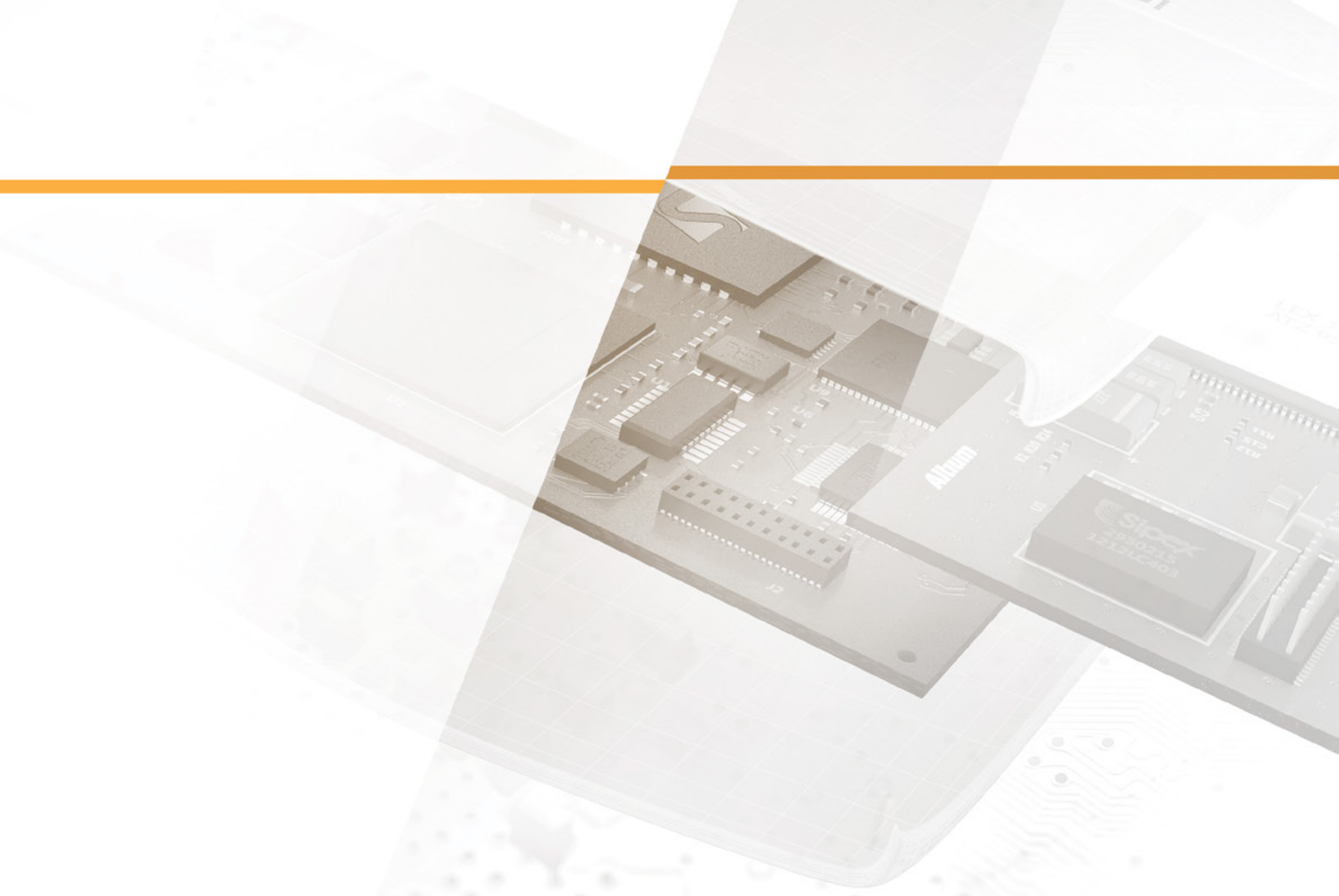
Altium



An Introduction to Rigid-Flex PCB Design Best Practices

Golden Rules for First Time Success in Rigid-Flex

An Introduction to Rigid-Flex PCB Design Best Practices

More designers increasingly face project requirements for densely populated electronic circuits including pressures to reduce manufacturing times and costs. To meet these requirements, design teams have increasingly turned to 3D rigid-flex circuits to meet their project's performance and production requirements. As a first step, rigid-flex designs require closer collaboration between the designer and fabricator than traditional board-and-cable designs. The trade-offs required to produce a successful rigid-flex design translate to a set of design rules the designer can develop with the fabricator's input. These considerations include the number of layers in the design, materials selections, recommended sizes for traces and vias, adhesion methods, and dimensional control.

