Rigid-Flex PCBs: The Right Choice For Your Product?

by Ian Gabbitas,
Mentor Graphics

The decision to apply rigid or rigid-flex technology to a design should be a collaborative effort. The design of the rigid-flex PCB poses challenges to every member of the product engineering team, involving all disciplines of the team and representatives of the PCB fabricator. Invariably, the fabrication and assembly costs of a rigid-flex design are equal or greater than interconnected rigid PCBs. Involving the fabricator in the design process ensures that designers optimise their capabilities to produce higher yields and reduced costs. The mechanical engineer must understand the technology to properly design the rigid-flex footprint and using the proper drafting objects in the MCAD tools ensures the accurate transition of data to the PCB designer. The electrical engineer must know how a rigid-flex circuit effects performance requirements while the PCB designer must design for the rigid-flex application - not as if it were just another rigid PCB. Most importantly, designers must work closely with fabricators during the final verification process to ensure the design can be fabricated, avoiding field failures during the product life cycle.

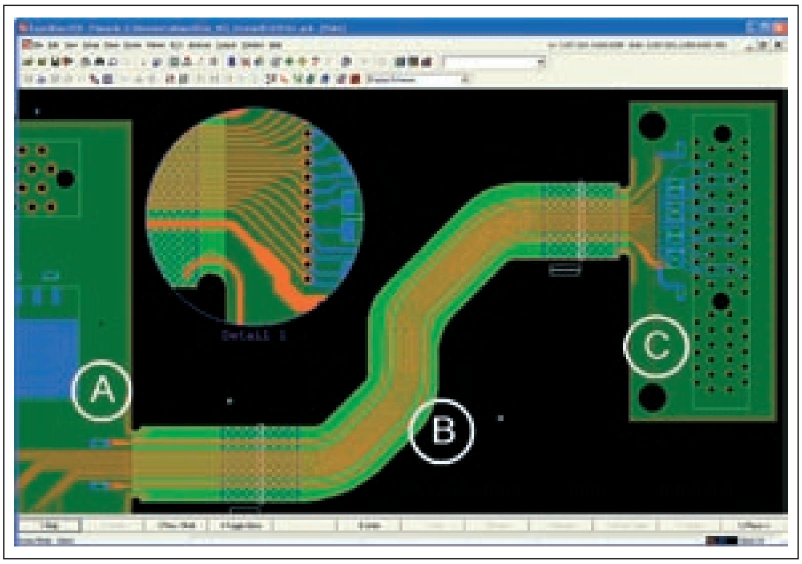


Figure 1 - PCB editor showing flex sub-circuit connected to two rigid sections. The bend regions can clearly be seen

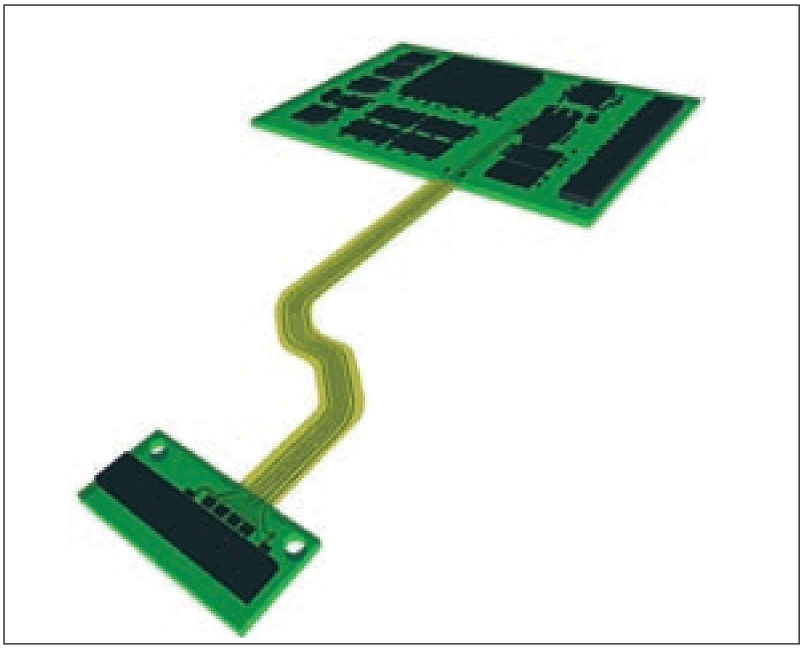


Figure 2 - Viewing the rigid-flex layout database in a 3D Viewer. Here the board designer can verify the flex layout either as a flatten view (shown) or folded into its assembled state along with the mechanical enclosure

