Rigid-Flex, Flex and Semi-Flex PCB Technology

Albert Schweitzer
Fine Line Gesellschaft
Für Leiterplattentechnik mbH
Itterpark 4, D-40724
© Copyright Fine Line
19.03.2017 Vers. 1.1
# Table of Content

<table>
<thead>
<tr>
<th></th>
<th>Content</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>General</td>
</tr>
<tr>
<td>2</td>
<td>Advantages of flexible and rigid-flexible PCBs</td>
</tr>
<tr>
<td>3</td>
<td>IPC Standards for flex and rigid-flex PCBs</td>
</tr>
<tr>
<td>4</td>
<td>PCB Base Materials for flex and rigid-flex PCBs</td>
</tr>
<tr>
<td>5</td>
<td>Flex Layout Options</td>
</tr>
<tr>
<td>6</td>
<td>Flex and Rigid-Flex Design Rules</td>
</tr>
<tr>
<td>7</td>
<td>Bending and Bending Radius</td>
</tr>
<tr>
<td>8</td>
<td>Handling of Rigid-Flex PCBs</td>
</tr>
<tr>
<td>9</td>
<td>Golden Rules</td>
</tr>
</tbody>
</table>
The use of flexible and rigid-flexible PCBs opens for many applications completely new possibilities and advantages regarding signal transmission, size, stability and long-term reliability.
Advantages of flexible and rigid-flexible PCBs
Advantages of flexible and rigid-flexible PCBs

The Flex and Rigid Flex technology offers the following advantages:

- Reduction of weight and volume
- Freedom to design in 3 dimensions
- Reliability of the electrical connections due to reliable orientation and reliable contacts as well as savings on connectors and wiring
- Defined characteristics of the circuit systems on the printed circuit (e.g. Impedances)
- Dynamically and mechanically robust
- Often cost savings for the entire system
IPC Standards for flex and rigid-flex PCBs
IPC Standard for flex and rigid-flex PCBs

For first time designers and engineers it is a good idea to become familiar with the specifications that have reference to flex and rigid-flex manufacturing.

Most important are the following two IPC Standards:

- IPC 2223C
- IPC-6013C.
For designers and engineers most important is Standard IPC-2223C: "Sectional Design Standard for Flexible Printed Boards".

This standard provides guidance on selecting adhesive materials as well as connecting flex and rigid PCBs, with specific tips such as placing flex adhesives away from plated through-holes and vias. This specification has the most comprehensive tips for creating your design.
Important for everybody in the field of flex and rigid-flex PCBs is the following IPC standard IPC-6013C: “Qualification and Performance Specification for flexible Printed Boards”.

This specification covers qualification and performance requirements of flexible printed wiring. The flexible printed wiring may be single-sided, double-sided, multilayer or rigid-flex multilayer. All of these constructions may or may not include stiffeners, through holes and blind/buried vias.
Important in the selection of materials are further the following IPC standards:

- **IPC-4202** „Flexible Base Dielectrics for Use in Flexible Printed Circuitry”
- **IPC-4203** „Adhesive Coated Dielectric Films for Use as Cover Sheets for Flexible Printed Circuitry and Flexible Adhesive Bonding Films”
- **IPC-4204** „Flexible Metal-Clad Dielectrics for Use in Fabrication of Flexible Printed Circuitry”
IPC Standard for flex and rigid-flex PCBs

IPC-2223 Board Type: **Type 1**

- Single-sided flexible printed circuit,
- with one conductive layer,
- with or without stiffener,
- with adhesive-coated or adhesiveless substrate

**Adhesive-coated laminate:**

**Adhesiveless laminate:**

Source: IPC-2223
IPC Standard for flex and rigid-flex PCBs

IPC-2223 Board Type: **Type 2**

- Double-sided flexible printed circuit,
- with two conductive layer,
- with or without stiffener,
- with adhesive-coated or adhesiveless substrate

**Adhesive-coated laminate:**

**Adhesiveless laminate:**

Source: IPC-2223
IPC Standard for flex and rigid-flex PCBs

IPC-2223 Board Type: **Type 3**

- Multilayer flexible printed circuit,
- with three or more conductive layer,
- with or without stiffener,
- with adhesive-coated or adhesiveless substrate

**Adhesive-coated laminate:**

**Adhesiveless laminate:**

Source: IPC-2223
IPC Standard for flex and rigid-flex PCBs

IPC-2223 Board Type: **Type 4**

- Rigid-Flex printed circuit with plated-through holes
- with three or more conductive layer,
- with adhesive-coated or adhesiveless substrate

**Adhesive-coated laminate:**

- Adhesive
- Coverlayer
- Polyimide Substrate
- Copper Pads
- Copper-Plated Through-Hole

