Rigid flex printed circuit boards (PCBs) are hybrid circuit boards that combine the characteristics of hard boards and flexible circuits, with areas that are rigid for component assembly, providing the same density as hard boards, and flexible areas connecting the rigid boards to one another. The design and construction of these circuits include multiple layers of circuitry in the rigid and flexible sections of the board.

Rigid flex PCBs cost more than traditional rigid, and flexible circuits, but there are times when their performance advantages outweigh the additional cost.

*Rigid flex PCBs cost more, but their performance advantages at times outweigh the additional cost.*
Some applications where they excel in performance are:

**High shock and/or high vibration applications:**
Rigid flex boards are capable of performing and surviving in high shock and/or high vibration environments where traditional PCBs, flexible cables, and connectors will fail.

**Flex endurance:**
Rigid flex boards, properly designed, can provide hundreds of thousands of flex cycles without failure, allowing the designer greater creativity and electronic connectivity in their design.

**Design simplification at a lower overall cost:**
Rigid flex PCBs can reduce overall assembly cost when five or more hard boards are connected together, compared to the cost of using flex cables and connectors. Elimination of the connectors and flexible cables between the rigid boards provides a lower overall packaging cost.

**Slim profile:**
Rigid flex PCBs can be built using very thin dielectrics and still accommodate high density packaging, making them ideal for very thin and/or very lightweight applications.
These characteristics make rigid flex PCBs an ideal solution for use in a wide range of industries—including medical devices, military electronics, aerospace and industrial applications—especially when reliability is a priority; high performance and extended service life without interruption are a must, and equipment downtime cannot be tolerated. It is ideal for “never fail” electronics.

*Rigid flex PCBs are an ideal solution for many industries including:*

- **Medical**
- **Military**
- **Aerospace**
- **Industrial Controls**

Rigid flex PCBs are generally categorized into two major types: flex to install and dynamic flex. The primary distinction of a flex to install PCB is that the board is bent one time during installation or repair operations. In contrast, the design of a dynamic flex board allows for repeated or continuous bending during use—up to hundreds of thousands of flex cycles without failure.
GETTING STARTED ON A RIGID FLEX PCB DESIGN

Here is an examination of several design and installation considerations that will improve the overall manufacturability, robustness, and performance of your rigid flex PCB.

It is important to budget ample time for the initial design stage, as the quality of the design will be an important factor for the resulting PCB’s manufacturability, yield, cost, and lead time. Also, manufacturing rigid flex PCBs takes longer and is much more process intensive than either hard boards or flexible circuits. It is wise to consult your fabricator about their estimated lead times, and budget that into your total design cycle lead time.

It is also highly recommended to involve your PCB fabricator in the beginning to verify that the final design is manufacturable and to avoid common problems in rigid flex design and manufacturing.
Key Considerations Before Designing

Before designing your PCB, there are several considerations to keep in mind, including:

**Material type and layup:**
Your PCB fabricator should be able to help you choose the most suitable materials, suggest the best possible physical stack up, and provide insight into optimal signal integrity considerations.

**Lowering production costs at higher volumes:**
While rigid flex PCBs are a higher-cost solution than rigid and flexible boards, there are ways to keep your design economical. To help you design and build a rigid flex PCB to be made at high volumes at the lowest possible cost, take a look at our Valu Builds for Rigid Flex brochure.

**Lead times and procurement costs:**
Your fabricator should also provide you with some estimates concerning probable lead times and pricing. Our rigid flex cost estimator can give you a rough idea of what your design will cost in production.
Example of Printed Circuits’ Valu Build Program: 10 Layer Quad Flex Valu Build

- 0.0007 Photomagable Soldermask
- 0.0012 Plated Copper
- 0.0007 1/2 oz copper foil
- 0.005 H/H core
- 0.0007 1/2 oz copper foil
- 0.005 1080 prepreg
- 0.001 1080 prepreg
- 0.0007 1/2 oz copper foil
- 0.002 Flexible Laminate
- 0.007 1/2 oz copper foil
- 0.001 1080 prepreg
- 0.001 Bondply
- 0.001 acrylic adhesive
- 0.001 acrylic adhesive
- 0.0007 1/2 oz copper foil
- 0.002 Flexible Laminate
- 0.007 1/2 oz copper foil
- 0.001 1080 prepreg
- 0.001 acrylic adhesive
- 0.001 Coverlayer
- 0.005 H/H core
- 0.0007 1/2 oz copper foil
- 0.0138 Flexible composite thickness