Today's consumers incessantly demand smaller and faster products that deliver more features and functionality at a lower cost while at the same time expecting reliability and extended battery life. The convergence of communications and computing technology in mobile consumer products with increasingly smaller footprints (PDAs with integrated cell phones and digital cameras, for example) is driving the need to re-package the electronics. Because of this, there is an ever-increasing need to maximise available space within the electronic assembly using rigid-flex printed circuit boards (PCBs). This is one of the driving factors for incorporating flexible circuits into consumer and aerospace electronics as well as other fields of electronic design.

Flexible circuits are exactly that: flexible. A rigid PCB with a flexible core allows engineers to design electronic sub-systems that can be bent, twisted or otherwise articulated, yet have improved reliability over interconnected rigid PCBs without degrading the performance of the system. One example of flexible circuitry is a 'flip-phone'. The major advantage of a flip-phone is that the screen and keyboard are mutually exclusive, where the size of the screen has little impact on the size of the keyboard and vice versa. The challenge is to connect them together in a reliable manor by employing rigid-flex technology. In this way, the circuit assembly is converted from multiple rigid PCBs joined by wires or ribbon cables to a single rigid PCB with a flexible core. The rigid portions are found under the keypad and display and the flexible core of the PCB provides the interconnect between keypad and display through the hinge.

The use of rigid-flex printed circuits allows multiple rigid PCBs, joined with cables or wires, to be repackaged as a single circuit instead of individual boards.

Application of the rigid-flex design

The application of the product is an important consideration when determining the use of rigid-flex technology. There are two basic types of flex applications – Flex-to-Install and Dynamic Flex. With Flex-to-Install, the circuit is installed flat in the product assembly or bent once to install and not bent again throughout the life of the product. With Dynamic Flex, it is connected to a moving part of the product and subjected to continuous bending throughout the life of the product.

Once the decision has been made to use rigid-flex technology for a project, the manufacturer should be involved as early as possible to ensure that design guidelines are implemented properly. It is also important that the manufacturer knows the intended application so the appropriate materials and fabrication process is chosen.

Should you consider rigid-flex?

The benefits of rigid-flex PCBs can be significant and knowing what they are can help determine whether they are right choice for your product. Some of the expected benefits include:

- Increased reliability by eliminating interface connections
- Reduced manufacturing costs by using a single board
- Reduced packaging weight
- Better control of impedances
- A single CAD design rather than individual PCB databases.

However, there are some challenges that engineers and designers must address when considering the application of rigid-flex PCBs. For one, rigid-flex designs cannot be designed in the same manner as standard rigid PCBs. There must be a more thorough understanding of how rigid-flex PCBs are fabricated and how they operate during the lifetime of the product to ensure cost-efficient designs.

Laying out a rigid-flex design

Without a doubt, rigid-flex circuits are more expensive than rigid counterparts. However, they do offer many benefits as previously discussed. For the PCB designer, there are challenges that must be overcome – not just in the complexity of design but in the CAD tools they use.

Technical capabilities

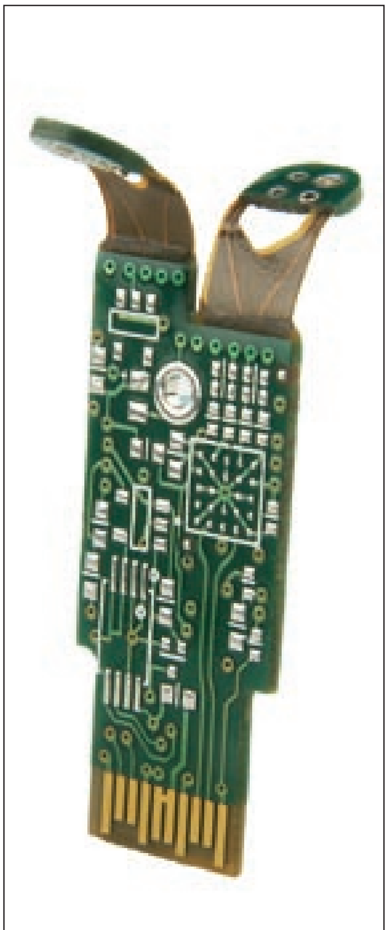
The CAD system must support the specialised capabilities required for rigid-flex PCBs as well as the typical PCB design constraints. These include:

- Variable material properties in the layer stack-up
- True-arced traces not segmented arcs
- Trace tapering and corner filleting
- Bendable areas of the PCB. These features must be supported by a CAD architecture that accommodates these additional design constraints to provide seamless interaction between the rigid and flexible areas of the design.

Mixed constraint regions

Rigid-flex circuits not only require constraints by layer but also by region. Indeed, if the route path from a component pin on one end of the circuit was traced through the layer stack, along the flex layer to the terminating component, varying constraints would appear, dictating the topology of the interconnect. For example, in Figure 1, a signal is routed from a component pin on the rigid portion A, along the flex inner core B, to the termination on rigid portion C. The actual path is quite straightforward, yet the rules that govern its passage across the board change considerably. We start at the keypad where the base substrate is rigid so the routing constraints allow for 90 and 45 degree bends in traces on the 'top' surface of the board. Travelling through the layer stack we get to the inner flex layers, where the rules change.

Figure 3 - A rigid-flex circuit board



No matter how carefully the board is manufactured, allowances must be made for the flexible nature of the material during the fabrication process. Sharp angles or transitions can potentially cause fractures or peeling – and this is well before the product has been assembled. To avoid this, all sharp angles must be filleted, corners changed to curves, track spacing increased, trace width changes tapered, and so on. It is important for the layout tool to recognise the different constraints applied to the flex layers. Even in areas where the flex material is sandwiched between rigid layers, items may change again when the trace travels out of the rigid section and onto the exposed flex region. This is an area where the mechanical stresses are more severe, especially considering a dynamic flex hinge, such as the sliding keypad.

Mechanical collaboration

Rigid-flex PCBs have one unusual benefit over their rigid counterpart in that they allow designers to work in the third dimension. Portions of the PCB can be rolled, folded, twisted and wrapped to follow the contours of an enclosure or hinge mechanism. Unfortunately, most MCAD tools currently recognise only flat (not bent or folded) PCBs. Mechanical engineers are therefore forced to define flexible PCBs as sheet metal objects. This leads a designer to laboriously create board outlines, flex contours and placement constraints derived from mechanical assembly drawings. In a standard rigid design, designers can import the data directly from the MCAD system using various interface formats such as DXF, IGES, STEP or IDF. Today these formats do not support the definition of a folded PCB. The mechanical engineer must convert their flattened sheet metal boards and pass them to the PCB designer as generic graphic objects. The savvy designer can then convert them to a recognisable PCB outline. The problem is exacerbated by the need for the PCB designer to transfer the PCB outline back to the mechanical designer during final validation.

This process is not usually a problem if the interaction between the two CAD tools is infrequent. However, when design cycles are compressed to a matter of few weeks, the process rapidly breaks down. With such a compression of project timescales, keeping the two designs in sync becomes very problematic, as there could be many file versions to manage. More often than not, a Mylar mock-up of the actual 3D design is created and inserted into the assembly to evaluate the correctness of the circuit. Once the desired 3D layout is obtained, the card strips are carefully removed from the housing and laid flat to show the two-dimensional profile. This is hardly rocket science stuff and is certainly not very efficient.

What is required is a transparent, dynamic collaboration between the two design disciplines where data exchange is performed frequently throughout the project life cycle to ensure that both designs are always in sync. This should not be a complicated or time-consuming task as only the changes need be transferred between the systems. With such an infrastructure in place, the board designer could also look at the design in a simple 3D viewer; the flex circuit read directly from the ECAD system and the mechanical housing loaded from the MCAD database. With real-time data synchronisation, the PCB designer could use a 3D viewer as a placement aid and a tool to verify part placement, routing paths or flex bend lines. The viewer would also read in the mechanical assembly and show the flex circuit in place. This would not be a replacement for the MCAD tools but rather an easy-to-use visualisation tool for the board designer.