In the past, seasoned PCB designers bypassed rigid-flex circuit design by connecting two rigid PCBs with a flexible cable. This approach worked well for short-run designs. However, this approach adds the cost of connectors on each board, the cost of assembling the connectors to the board, and the flexible cable. In Figure 1 below, the chart maps the cost trade-off between a traditional rigid PCB-and-cable assembly and a 3D rigid-flex design.

This chart relies on simulated manufacturing costs based on real PCB fab quotes for a four-layer PCB with two inner flex layers in a rigid-flex board. The alternative, rigid boards using flexible cable and connectors between them, is also based on quotes from PCB fabricators. In the latter case, the calculation totals the costs of two separate four-layer boards plus the costs of connectors and cable including assembly costs for both alternatives. This calculated "what-if" scenario does not take into account the improvements in reliability and overall higher product quality of the rigid-flex circuit. Among other reasons, the individual boards with flexible connectors and cable can form electrically 'cold' joints, causing malfunction. By comparison, the rigid-flex circuit obviously eliminates these joints.

Surprisingly, as soon as the project involves any volume over 100 units, a rigid-flex circuit design becomes the obvious choice. Why? Because rigid-flex designs eliminate connectors, connector assembly, and increase reliability and process yields.

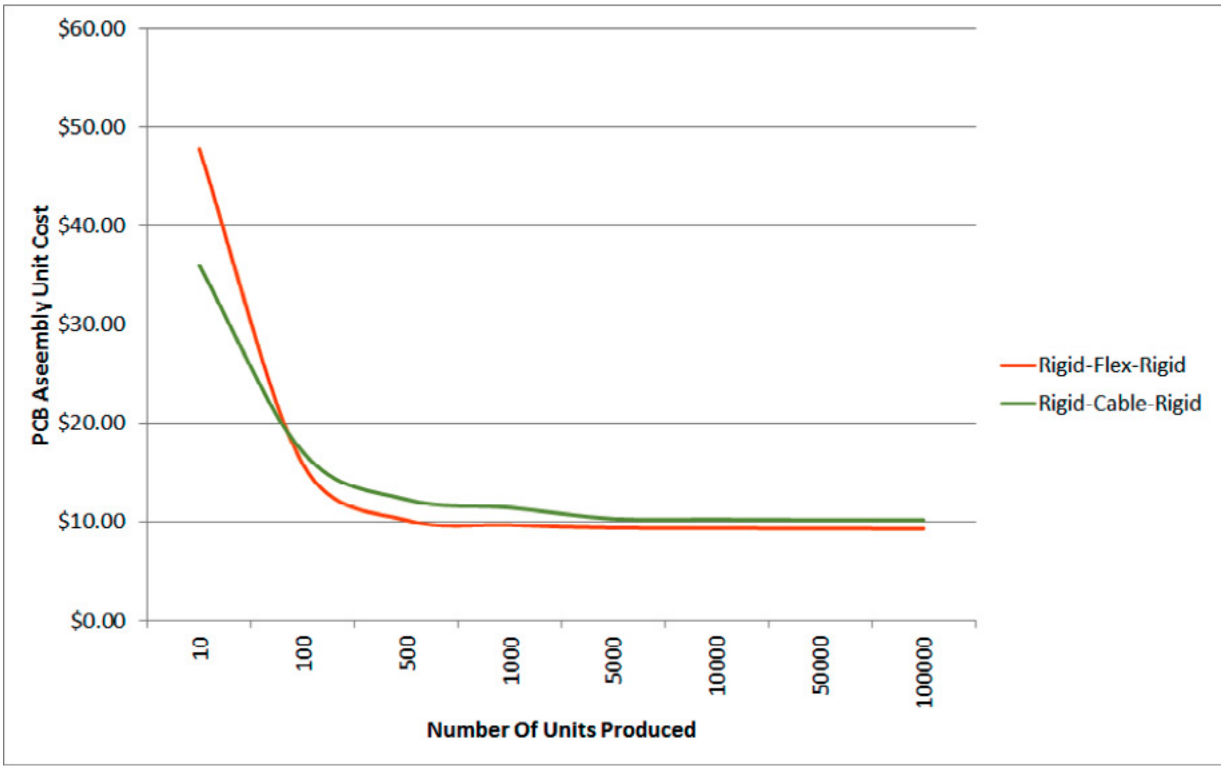


Figure 1: Quote-generated cost comparison of Rigid-Flex design versus Rigid-Cable-Rigid PCB assembly.