**Adhesiveless laminate:**

- Prepreg
- Adhesiveless Substrate
- Coverlayer
- Copper-Plated Through Hole
- Copper Pads
- Rigid Material

Source: IPC-2223
IPC Standard for flex and rigid-flex PCBs

IPC-2223 Board Type: **Type 5**

- Rigid-Flex printed circuit without plated-through holes
- with two or more conductive layer,
- with adhesive-coated or adhesiveless substrate

**Adhesive-coated laminate:**

**Adhesiveless laminate:**

Source: IPC-2223
PCB Base Material
The base materials for flex and rigid-flex printed circuit boards can be divided into three categories:

- **Flexible, Copper Clad Laminate**
- **Coverlayer, protective Foils, flexible Coating systems**
- **Adhesive Systems (rigid and flexibel)**

- Copper foil
  - One-sided
- Copper foil
  - Double-sided
- Bondply (Adhesive-Polyimid-Adhesive)
- Polyimid Coverlayer
- Acryl Adhesive Sheet
- No Flow Prepreg
The base materials for flex and rigid-flex printed circuit boards can be divided into three categories:

- **Flexible, Copper Clad Laminate**
- Coverlayer, protective Foils, flexible Coating systems
- Adhesive Systems (rigid and flexibel)
Overview of flexible Laminates:

- **PET (Polyethylenterephthalat)**
  Thermoplastic polyesters for simple applications

- **PEN (Polyethylennaphthalat)**
  Thermoplastic polyesters for simple applications

- **LCP (Liquid Crystal Polymer)**
  As an alternative to polyimide, for extremely demanding applications as RF applications

- **Polyimide (Trade name e.g. Kapton)**
  Industry standard, very flexible use

- **Semiflex (Based FR4 or special materials)**
  Only limited flexible
## Comparison of the technical characteristics of flexible laminates

<table>
<thead>
<tr>
<th>Feature</th>
<th>PET/PEN with Adhesive</th>
<th>Polyimid without Adhesive</th>
<th>LCP</th>
</tr>
</thead>
<tbody>
<tr>
<td>E modulus</td>
<td>3000MPa</td>
<td>4500MPa</td>
<td>2300MPa</td>
</tr>
<tr>
<td>Bending load</td>
<td>akzeptabel</td>
<td>sehr gut</td>
<td>Sehr gut</td>
</tr>
<tr>
<td>Peel Strength</td>
<td>1050N/m</td>
<td>1600N/m</td>
<td>1000N/m</td>
</tr>
<tr>
<td>Dielectric constant</td>
<td>3,3 (1MHz)</td>
<td>3,2 (1MHz)</td>
<td>2,9 (10GHz)</td>
</tr>
<tr>
<td>max. Temperature</td>
<td>85°C/160°C</td>
<td>220°C</td>
<td>280°C</td>
</tr>
<tr>
<td>Dielectric strength</td>
<td>200V/µm</td>
<td>250V/µm</td>
<td>150V/µm</td>
</tr>
<tr>
<td>Insulation Resistance</td>
<td>1x10^{12}Ω</td>
<td>1x10^{13}Ω</td>
<td>1x10^{11}Ω</td>
</tr>
<tr>
<td>Solder Resistance</td>
<td>No/260°C (5s)</td>
<td>400°C (30s)</td>
<td>288°C (30s)</td>
</tr>
<tr>
<td>Moisture Absorption</td>
<td>&lt;0.5%</td>
<td>&lt;1%</td>
<td>0.04%</td>
</tr>
<tr>
<td>Enlargement</td>
<td>&lt;80ppm</td>
<td>&lt;50ppm</td>
<td>&lt;20ppm</td>
</tr>
</tbody>
</table>
Base Material for flex and rigid-flex PCB

Basics about flexible laminates:
Layer Thickness, Kind of Copper and Adhesive:

- Thickness of Copper: 9, 18, 35, 70µm, RA or ED copper
  (RA = Rolled Annealed; ED = Electrodeposited)
- Thickness of Adhesive: 25–35µm, Acrylic or Epoxy based
- Thickness of Polyimide: 25, 50, 75, 100, 125µm

Adhesive based: double sided copper laminate
Adhesiveless: Double sided copper laminate

Source: Fineline
We have just learned, that there are two types of copper foils for flexible circuit boards used:

- **RA (Rolled Annealed) Copper**
  RA copper is due to its property of being under mechanical loading well to deform (ductility), in particular used for flexible and rigid-flex printed circuit boards. Because of the smooth surface is RA copper also the first choice for RF applications.

