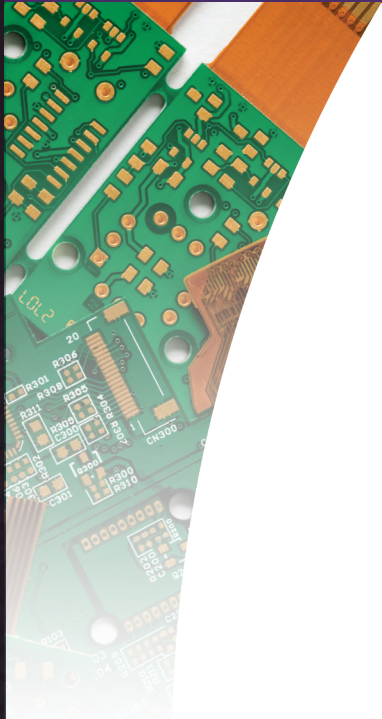




# FLEXIBLE & RIGID-FLEX PCBs



Flex-circuits (FPCs) are a reliable alternative to conventional wiring. By eliminating bulky wires, flex-circuits provide a lighter, cleaner and neater appearance along with the capability of assuming 3D configurations. Flex-circuits have many advantages:

- They improve connection reliability
- They simplify assembly
- They improve component appearance
- They only fit one way for fewer wiring errors during installation and servicing
- They reduce rework and trouble-shooting time

## Material Properties Comparisons for FPCs

Substrate Material	Dielectric Constant	Dissipation Factor	Dielectric Strength	Moisture Absorption	Tensile Strength	Elongation
Polyester	3.2	0.005	7000 v/mil	< 0.08%	25 kpsi	~120%
Polyimide	3.5	0.003	7000 v/mil	1.3–3.0%	25 kpsi	~60%
PEN	2.9	0.004	7500 v/mil	1.0%	30–35 kpsi	~75%

Common Substrate Thickness: 25 µm / 50 µm / 75 µm

### Cover Material

A thin dielectric material applied to the outside layers of the circuit to insulate the copper conductor. Types available:

#### Coverlay/Coverfilm

- Combination of film (polyester or polyimide) and adhesive
- Insulated film applied by hot laminating

#### Cover Coating:

- Photo-imageable covercoat (LPI) is produced by a photo controlled process and are used for tight pad spaces

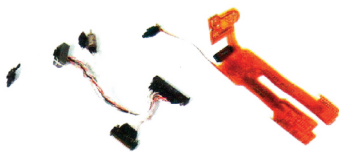
#### Stiffener:

- Used in circuits for applications that require support in areas where connectors or other components are applied
- Used as a carrier panel for automated assembly processing

- Raw materials: PET PEN, PI, FR4, Metals, Aluminum
- Stiffener and coverlay termination points should overlap a minimum of 0.030" (0.76 mm) to avoid stress points

### Capabilities:

- Holes size by CNC Drilling: 0.30 mm
- Holes size by punching: 0.50 mm
- Track width/spacing: 150 µm
- Cutting by CNC milling or soft tool with cutting blades: +/- 0.40 mm
- Cutting by hard tool: +/- 0.25 mm (tolerance)
- Tracks location and outline: +/- 0.15 mm



*Above Left:* Before — A tangle of wires connects four circuit boards

*Above Right:* The flex circuit solution — A single circuit with 7 stiffeners and 2 connectors provides all the needed interconnects

**IMP can offer you many solutions for your flexible PCB application requirements**



# FLEXIBLE & RIGID-FLEX PCBs

## Technical suggestions

### To avoid delamination:

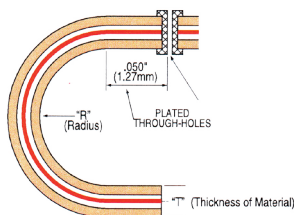
- Increase the pad size without increasing the solder mask opening
- Incorporate tie-downs into the design

### Filleting:

- All pads, on both through-hole and surface mount pads, should be filleted to reduce stress points
- This helps eliminate breaking during flexing

### Bending:

- Bend radius should be approximately 10 times the material thickness and 1.27 mm away from the plated through hole
- Avoid putting SMD components in the bending area



### Slots:

- Avoid rectangular holes for small quantities
- Minimum space between board edge and tracks: 0.5 mm

## Installation Uses Classified as per IPC-6013

**Use A:** Capable of withstanding flex during installation

**Use B:** Capable of withstanding continuous flexing for the number of cycles as specified in the procurement documentation

**Use C:** High temperature environment (over 105 °C)

**Use D:** UL Recognition

## Key IPC Standards for Flex Circuits

**IPC-2223:** Sectional Design Standard for Flexible Printed Boards

**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-6013:** Qualification and Performance Specification for Flexible Printed Wiring

**MIL-P-50884:** (inactive for new designs), Printed Wiring, Flexible and Rigid-flex for Electronic Printed Wiring

**MIL-PRF-31032/3A:** Printed Wiring Board, Flexible, Single and Double Layer, With or Without Plated Holes, With or Without Stiffeners, for Soldered Part Mounting

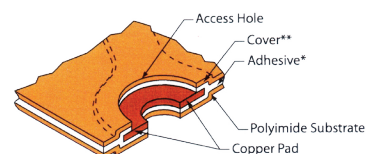
**MIL-PRF-31032/4A:** Printed Wiring Board, Rigid-Flex or Flexible, Multilayer, with Plated Holes, With or Without Stiffeners, for Soldered Part Mounting

## Soldering Considerations

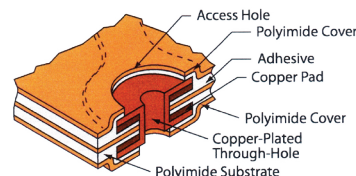
- Since polyimide absorbs moisture, circuits must be baked (1 hour @ 120°C) before soldering (reference IPC-FA-251 Assembly Guidelines for Single-Sided & Double Sided Flexible Printed Circuits, IPC-TM-650 Test Methods Manual).
- Pads located in large conductor areas; such as ground planes, voltage planes, or heat sinks; should be provided with relief areas. This limits heat dissipation for easier soldering.
- When hand soldering pins in dense clusters try not to solder adjacent pins one after another. Move around to avoid local overheating (refer IPC-FA-251).

## Flexible Circuit Types as per IPC-6013

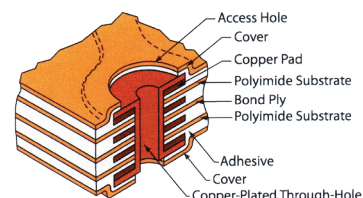
**Type 1:** Single-sided flexible PBs containing one conductive layer, with or without stiffeners



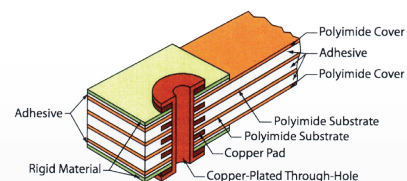
**Type 2:** Double-sided flexible PBs containing two conductive layers with PTHs, with or without stiffeners



**Type 3:** Multilayer flexible PBs containing three or more conductive layers with PTHs, with or without stiffeners



**Type 4:** Multilayer rigid and flexible material combinations containing three or more conductive layers with PTHs



**Type 5:** Flexible or rigid-flex PBs containing two or more conductive layers without PTHs