Until recently, when a traditional PCB designer first tackled a rigid-flex design, the available PCB design tools did not support board design in 3D. Existing tools also did not support defining and simulating bends and folds in the flex portion of the design. Worse yet, they didn't even support defining different layer stacks in different parts of the design or even what areas constituted the flex part of the design.

As a result, rigid-flex designers had to manually determine how to translate a 3D design of both the rigid and flexible sections into a flat, 2-D representation suitable for fabrication. This further required them to manually document all areas that were flexible. They also had to double-check that they did not place components or vias near the transitions between rigid and flexible portions of the design. This also included many additional rules which were mostly not supported in the PCB design software at the time.

Pre-Design Considerations

As suggested earlier, any successful rigid-flex design requires that the design team work closely with the fabricator. The following "Golden Rules" for the project aim for successful production and eliminating preventable respins.

Golden Rules

- Communicate with the fabricator!
- Qualify the fabricator's capability to build the planned rigid-flex design.
- Involve the fabricator as early as possible in the design process.
- Collaborate so the design's layer stack matches the fabricator's processes.
- Use IPC-2223 as the common point of reference with the fabricator. Otherwise, communication in the form of documentation can cause errors and misunderstandings resulting in costly delays.
- Graduate from delivering Gerber files to the fabricator. Instead, deliver files in ODB++ (v7.0 or later) or in a format that meets IPC-2581 because either format identifies specific layer types for clear documentation. [5]

For a successful rigid-flex design, the design team must also select the materials that balance, as always, cost versus performance. Most conventional PCB boards start with a rigid fiberglass/epoxy substrate. Although termed "rigid", fiberglass/epoxy substrates exhibit some flexibility, but not enough for more complex applications that involve movement.[3]

For 3D rigid-flex designs, dimensionally stable, flexible and heat-resistant polyimide (PI) film is the most common choice. It remains reasonably stable due to low thermal expansion and contraction (relative to PET) while also tolerating multiple reflow cycles. Polyester (PET), also commonly used, but does not tolerate high temperatures well and is less dimensionally stable than PI.[3] As well, thin fiberglass/epoxy cores also find application in rigid-flex circuits. Besides substrates, the design will require additional films (usually PI or PET, but sometimes flexible solder mask ink) for coverlay. The coverlay protects the outer surface components and conductors from damage and corrosion and insulates the conductors as well.

By definition, rigid-flex circuits impose additional requirements when selecting conductor materials. Electrodeposited (ED) copper foils used in traditional PCB designs fall short of the necessary flexibility and toughness properties needed in a rigid-flex design. Rigid-flex designs utilize a variety of higher-performance conductor materials and methods. However, the two most common are medium-priced high-ductility electrodeposited (HD-ED) and higher-cost rolled-annealed (RA) copper foils. [3]

In the early stages of the process, the rigid-flex design team faces a balancing act. They must define the mechanical challenges of the projected use cases balanced against the electrical performance requirements. These two considerations often butt heads, requiring the designer to balance and resolve the two. As a first step to arriving at the optimal design, the design team can gain considerable insight by producing a physical "paper-doll" mock-up of the circuit. The mock-up pinpoints potential form and fit problems early in the design process. As modern eCAD tools progress, they include 3D modeling of rigid-flex designs. The most up-to-date add animation. However, developing a 3D computer model involves considerable design steps, so an initial paper mock-up still proves to be informative.

Rigid-Flex Design: Best Practices

The term "rigid-flex" points to one of the most significant design details. Rigid-flex circuit designs involve multiple elements that, when combined, result in a high level of complexity. For designers developing a rigid-flex design, the biggest challenge remains: "How do I define all the areas, layers, and stacks?" The answer: Use a table to define the stack layer design. As a general characteristic, most rigid-flex designs exhibit different layer stacks in different areas of the design.

One simple way: Copy the design outline on a mechanical layer. Then create a fill-pattern identifies the rigid and flexible portions of the design that contain a different layer stack as shown in Figure 2.