- **ED (Electrodeposited) Copper**
  ED copper foil is the standard copper foil for rigid PCBs and the most commonly used copper type in the PCB industry. It is on a large scale and for flexible printed circuit boards used when the flexibility of the application is not the most important feature within the application.
Material for flex and rigid-flex PCB

Important to know:

- **RA Copper**
  The rolling process creates a very smooth surface reducing skin resistance and becomes very beneficial at high frequencies above about 10 GHz

- **RA Copper**
  Due to ductility, RA Copper is the first choice for flex application

- **RA Copper**
  RA foil's crystalline grain structure is oriented parallel to the foil plane

- **RA Copper**
  Rolled copper foil is considerably more expensive than ED foil

- **ED Copper**
  New ED copper foils with very fine non-columnar grain structure are starting to replace RA foil in some applications
Material for flex and rigid-flex PCB

Manufacture of RA copper foil:

High pressure mechanical rollers

In several passes the rolling mill reduces the copper thickness until it produces copper foil suitable printed circuits. It is then annealed and treated for adhesion.

Source: TTM
Material for flex and rigid-flex PCB

Manufacture of ED copper foil:

As the drum rotates slow clockwise, copper is plated on to the drum. After 180 degree rotation the copper is at its final thickness and it is removed from the drum in a continuous web.

Source: TTM
Material for flex and rigid-flex PCB

Manufacture of ED copper foil:

Electrodeposited copper foil has two surfaces. The drum side is basically an image of the drum surface and is quite smooth, while the matt side exhibits dendritic features.

- Copper foil requires a surface treatment prior to bonding prepreg resins
- Surface treatment can be applied to either side or both
- Some resins can require different treatments

Drum side

Matt side

Source: TTM
Another very important feature in use of flexible laminates is the distinction in:

- adhesive laminates
- adhesive-free laminates.

The advantages of adhesiveless flexible laminates will be showed on the next slide.
Advantages of adhesiveless flexible Material:

- **Chemical Resistance**
  Adhesiveless laminates are more resistant to harsh chemicals.

- **High Temperature Application**
  Adhesiveless laminates can withstand higher temperatures and have better dimensional stability than adhesive based material. Up to 180°C.

- **Thin and ultra thin Application**
  Adhesiveless laminates can be produced as thinner materials.

- **Controlled Impedance Application**
  Adhesiveless laminate have a homogeneous dielectric structure which can make them an excellent choice when impedance control is critical.

- **High layer count Application**
  Adhesives have a relatively high rate of thermal expansion. This can cause reliably problems when going through thermal cycling.

- **High flexibility Application**
  Adhesiveless laminates have stronger mechanical bonds then adhesive systems.
Base Material for flex and rigid-flex PCB

Examples for flexible copper clad material:

All Pyralux® copper-clad laminates are available with rolled, annealed copper or electro-deposited copper.

**Dupont Pyralux® Adhesive Based Clads**

**Pyralux® FR** - Pyralux® FR acrylic based laminates are made with DuPont™ Kapton® polyimide film and are available in sheet form as single or double sided clads in a wide variety of thicknesses.

**Pyralux® LF** - DuPont™ Pyralux® LF products are acrylic-based copper clad laminates, coverlays, bondplys and sheet adhesives and have been the industry standard in high reliability applications in the consumer electronics industry for over 35 years with a proven record of consistency and dependability.
Pyralux® All Polyimide Clads (adhesiveless)

**Pyralux® AP** - DuPont™ Pyralux® AP is an all polyimide double sided copper clad laminate that has excellent thermal, chemical and mechanical properties ideal for use in high reliability flex and multilayer flex circuitry.

**Pyralux® AC** - DuPont™ Pyralux® AC is an all polyimide single sided copper clad laminate ideal for applications that require thin, light and high density circuitry along with chip-on-flex attachment.

**Pyralux® APR** - DuPont™ Pyralux® APR is an all polyimide double sided resistor laminate ideal for advanced applications in military, aerospace, automotive and consumer electronics markets, where reliable embedded resistor technology, temperature tolerance, and robust processing are required.

**Pyralux® HT** - DuPont™ Pyralux® HT is an all polyimide flexible laminate system with the highest service temperature available today, that includes a double-sided copper-clad laminate and a unique all polyimide coverfilm or bonding material.
### Base Material for flex and rigid-flex PCB

