Sierra Circuits is the leading name in printed circuit boards, specializing in PCB manufacturing and the assembly of High Density Interconnect Technology. From PCB layout and design to assembly and manufacturing, Sierra Circuits handles all aspects of PCB production.
Why Design A Flex PCB?

Flexible printed circuits are quickly becoming the preferred printed circuit board. Unlike other boards and interconnects, flex circuits are lightweight, easy to install, durable, compact, and—yes—flexible. The range of motion can include over 360, making it the perfect choice for nearly any/every situation. You can use flexible circuits in “bend to install” applications or dynamic applications, where the circuits are continuously in motion. Flexible circuits are also advantageous for the design packages where space is a primary concern.

Flexible and rigid-flex circuits were originally used within the military industry, as they required durable, reliable, lightweight 3D circuitry. Now, flex boards are found in nearly every industry. They are used in devices we use on a daily basis—from phones to computers. Wearable technology almost requires flexible circuits: PCBs need to be durable, tiny, and conform to a myriad of shapes and movements. In the technical industry, they are used in everything from cars, trains, and airplanes to satellites, missiles and radios. In fact, NASA’s Mars Rover has flexible circuits within it. It is, at 140 million miles away, the furthest fully functioning circuit board from Earth.

This guide contains the top tips to remember when designing flex and rigid-flex.
Top Flex Design Tips

• Flex materials have less dimensional stability than standard rigid material so these products typically require looser tolerances. A typical outline tolerance for flexible circuits will be +/- 0.25mm or .010”.

• If your profile tolerance is +/- 0.1mm (4mils) you will require a hard tool or laser cutting which is much more expensive.

• When designing long flexible products, remember your drilled hole to pad sizes, as the dimensional stability is 3 times worse for flexible polyimide laminate @ 0.10% (1000ppm) as opposed to 0.035% (350ppm) for FR4.

<table>
<thead>
<tr>
<th>Layer Type</th>
<th>Standard (mils)</th>
<th>Advanced (mils)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flex</td>
<td>drill + 14</td>
<td>drill + 10</td>
</tr>
<tr>
<td>Outer Layer Rigid</td>
<td>drill + 10</td>
<td>drill + 6</td>
</tr>
<tr>
<td>Inner Layer Rigid</td>
<td>drill + 14</td>
<td>drill + 10</td>
</tr>
</tbody>
</table>

• Dual Access is possible with single-sided flex circuitry. Openings in the base polyimide dielectric material and the top coverlayer are pre-routed or laser ablated allowing access from both the top and bottom sides. Flexible circuits with stiffeners are less expensive in comparison to rigid-flex boards, however, rigid-flex board have more reliable interconnects.

• Use an even number of layers when designing rigid-flex boards to avoid warpage.

• Flexible circuits that require continuous flex cycles must be designed with the copper layer sandwiched between dielectric materials of the same thickness. This is called the neutral axis.

• The bond strength between flexible dielectric's and copper is not as good as FR4 laminates. Tear drops improve this bond strength and solder joints. Tear drops a considered a “must have” for flexible products.
Rigid-Flex Design Tips

All of the flex rules apply to the flex portion of rigid-flex. For the rigid portions, most of the rules are the same as a flex PCB. Exceptions are defined below.

- Drill to copper still has a 10 mil standard. Drill to copper is very important on rigid-flex.
- Hole to flex is the most commonly overlooked flex/rigid-flex design rule.
- Place the flex layers in the middle of the stackup and use an even number of layers.
- Vias in flex areas of a rigid-flex are considered buried.
- Like stiffeners, rigid portions of rigid flex add thickness, but circuitry can also be found in the rigid portion of a rigid-flex.
- The rigid portion of a rigid-flex is usually home to dense components that require a lot of circuitry.
- Sierra recommends putting flexible layers as inner layers when designing rigid-flex boards as opposed to flex outer layers.
- To allow multilayer flex to bend in a tight radius without deformation, a technique called “bookbinding” is used and the layers are manufactured in progressively longer lengths around the outside bend radius.