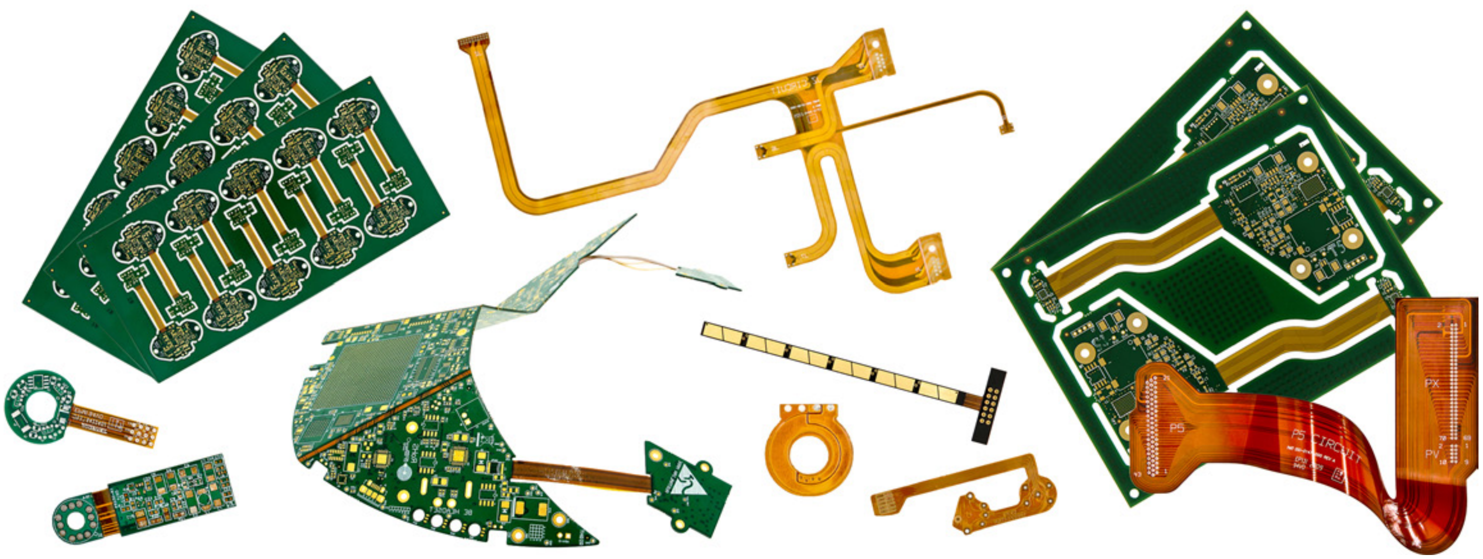
Engineered Technologies

FLEXIBLE & RIGID-FLEX CIRCUITS

TECHNICAL ENGINEERING GUIDE



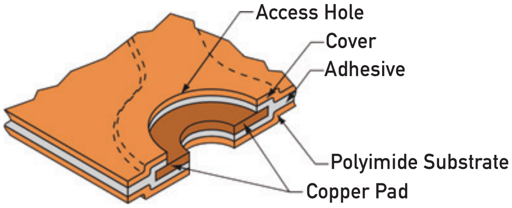
Delivering Quality Since 1952.

FLEX CIRCUIT DESIGN

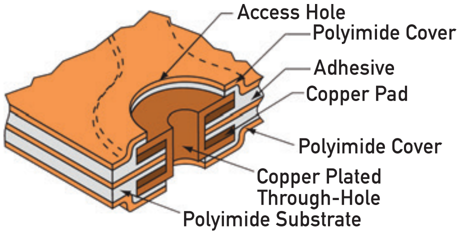
Superior Packaging Options
Flex circuits can be shaped to fit where no other design can. They are a hybrid of ordinary printed circuit boards and round wire, exhibiting benefits of each. In essence, flex circuits give you unlimited freedom of packaging geometry while retaining the precision density and repeatability of printed circuits.

Epec has over 60 years of experience in building Printed Circuit Boards. Let our expertise in engineering and manufacturing assist you in meeting your Flexible Circuit Board needs.