---

**Data Sheet**  
**Pyralux AP Flex Material:**

<table>
<thead>
<tr>
<th>Laminate Property</th>
<th>IPC TM-650 (* or other)</th>
<th>AP-9111 1 mil dielectric</th>
<th>AP-9121 2 mil dielectric</th>
<th>AP-9131–9161 3–6 mil dielectric</th>
</tr>
</thead>
<tbody>
<tr>
<td>Adhesion to Cu (Peel Strength)</td>
<td>Method 2.4.9</td>
<td>1.8 (9)</td>
<td>&gt;1.8 (10)</td>
<td>&gt;1.8 (10)</td>
</tr>
<tr>
<td>As fabricated, N/mm (lb/in)</td>
<td></td>
<td>1.8 (9)</td>
<td>&gt;1.8 (10)</td>
<td>&gt;1.8 (10)</td>
</tr>
<tr>
<td>After solder, N/mm (lb/in)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Solder Flex at 268°C (550°F)</td>
<td>Method 2.4.13</td>
<td>Pass</td>
<td>Pass</td>
<td>Pass</td>
</tr>
<tr>
<td>Dimensional Stability</td>
<td>Method 2.2.4</td>
<td>-0.04 to -0.08</td>
<td>-0.04 to -0.07</td>
<td>-0.03 to -0.08</td>
</tr>
<tr>
<td>Method B, %</td>
<td></td>
<td>-0.05 to -0.08</td>
<td>-0.04 to -0.07</td>
<td>-0.03 to -0.08</td>
</tr>
<tr>
<td>Method C, %</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Dielectric Thickness Tolerance, %</td>
<td>Method 4.6.2</td>
<td>±10</td>
<td>±10</td>
<td>±10</td>
</tr>
<tr>
<td>UL Flammability Rating</td>
<td>*UL-94</td>
<td>E16</td>
<td>E17</td>
<td>E17</td>
</tr>
<tr>
<td>Dielectric Constant*, 1 MHz</td>
<td>Method 2.5.5.3</td>
<td>3.4</td>
<td>3.4</td>
<td>3.4</td>
</tr>
<tr>
<td>Dissipation Factor*, 1 MHz</td>
<td>Method 2.5.5.3</td>
<td>0.003</td>
<td>0.002</td>
<td>0.002</td>
</tr>
<tr>
<td>Dielectric Strength, kV/mil</td>
<td>Method 2.5.6.1</td>
<td>6–7</td>
<td>6–7</td>
<td>6–7</td>
</tr>
<tr>
<td>Volume Resistivity, ohm·cm</td>
<td>Method 2.5.17.1</td>
<td>E16</td>
<td>E17</td>
<td>E17</td>
</tr>
<tr>
<td>Surface Resistance, ohms</td>
<td>Method 2.5.17.1</td>
<td>&gt;E16</td>
<td>&gt;E16</td>
<td>&gt;E16</td>
</tr>
<tr>
<td>Moisture and Insulation Res., ohms</td>
<td>Method 2.6.3.2</td>
<td>E11</td>
<td>E11</td>
<td>E11</td>
</tr>
<tr>
<td>Moisture Absorption, %</td>
<td>Method 2.6.2</td>
<td>0.8</td>
<td>0.8</td>
<td>0.8</td>
</tr>
<tr>
<td>Tensile Strength, MPa (kpsi)</td>
<td>Method 2.4.19</td>
<td>&gt;345 (&gt;50)</td>
<td>&gt;345 (&gt;50)</td>
<td>&gt;345 (&gt;50)</td>
</tr>
<tr>
<td>Elongation, %</td>
<td>Method 2.4.19</td>
<td>&gt;50</td>
<td>&gt;50</td>
<td>&gt;50</td>
</tr>
<tr>
<td>Initiation Tear Strength, g</td>
<td>Method 2.4.16</td>
<td>700–1000</td>
<td>900–1200</td>
<td>900–1200</td>
</tr>
<tr>
<td>Propagation Tear Strength, g</td>
<td>Method 2.4.17.1</td>
<td>&gt;10</td>
<td>&gt;20</td>
<td>&gt;20</td>
</tr>
<tr>
<td>Chemical Resistance, min. %</td>
<td>Method 2.3.2</td>
<td>Pass, &gt;95%</td>
<td>Pass, &gt;95%</td>
<td>Pass, &gt;95%</td>
</tr>
<tr>
<td>Soldorability</td>
<td>*IPC-S-804, M. 1</td>
<td>Pass</td>
<td>Pass</td>
<td>Pass</td>
</tr>
<tr>
<td>Flexural Endurance, min. cycles</td>
<td>Method 2.4.3</td>
<td>6000</td>
<td>6000</td>
<td>6000</td>
</tr>
<tr>
<td>Glass Transition (Tg), °C</td>
<td>--</td>
<td>220</td>
<td>220</td>
<td>220</td>
</tr>
<tr>
<td>Modulus, kpsi</td>
<td>--</td>
<td>700</td>
<td>700</td>
<td>700</td>
</tr>
<tr>
<td>In-Plane CTE (ppm/C) T&lt;Tg</td>
<td>--</td>
<td>25</td>
<td>25</td>
<td>25</td>
</tr>
<tr>
<td>In-Plane CTE (ppm/C) T&gt;Tg</td>
<td>--</td>
<td>40 (est.)</td>
<td>40 (est.)</td>
<td>40 (est.)</td>
</tr>
</tbody>
</table>
The base materials for flex and rigid-flex printed circuit boards can be divided into three categories:

- **Flexible, Copper Clad Laminate**
- **Coverlayer, protective Foils**
- **Adhesive Systems (rigid and flexibel)**

- Copper foil One-sided
- Bondply (Adhesive-Polyimid-Adhesive)
- Polyimid Coverlayer
- Acryl Adhesive Sheet
- Copper foil Double-sided
- No Flow Prepreg
What is a Coverlayer?