X 12 "Rule of thumb" for minimum bend radius - see IPC 2223

Adder for 1 oz copper

0.0460 Total Thickness With 1 oz Copper

Minimum bend radius for 1/2 oz flex area

0.1656

Minimum bend radius for 1 oz flex area

0.1992

Total Thickness With 1 Oz Copper
Having taken these preliminary considerations into account, you are ready to begin designing your rigid flex PCB. During this phase there are several points to address prior to starting production. These recommendations are briefly discussed in our article, Pointers for Your Rigid-Flex Design, and include:

**Layers**

The layers of a rigid flex PCB are similar in appearance to that of a rigid board, but the flexible layers run all the way through the board in both the flexible and rigid sections of the board. The difference is that some layers are rigid, similar to hard boards, and some layers are flexible, similar to flexible circuit boards.

Rigid flex manufacturers use the same materials that hard board manufacturers and flexible circuit manufacturers use, with one difference. Rigid flex manufacturers have to use a no flow prepreg to bond the layers together. No flow prepreg is critical to rigid flex manufacturing, as it flows up to the edge of the rigid flex transition area, without flowing out onto the flexible sections of the board. Standard prepregs would flow out onto the flex sections of the board, rendering it inflexible.

Other than that, the other materials—core laminate, flexible laminate, coverlayers, bondplies, soldermask, etc—are the same materials used by hard board and flexible circuit shops.
Impedance Considerations

The first consideration to accommodate in your design is any impedance requirements that you might have. Your fabricator should be able to help you model the impedance values that you need, and at the same time develop the material layup that will meet your requirements—both mechanical and electrical.

To start, your fabricator will need the following information from you:

- Which layer(s) require impedance controlled circuits
- Desired value for each circuit
- Which layer(s) are the reference planes for those circuits
- Copper thickness(es) desired
- Type of impedance required—characteristic or differential
- Trace width and spacing desired

Be aware that free online impedance calculators tend to provide erroneous values, and the Dk values provided by many material suppliers are global and do not match up with the actual values of specific materials.

Also unique to rigid flex designs, the circuits transverse both the flexible and rigid areas of the PCB. Impedance values need to be modeled for each area separately because the material Dk’s are different. Adjacent planes and circuits can also interfere with your desired values.

This is why it is critical to involve your fabricator at the beginning of your design, because they will have the experience and software to model even very complex impedance constructions, and then verify the results when the boards are completed. They also will have a library of materials typically used in rigid flex manufacturing, built into their software, providing more accurate impedance modeling for your design.
Material Layup

Once your fabricator has modeled your impedance requirements, or as they are doing so, they will, at the same time be formulating a recommended material layup that will meet all of your requirements. It should include a complete material stack up for both the rigid areas of the board as well as the flexible sections. Each impedance trace will be modeled and will include all of the information above for each circuit.

Additionally the impedance model will give you, the designer, the appropriate layup of materials (example shown below), as well as the circuit geometries and spacing you will need to use in your design to achieve the desired impedance values:

- **Laminate** - Dk value, material type, thickness and copper weights
- **Prepreg** - Dk value, material type, glass thickness and how many plies
- **Flexible copper-clad laminate** - Dk value, material type, thickness, and copper weights
- **Coverlay, bondply, and cast adhesive** - Dk value, material type, thicknesses

---

<table>
<thead>
<tr>
<th>Laminate</th>
<th>Dk value, material type, thickness and copper weights</th>
</tr>
</thead>
<tbody>
<tr>
<td>Prepreg</td>
<td>Dk value, material type, glass thickness and how many plies</td>
</tr>
<tr>
<td>Flexible copper-clad laminate</td>
<td>Dk value, material type, thickness, and copper weights</td>
</tr>
<tr>
<td>Coverlay, bondply, and cast adhesive</td>
<td>Dk value, material type, thicknesses</td>
</tr>
</tbody>
</table>

---

**Printed Circuits’ Impedance Modeling**
A detailed print explicitly communicates the design requirements and the desires of the designer, including any and all critical considerations.

While hard board designs often allow some details to be left out, the more complex nature of rigid flex PCBs makes detailed prints essential.

Also, it is usually very difficult to see the rigid to flex transition areas in the gerber files. Detailed prints help your fabricator see your intent, and build your boards to your requirements.

Detailed print of rigid flex circuit board
Rigid-to-Flex Transition Area Considerations

Maintaining distance from what rigid flex fabricators refer to as the “keep-out” area is an essential consideration in rigid flex board design. The location of this area is on the hard board side of the rigid-to-flex transition zone.

