A straight line is not always the shortest route between two points in electronic products: Thanks to rigid-flex PCB architecture, circuits can be folded onto themselves with 180° bends—superimposed at minimum height—thereby shrinking product dimensions. Moreover, rigid-flex construction conserves the board territory that cable connectors would otherwise consume, improves system reliability by eliminating connector solder junctions, tightens impedance control, and greatly increases the number of potential paths for board I/O compared to coaxial flat cables. Furthermore, if a product has moving sections with electronics embedded, rigid-flex construction is the ticket.

Credit rigid-flex architecture for the existence of smartphones and other pocket-size electronic wonders. There’s simply no alternative for stuffing that much functionality into such a small space. However, to realize the advantages of rigid-flex construction (including lightweight assemblies) certain design constraints apply specifically to the flex layers in a stackup. Do not attempt your first (or second, or third) rigid-flex design before you consult your prototype manufacturer.

Unlike a conventional PCB stackup, foil construction cannot be used for flex layers. The flex layers in a rigid-flex assembly are built from unreinforced base substrates typically consisting of polyimide dielectric film, clad with rolled annealed copper. The rolled copper is much more flexible than the copper used for rigid boards, but it cannot be plated without becoming brittle. Therefore, the clad base material is first drilled, holes are selectively plated, then the traces and pads are etched. Bondply, a layer of polyimide film with adhesive coating on both sides, isolates that conductor layer from the next, and so forth. A cover layer of adhesive polyimide film insulates and protects the top and bottom surfaces of the flex stack along the ribbon that extends between rigid board sections.

Flex materials are elastic under all circumstances, including processing. During the final lamination of the rigid-flex stack, they are less dimensionally stable than the rigid core and prepreg materials that sandwich them. Hole-to-copper clearance must be greater than the minimum possible with a rigid-only stackup, ideally at least 10 mils. Vias must be farther from the edge of the rigid area adjoining the flex ribbon than the minimum distance in rigid-only stacks, preferably at least 50 mils from the edge, but certainly no less than 30 mils. This rule is the one most violated in rigid-flex designs.
Seek guidance to develop your stackup and design rules. Differing coefficients of thermal expansion among the flex base material, adhesives, prepreg, and rigid cores requires a very careful balance of thicknesses, especially for impedance-controlled designs. There can be many layers of flex in a rigid-flex design, depending on the bend radius of the ribbon portion and whether it will remain stationary after assembly. Flex layer count must be limited in dynamic applications. Consult your manufacturer. If more than four flex layers are required, bonding adhesive must be absent in the sections that are designed to bend. The bend radius should be no less than 12 times greater than the circuit thickness.

Trace routing in the ribbon area should be curved, not angled, to increase peel strength. This recommendation is opposite the routing practice for rigid boards. To increase ribbon flexibility, planes should be cross-hatched; however, the cross-hatch complicates impedance control. Again, a careful balance is required. In some applications, a wide, solid strip under critical traces suffices. Traces on different layers should be staggered vertically, not placed atop each other, to increase ribbon flexibility.

Annular rings should be as large as possible in flex-only regions to reduce the risk of peeling, and the transition from the annular ring to the trace should be teardrop-shaped for the same reason. Adding tabs or anchors (Figure 1) also helps to prevent peeling.

Traces should always be perpendicular to the fold in the flex areas that will be bending. Where flex ribbons have sharp interior corners, tear stops should be added. Copper can be incorporated during layout at the elbow of those corners for reinforcement or polyimide stiffeners can be specified for the inside corner radii. The stiffeners can be laminated when the covercoat is bonded and are the preferred method to prevent tears. The best strategy is to avoid using sharp corners in a flex design.

A very basic checklist for rigid-flex designs includes these routing considerations:

- Stagger flex traces vertically layer to layer
- Turns should be gradual
- Vias should be no closer to the edge of the rigid board than 30 mils at the flex transition
- Minimize flex layers

Remember, rigid-flex PCB designs may be expensive to fabricate, but they can save costs during system assembly. Such architecture often is the only way to squeeze the required product functions within the target package volume. The earlier you consult a PCB manufacturer during product definition, the better your results. PCBDESIGN

Figure 1: Graphic showing anchors, coverlay opening and teardrop-shaped transition from annular ring to trace.