A cover layer is a form of protective coating on top of the metal foil in a flex design. It offers better protection against wear and scratch compared to solder mask, and also helps the metal foil stick to the base material, providing improved adhesion.
Base Material for flex and rigid-flex PCB

There are different kinds of Coverlayers:

**Based on Polyimide**
- Includes adhesive layer
- Clearance must be milled or done by laser
- Bonded by the use of heat and pressure
- Very well suited for dynamic flex applications

**Photo imageable covercoat**
- Laminated by Vacuum-Laminator
- Foil structured by film exposure and wet chemical developing (application always includes solder mask function)
- Use for semi-flexible applications
Base Material for flex and rigid-flex PCB

There are different kinds of Coverlayers:

Flexible printed circuits can be also coated with flexible solder masks. Besides thermal curing types, flexible solder masks are available as UV curing and photoimage able systems. A number of aspects must be considered when selecting the ideal ink for the application at hand, such as the demands on registration accuracy and resolution as well as the compatibility between the foil substrate and the solder mask process.
There are two possibilities to apply the coverlayer:

„Embedded“ Coverlayer, the coverlayer is embedded in the rigid stack-ups of the flex-rigid design.

„Selektive“ Coverlayer, also so called „Bikini“ Coverlayer. Cover only the bare necessities.
Base Material for flex and rigid-flex PCB

„Bikini“ Coverlayer:

Eliminate the amount of acrylic adhesive in critical areas

Important!
It is important to use “no flow” prepregs
In case you realize a Bikini Cover Layer

The fabricator will let the cover layer extend a bit into the rigid stack-up for stability.
The „CTE“ value (Thermal Expansion Coefficient):

One of the most important parameter at Circuit Board base-materials is the CTE (Coefficient of Thermal Expansion) Value. Particularly important is the consideration of Z-axis.

Determination of the Coordinates in printed circuit boards:
Base Material for flex and rigid-flex PCB

Any Adhesive material increase the risk for Delamination:

**Z-Axis Expansion vs. Temperature**

![Graph showing Z-Axis Expansion vs. Temperature with points marked for 215°C, 245°C, and 260°C reflow.]
The „CTE“ value:

The determination of the thermal expansion coefficients

\[
\text{ppm/K} = \frac{\mu m}{m*K}
\]

- \(\alpha_z = \Delta s_1 / \Delta T_1\)
  - Expansion above Tg

- \(\alpha_z = \Delta s_2 / \Delta T_2\)
  - Expansion below Tg

- \(\alpha_z\) = Expansion
  - In the range between 50°C and 260°C

Source: Isola
There is an analogy between a coupled spring system and the layer structure of a PCB:

The different materials involved in the PCB structure were treated as springs with different elastic constants, providing an alternative method to express the different coefficients of thermal expansion of each material.
Base Material for flex and rigid-flex PCB

Example for Pyralux® Coverlays

**Pyralux® FR** - DuPont™ Pyralux® FR coverlay composites are constructed of DuPont™ Kapton® polyimide film, coated on one side with a proprietary B-staged modified acrylic adhesive.

**Pyralux® LF** - DuPont™ Pyralux® LF coverlay composites are constructed of DuPont™ Kapton® polyimide film, coated on one side with a proprietary B-staged modified acrylic adhesive.

**Pyralux® LF-B** - DuPont™ Pyralux® LF-B is a black polyimide acrylic coverlay made with DuPont™ Kapton® B polyimide film ideal for products where a uniform, aesthetically pleasing appearance is desired.

**Pyralux® PC** - DuPont™ Pyralux® PC 1000 is a modified acrylic flexible photoimagable dry film coverlay used for single and double sided applications that require fine line resolution along with bend and crease flexibility.

**Pyralux® HXC** - DuPont™ Pyralux® HXC is DuPont™ Kapton® MBC black polyimide film coated with epoxy ideal for products where uniform matte black appearance is desired.
The base materials for flex and rigid-flex printed circuit boards can be divided into three categories:

- **Flexible, Copper Clad Laminate**
- **Coverlayer, protective Foils**
- **Adhesive Systems (rigid and flexible)**

**Copper foil**
- One-sided
- Double-sided

**Bondply** (Adhesive-Polyimid-Adhesive)

**Polyimid Coverlayer**

**Acryl Adhesive Sheet**

**No Flow Prepreg**
Base Material for flex and rigid-flex PCB

Adhesive Systems (rigid and flexible):

There are a lot of different materials and methods available to stick together the different layers within an rigid-flex PCB.