Optimal distances for pads and traces from the rigid flex transition line are 0.025 inches. Optimal distances for the vias from the rigid flex transition line are .050 inches - from the edge of the drilled hole to the transition line.

Additional Design Considerations and Tips

Other considerations to keep in mind include:

- Avoid specifying filled and cap plated vias on rigid flex boards if at all possible.

- In dynamic flex and high reliability applications, specify a soft epoxy strain relief beading 0.060 inch wide along with the rigid-to-flex interface to reduce stress on the circuits at the rigid to flex transition.

- Avoid dual finishes on rigid flex PCB surfaces (i.e., where a PCB features one pad finish on one part of the board and an alternative finish in other locations).

- When UL ratings are desired, or required, employ fabricators that are certified to UL 94V-0, which greatly simplifies the process of getting your device recognized by UL.
Rigid flex boards are assembled similarly to rigid boards, but there are a few differences to keep in mind.

The flexible layers, coverlayers, and bondplies in rigid flex boards are hygroscopic, so they will absorb water from their environment. Even small amounts of absorbed water in the board will greatly expand during the thermal excursion of assembly and will cause the board to delaminate and rupture vias. It is important to sufficiently bake, or otherwise dessicate, rigid flex boards prior to assembly.

Assemblers of rigid flex PCBs typically want their parts supplied in arrays to hold the boards stable during assembly. The assembler then removes them from the arrays after assembly. As a designer, you should work to accommodate their requirements and think about where in your design is an ideal location for the array tabs or where you might want to avoid tabs due to mechanical constraints. You can then ask your fabricator to design the array to the assembler’s requirements or do it yourself. Either is acceptable.

Also scoring in arrays is a common practice in rigid boards but this process does not achieve ideal results, as the flexible layers are still buried in the rigid board, making separation very difficult on the score line.
When designing your rigid flex PCB, it is important to keep the logistics of installation in mind. Doing so will save you any frustration when it comes time to implement your PCB. Some of these installation considerations include:

**Minimum bend radius:**
The IPC 2223 Sectional Standard on Flexible Circuit Design provides guidelines for minimum bend radius. You may also refer to manufacturers' recommendations for the flexible materials.

In high reliability applications, the typical minimum bend radius is 6x the thickness of your flexible composite for single- or double-sided boards, and 12x the thickness of your flexible composite with three or more layers.

**Consideration of the end application:**
Consider whether flex to install or dynamic flex would be more suitable for the end application, and whether there will be high levels of shock or vibration involved.

For example: bicycle computers, missile guidance systems, and wearable electronics, even when they are flex to install, are often dynamic flex applications due to the shock and/or vibration of the application.
Avoid bends of 180° whenever possible:
180° bends, particularly in multilayered flex regions of rigid flex boards, can put a tremendous amount of strain on the circuits in the bend area that can be difficult for them to survive over time, unless the bend radius is very large.

Consider using two 90° bends or incorporating an “S” type bend to reduce stress on the individual circuits.

Create “paper dolls”:
Create a design mockup—also referred to as a paper doll—with the same mechanical features as the intended PCB.

This provides designers and manufacturers insight into the mechanical aspects of the assembly and installation phases during the design phase, when it is still possible to correct any issues before moving into production.

Pre-heating before installation:
Rigid flex PCBs, even properly designed boards, may need extra measures to be seamlessly installed.

Pre-heating the assembly at 225°F for a few minutes will greatly increase the flexibility of the board, enabling an easier bend and installation.
When designing a rigid flex PCB, it is important to consider all of the relevant factors that will go into producing the right board for your application. This includes desired properties for it to have excellent performance in the end application, construction materials and their configurations, and conditions for assembly and installation. Involving your PCB fabbricator from the very start of your PCB building process helps each of the design, assembly, and installation processes to be smooth and successful.

At Printed Circuits, we have more than 40 years of experience crafting custom, high-performance rigid flex PCBs. Located in Minneapolis, Minnesota, we are at the heart of one of the highest concentrations of flexible circuit manufacturers in the world, providing us with all the technology, skills, and experts necessary to make superior rigid flex PCB designs. We also hold several certifications, including UL 94V-0, ISO 9001:2015, and MIL-PRF-31032. Our services cater to a wide range of industries; we serve a large number of military electronics, medical devices, computer, telecommunication, and industrial instrumentation clientele.

Contact us today to see how our rigid flex PCB solutions can be an optimal fit for your application needs.

Contact Us  Resource Library