Layer	Name	Material	Thickness	Constant	Flexible	Rigid
1	Top Paste					
2	Top Overlay					
3	Top Solder	Coverlay (PI)	0.50mil	4		
4	Top Layer	Copper	1.40mil			
5	PIBase	Polyimide	1.40mil	4.8		
6	Dielectric 1	FR-4	12.00mil	4.2		
7	Bottom Layer	Copper	1.40mil			
8	Bottom Solder	Coverlay (PI)	0.50mil	4		
9	Bottom Overlay					
10	Bottom Paste					

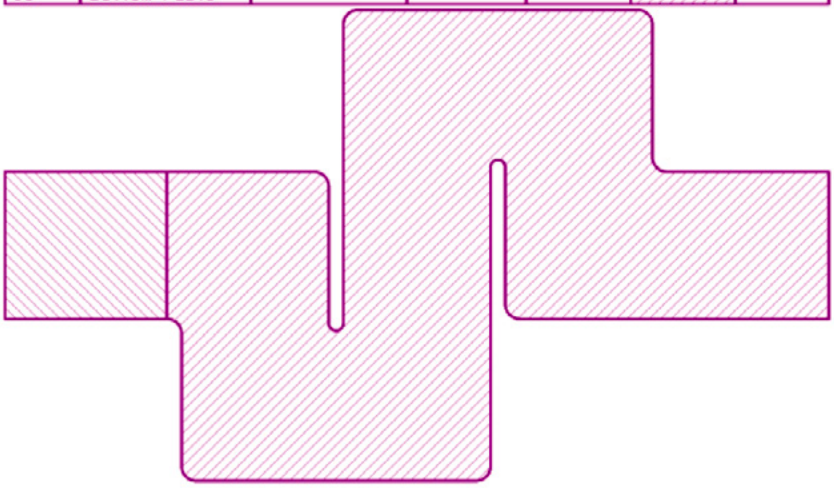


Fig. 2: Fill patterns in the "Flexible" and "Rigid" columns in the table identify the rigid and flexible areas of the board. [3]

The simplified design in Figure 2 above uses the matching graphic fill patterns (the two columns on the right of the table) to identify the "Flexible" and "Rigid" areas of the board, respectively. For example, the layer named "Dielectric 1" is an FR-4 core. With the different layer stacks defined, any rigid-flex design team now confronts specifying bends and folds in 2-D space. This means that every rigid-flex designer must document where all critical design elements cross the boundaries between rigid and flexible sections.

Keeping the "Flex" in Rigid-Flex: Staggered Length Circuits

This design practice, also called "bookbinder construction", adds a small amount of length layer-by-layer, moving outward from the bend radius. Using this method means that the circuit can only bend in one direction. A common rule of thumb is to add additional length to a subsequent layer roughly 1.5 times the thickness of the individual layer. But that value varies depending on the tightness of the bend and the number of layers. This is another instance where a paper-doll mock-up can provide an informative quick check. Bookbinder construction relieves tension produced during bending and also prevents buckling of the inner layers near the bend radius.

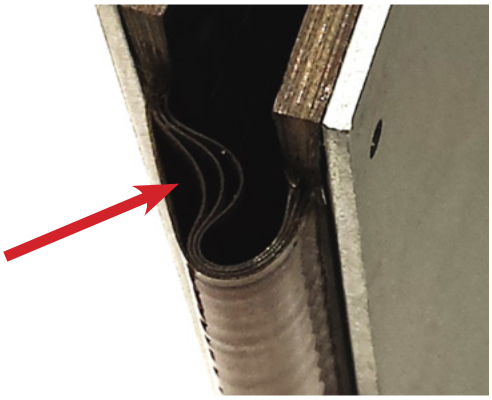


Fig. 3: In this commercially produced design, the designers obviously did not apply "bookbinder construction", resulting in additional stress on the separated layers (red arrow). [7]

More Best Practices [6]

- Avoid Bending at the Corners. Copper traces perform best when placed at right angles to the flexible circuit bend. In cases where bending is unavoidable, one alternative is to use conical radius bends. [6]
- Use Curved Traces. Avoid hard right angle traces and even 45° hard corners because they increase stresses on copper traces during bending. [6]
- Do Not Abruptly Change Trace Widths. When traces in a design enter a pad, often in alignment, an abrupt change in trace width creates a weak spot. As a good design practice, use a teardrop pattern to gradually change the width of traces connecting to pads and vias in the flex circuits. [6]
- Use Hatched polygons. Using a normal rectangular pour still retains heavily biased stresses in 0°, 45°, and 90° directions. A hexagon by comparison statistically provides a more optimal hatch pattern. [6]
- Add Pad Support. Compared to FR-4, copper on a flexible PI substrate is more likely to detach due to repeated stresses involved in bending plus lower adhesion. Consider surface mount pads and non-plated through holes to be unsupported. Many fabricators recommend additional through-hole plating and recommend additional SMT pad support such as anchoring stubs and reduced coverlay openings. [6]
- Stagger Double-Sided Flex Traces. Running traces over each other in the same direction distributes tensions between the copper layers unevenly. Staggering the traces reduces or eliminates the problem. [6]

Conclusion

In harmony with the clear trend of increasing rigid-flex PCB manufacturing, updated PCB CAD tools now include the necessary features needed to design rigid-flex circuits. These include multiple layer stack management, components mounted on "inner" flex-circuit layers, and 3D visualization and simulation of the flex circuit portions. Figure 4 below, an example of a rigid-flex PCB design produced in Altium Designer 14, demonstrates these capabilities.