One of them is the “Bondply” material. Bondplys are coated on both sides with acrylic adhesive.
Base Material for flex and rigid-flex PCB

Examples for Pyralux® Bondplys

**Pyralux® FR** - DuPont™ Pyralux® bondply composites are constructed of DuPont™ Kapton® polyimide film coated on both sides with a proprietary B-staged modified acrylic adhesive.

**Pyralux® LF** - DuPont™ Pyralux® bondply composites are constructed of DuPont™ Kapton® polyimide film coated on both sides with a proprietary B-staged modified acrylic adhesive.

**Pyralux® TK** - DuPont™ Pyralux® TK is a fluoropolymer/polyimide composite double sided copper clad laminate and bondply ideal for high speed digital and high frequency flexible circuit applications.
What is a Stiffener?

Stiffeners are pieces of rigid material bonded to flex regions to “rigidize” a section of a flex design, to allow components to be assembled on the flex area, or to provide rigid mounting holes. The stiffener material can be conductive, such as metal, or non-conductive, such as plastic or FR4.
Important Features to order a rigid-flex PCB:

<table>
<thead>
<tr>
<th>Feature</th>
<th>Option 1</th>
<th>Option 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer count</td>
<td>1,2</td>
<td>more</td>
</tr>
<tr>
<td>Flex clad material</td>
<td>adhesive</td>
<td>adehessivlessless</td>
</tr>
<tr>
<td>Flex clad copper</td>
<td>ED</td>
<td>RA</td>
</tr>
<tr>
<td>Coverlayer</td>
<td>Acrylic based</td>
<td>Epoxy based</td>
</tr>
<tr>
<td>No flow Prepreg</td>
<td>Epoxy based</td>
<td>P&quot;I based</td>
</tr>
<tr>
<td>Bikini</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Rigid clad material</td>
<td>Epoxy based</td>
<td>P&quot;I based</td>
</tr>
<tr>
<td>Stiffener</td>
<td>Yes</td>
<td>No</td>
</tr>
</tbody>
</table>

Base Material for flex and rigid-flex PCB

Important parameter for quotation of Rigid-Flex PCBs:
Fastprint can provide the following materials from stock:

<table>
<thead>
<tr>
<th>Fastprint Rigid-Flex Material</th>
<th>Option 1</th>
<th>Option 2</th>
<th>Option 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Item</td>
<td>Option 1</td>
<td>Option 2</td>
<td>Option 3</td>
</tr>
<tr>
<td>Adehise flex clad material</td>
<td>Taiflex</td>
<td>Shengyi</td>
<td>Dupont</td>
</tr>
<tr>
<td>Adhesiveless flex clad material</td>
<td>Dupont</td>
<td>Panasonic</td>
<td>Taiflex</td>
</tr>
<tr>
<td>Acryl based coverlayer</td>
<td>Dupont</td>
<td>Shengyi</td>
<td>Panasonic</td>
</tr>
<tr>
<td>Exopx based coverlayer</td>
<td>Taiflex</td>
<td>Ventec</td>
<td>Taiflex</td>
</tr>
<tr>
<td>Epoxy no flow prepreg</td>
<td>49N Arlon</td>
<td>Shengyi</td>
<td>Isola 406</td>
</tr>
<tr>
<td>P&quot;I no flow prepreg</td>
<td>37N Arlon</td>
<td>Ventec</td>
<td>Isola 406</td>
</tr>
</tbody>
</table>
Flex Layout Options
Flex und Rigid-Flex Technology

Examples for Rigid-Flex PCBs:

[Images of PCBs]
Flex und Rigid-Flex Technology

Symmetrical Structure:

Unsymmetrical Structure:
Example: 2-Layer Flex-PCB with Stiffener:

- Adhesive
- Adhesiveless

- Copper
- Adhesive
- Polyimide
- Coverlayer
- Bondbly
- Stiffener
Flex und Rigid-Flex Technology

Example: 4-Layer Multilayer PCB with Stiffener:

- Adhesiveless
- Copper
- Adhesive
- Polyimide
- Coverlayer
- Bondbly
- Stiffener
Flex und Rigid-Flex Technology

Example: 6-Layer Multilayer Rigid-Flex PCB:

Stack-up of Panasonic's material:

Layer 1:
- 15 μm: soldmask (green)
- 48 μm: base copper (30 μm) + plated: 28 μm
- 510 μm: FR-4 rigid
- 35 μm: inner copper

Layer 2:
- 150 μm: low flow prepreg
- 1080
- 35 μm: inner copper
- 50 μm: Polyimide (double side)

Layer 3:
- 35 μm: inner copper

Layer 4:
- 35 μm: inner copper
- 150 μm: low flow prepreg: coverlay PI: (25) μm
- 1080

Layer 5:
- 35 μm: inner copper
- 510 μm: FR-4 rigid

Layer 6:
- 48 μm: base copper (30 μm) + plated: 28 μm
- 15 μm: soldmask (green)

Total thickness: 1638 μm

The flex part thickness: (0.22) ± 0.05 mm

The total thickness: (1.6) mm ± 10%
Flex und Rigid-Flex Technology

Sample 1:

- Layers: 10 (2+6C+2)
- Min. Line Width: 0,109mm
- Min. Line Space: 0,102mm
- Min. PTH: 0,2mm
- LP Thickness: 1 +/-0,1mm
- Finish: ENIG
- Dimensions: 90mm x 78,5mm

Sample 2:

- Layer: 7
- Min. Line Width: 0,086mm
- Min. Line Space: 0,201mm
- Min. PTH: 0,25mm
- LP Thickness: 1,6 +/-0,16mm
- Finish: ENIG
- Dimensions: 94mm x 100mm
Flex and Rigid Design - Rules
Radiused Corners within Flex Bend Areas:

- Preferred
- Acceptable
- Not allowed

Reduces / eliminates stress concentrators, improve reliability.
Flex und Rigid-Flex Design Rules

Conductor routing:

When possible, conductors should be routed through bending or flexing areas with the conductors perpendicular to the bend. This will minimize stress on the conductors during flexing and maximize circuit life.
Do not stack conductors on top of each other:

Stacking conductors will essentially increase the overall circuit Thickness thereby decreasing flexibility and the circuit’s ability to bend reliably.

bad

good
Flex und Rigid-Flex Design Rules

Pad fillets:

Before fillets

After fillets

It is a good idea to insert fillets on pads at each location where a conductor enters a pad. Pad fillets will reduce or eliminate potential stress concentration points.
Whenever there is a sharp dimension change or angle, bending can cause excessive stress and lead to cracks in the copper foil in the flexible section of a design.
It is advisable to maximize conductor width wherever possible. For example, if your design requires .005" conductor width to squeeze between pads in an isolated area, the conductor should flare back out to .010”-.012” once the conductor clears the tight area. This will improve the manufacturing etch yields, which in turn means a lower overall circuit cost to you.
Don’t Abruptly Change Widths:

Whenever you have a track entering a pad, particularly when there is an aligned row of them as in a flex-circuit terminator (shown below), this will form a weak spot where the copper will be fatigued over time.
Tear Relief:

This illustration shows the most common and effective methods of eliminating tears in a flexible circuit.
Flex und Rigid-Flex Design Rules

Vias:

- Make annular rings as large as possible
- Vias should be teardropped
- Adding tabs or anchors to vias, as shown, will also help prevent peeling
Flex und Rigid-Flex Design Rules

Vias Locations:

- Vias are not reliable in areas that will flex.
- In a dynamic application, flexed vias can crack very quickly.
- Vias are okay over a stiffener, but vias just off the edge of a stiffener are at risk for cracking.
Flex und Rigid-Flex Design Rules

Distance between Vias and rigid-flex transition area:

Avoid going below 1,25mm/50 mils for high reliability applications.
Flex und Rigid-Flex Design Rules

Stiffener:

Stiffener Considerations:

- Maintain the same stiffener thickness when using multiple stiffeners to lower cost.
- Stiffeners should come to at least two edges of the board
- Stiffeners reinforce solder joints and increase abrasion resistance
- Stiffeners can be used for strain relief and heat dissipation.
Flex und Rigid-Flex Bending

Very common in the field of flex and rigid-flex design is the use of hatched plan layers.

Hatched plan layers increasing the flexibility of Flex and ridged flex PCBs.
Flex und Rigid-Flex Design Rules

Outer Flex layer:

1. Copper to edge spacing:
   - min. 0.3 mm/12 mil for milling
   - min. 0.6 mm/23.6 mil for scoring (with material thickness ≤ 1.6 mm)
   - min. 0.7 mm/27.6 mil for scoring (with material thickness > 1.6 mm)

Source: Haeusermann
Tolerances for interface flexible vs rigid area

\[ R \pm 0.3 - 0.5 \, \text{mm} \]
\[ F + 0 \, -0.6 \, \text{mm} \]
\[ P1 \, \text{min.} \, 1.5 \, \text{mm} \]
\[ P \, \text{min.} \, 0.8 \, \text{mm} \]
\[ H1 \, \text{min.} \, 0.9 \, \text{mm} \]

*) Resin flow 0.2 - 0.4 mm

**R** = rigid area  
**F** = flexible area  
**P** = Pad to flexible area  
**P1** = Pad on flexible area  
**H1** = Hole edge to flexible area

*) Resin flow is based on material thickness and type

Source: AT&S
What determines the flexibility and thus the bending radius of a flexible or rigid-flex circuit?