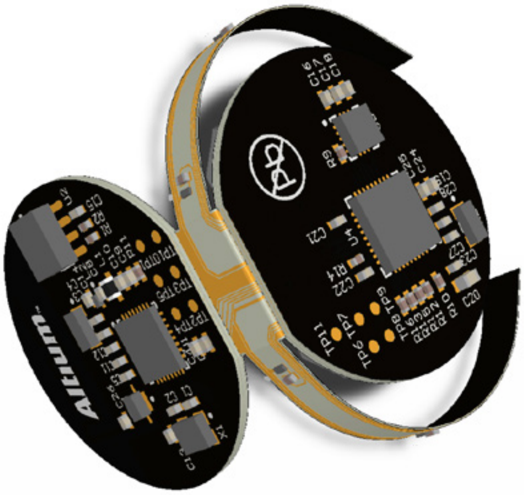


Fig. 4: An example rigid-flex PCB design in Altium Designer's 3D mode, showing intended bends in the flex-circuit areas. [7]

In addition to the layer stack management and 3D visualization, clearance checking of the components on the flex substrate is also possible. As part of the built-in Design Rule Check engine, the new release provides early warning for rigid-flex designs whenever the final bend radii results in mechanical interferences. As an example, see Figure 5 below.

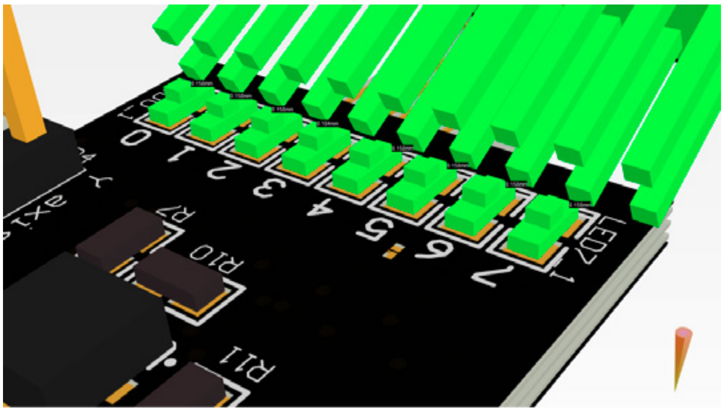


Fig. 5: Simulated bending of a flex circuit reveals a clearance violation between a flex-mounted pin-header and SMDs on the main rigid board. The violating components are highlighted green. [7]

To learn how Altium Designer 14 can help you develop and implement rigid-flex circuit designs, at www.altium.com

Sources & Additional Reading

1. Clyde F. Coombs, Jr (Ed.), "Flexible Circuit Applications and Materials," Chapter 61 in Printed Circuits Handbook, Sixth Edition, 2008 McGraw Hill.
2. Joseph Fjelstad, Flexible Circuit Technology, p. 186, Fourth Edition, 2011 BR Publishing.
3. Benjamin Jordan, Ruminating Rigid-Flex - Part 1, "Flex Circuit Materials," Altium Live Blog, July 13, 2013, http://blog.live.altium.com/Blogs/ruminating-rigid-flex-pt1
4. Benjamin Jordan, Ruminating Rigid-Flex - Part 2, "Flex & Rigid-Flex Fabrication Processes," Altium Live Blog, August 15, 2013, http://blog.live.altium.com/Blogs/ruminating-rigid-flex-pt2
5. Benjamin Jordan, Ruminating Rigid-Flex - Part 3, "Fab documentation for flex circuits and rigid-flex boards," Altium Live Blog, September 17, 2013, http://blog.live.altium.com/Blogs/ruminating-rigid-flex-pt3
6. Benjamin Jordan, Ruminating Rigid-Flex - Part 4, "Do's and Don'ts of Flex Circuit Copper," Altium Live Blog, September 20, 2013, http://blog.live.altium.com/Blogs/ruminating-rigid-flex-pt4
7. Benjamin Jordan, Ruminating Rigid-Flex - Part 5, "Applications and examples of rigid-flex PCBs," Altium Live Blog, October 14, 2013, http://blog.live.altium.com/Blogs/ruminating-rigid-flex-pt5