- The type of materials used
- The thickness of the flex part of PCB

Diagram:
- Compression Zone
- Neutral Zone
- Stretching Zone
Rule of thumb for calculating the bending radius:

1-Layer, Rigid-Flex-PCB: \( r(\text{min}) = 6 \times h \)

2-Layer, Rigid-Flex-PCB: \( r(\text{min}) = 10 \times h \)

Multilayer, Rigid-Flex-PCB: \( r(\text{min}) = (10-15) \times h \)

Strong dynamic loaded rigid-flex PCB: \( r(\text{min}) = 25 \times h \)

“h” is the thickness of the flexible PCB

Calculate 150µm for a 1-layer, and 200µm for a 2-layer printed circuit board. It guarantees are already included.

Example:

A 2-Layer rigid-flex PCB is 200 microns thick. From the formula above:

\[ 10 \times 200 \text{ microns} = 2000\mu\text{m} = 2\text{mm}. \]
Bookbinder bending:

Differential Lengths (Multilayer and Rigid Flex)

The bookbinder design of an unbonded flex area can be used in regions where a sharp bend (radius to thickness ratios <6) is required. This technique uses progressive lengths in the flex area (see Figure 5-12) and is costly to manufacture because of tooling complexity, processing difficulties, and reduced yields. Corresponding calculation See: IPC-2223
Bookbinder bending:

Differential Lengths (Multilayer and Rigid Flex)

To work out the additional length needed for each flexible circuit layer above the innermost layer, use the calculation given by IPC-2223:

\[ L_i = \frac{\theta \pi}{180} \cdot R_i + L_1 + L_2 \]

Then, the next flex circuit bend is given by:

\[ L_0 = L_i + \frac{\theta \pi}{180} \cdot H \]
Semiflex Bend-Radius:

Calculation of the necessary length \( L \) of Bending range:
\[
L = \text{angle} \times \text{radius} \times \pi / 180° + 2 \times 0.4 \text{ mm}
\]

<table>
<thead>
<tr>
<th>Angle</th>
<th>Bend Radius [mm]</th>
<th>Length L Bending area</th>
</tr>
</thead>
<tbody>
<tr>
<td>45°</td>
<td>4</td>
<td>3.9</td>
</tr>
<tr>
<td>90°</td>
<td>7.1</td>
<td>4.7</td>
</tr>
<tr>
<td>180°</td>
<td>13.4</td>
<td>16.5</td>
</tr>
</tbody>
</table>
Handling of Flex and Rigid-Flex PCBs
Handling of Rigid Flex PCBs

Baking (drying) of the rigid-flex circuits is essential before soldering.

Polyimide films are very hygroscopic (they absorb a lot of moisture). Even under standard room conditions, films that have already been dried will absorb moisture from the air and will reach their saturation level again within a few hours (up to max. 3%).
During the soldering process, the absorbed moisture can lead to delamination, blistering or breaks as a result of the thermal stress.
Handling of Rigid Flex PCBs

PCB Maximum Moisture Content

Unacceptable Range

Acceptable Range

Reflow Temperature (°C)

MMC (%)
Handling of Rigid Flex PCBs

DuPont Pyralux® Flex materials Baking Recommendations Prior to Reflow
We recommend that boards made with Pyralux ® Flex materials are baked prior to exposure to solder processes (e.g. solder leveling and reflow). Boards are generally baked at 250 °F (121 °C) from two to ten hours, * depending on the board thickness and design. Baking removes any moisture that may have been absorbed during processing. Polyimide films absorb moisture quickly; therefore, soldering and reflow should be done within 30 minutes after baking.
Vacuum ovens are also used to remove water. Lower temperatures, such as 150-175 °F (65-80 °C) can be used. This method also reduces the oxidation of the exposed copper pads.
Boards should be baked prior to soldering by hand, wave, IR and Vapor Phase soldering. This bake is typically done at 250 °F (121 °C) for two to ten hours, * depending on the board thickness and design.

Note: Moisture Absorption
Kapton® NH: 2.8%
Pyralux® LF: 1.8%
Pyralux® RF: 1.8%

* Times may vary based on type of materials in board, layer count, % copper ground planes, size of board, room/area conditions (%RH) etc...

Depending on the thickness of the printed circuit board Dupont recommends a drying time of 2 to 10 hours at 120 °C
Soldering

Baked rigid-flex PCBs can be soldered either manually or mechanically no later than 6 to 8 hours after drying. The techniques that are commonly used with rigid PCBs, such as infrared, convection and vapour-phase soldering, can also be applied to rigid-flex circuits.
9 Golden Rules
Golden Rules

- Communicate with the fabricator!
- 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.
THANK’S FOR LISTENING

Danke für’s Zuhören