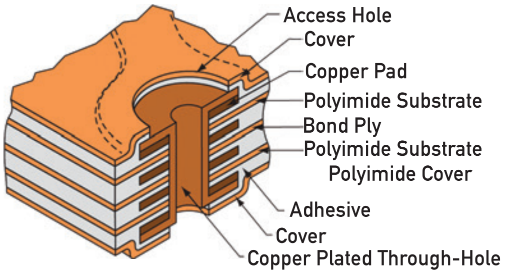
TYPES OF FLEX CIRCUITS



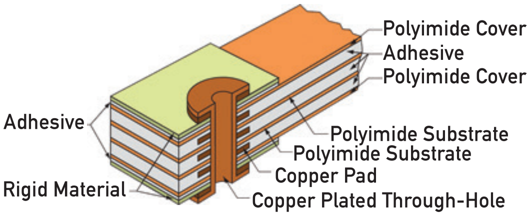
Single-Layer
- IPC 6013 - Type 1
- One conductive layer either bonded between two insulating layers or uncovered on one side.
- Stiffeners, pins, connectors, components, are optional.



Double-Sided
- IPC 6013, - Type 2
- Two conductive layers with an insulating layer between; outer layers may have covers or exposed pads.
- Plated through-holes provide connection between layers.
- Stiffeners, pins, connectors, components are optional.



Multi-Layer
- IPC 6013 - Type 3
- Three or more flexible conductive layers with flexible insulating layers between each one; outer layers may have covers or exposed pads.
- Plated through-holes provide connection between layers.
- Controlled impedance possible.



Rigid-Flex
- IPC 6013 - Type 4
- Two or more conductive layers with either flexible or rigid insulation material as insulators between each one; outer layers may have covers or exposed pads.
- Rigid-flex has conductors on the rigid layers, which differentiates it from multi-layer circuits with stiffeners. Plated through-holes extend through both rigid and flexible layers (with the exception of blind and buried vias). Rigid-flex costs more than a standard circuit with stiffeners.

BENEFITS OF FLEXIBLE CIRCUITS

Rigid Flex circuits gives the ability to design your circuitry to fit the device, instead of building a device to fit the circuit board.

Rigid Flex circuits are designed for the rigors of aerospace, medical and military applications, with dependable reliability. Flex assemblies have the ability to be folded or creased and positioned into the smallest areas making miniaturization of many devices possible, while offering a substantial weight reduction.

Package Size Reduction
- Flexible circuit thinness and light weight will enable a substantial packaging size reduction.
- Flexible circuit’s ability to be folded or creased and positioned into the smallest areas makes miniaturization of many devices possible.

Cost Reduction
- Thin and flexible polyimide film requires a much smaller area, reducing the packaging size and overall material costs.
- Flex circuits used as interconnects reduce the number of connections to be soldered.

Replacement for Wires & Wire Harness Assemblies
- Flexible circuits used to replace wiring reduces the errors common in hand wired assemblies.

High Temperature Applications
- Flex circuits dissipate heat at a better rate than other dielectric materials.
- Expansion and contraction are minimized when using polyimide material.

Weight Reduction
- Substantial weight reduction is a benefit over wires and wire harnesses.

High Density Applications
- Flexible circuits allow for narrow lines giving way to high density device population. Denser device populations and lighter conductors can be designed into a product, freeing space for additional product features.

Reliability & Durability
- The exceptional thermal stability of polyimide allows the circuit to withstand applications with extreme heat, as the materials excellent thermal stability provides a better base for surface mounting than traditional boards. Because the compliant base film places less stress on soldered joints, thermal mismatch is less likely to occur.

ADDITIONAL TECHNICAL INFORMATION

TYPICAL PROPERTIES OF DIELECTRIC MATERIAL FOR FLEXIBLE PRINTED CIRCUITRY

PROPERTY (TYPICAL)	UNITS	POLYIMIDE (Adhesive Based)	POLYIMIDE (Adhesiveless)	POLYESTER
REPRESENTATIVE TRADE NAME		**KAPTON**	**KAPTON**	**MYLAR**
PHYSICAL				
Thickness Range	mil	0.5 to 5	1-6	2-5
Tensile Strength (@25° C)	psi	25,000	50,000	20,000 to 35,000
Break Elongation	%	70	50	60 to 165
Tensile Modulus (@25° C)	100,000 psi	4.3	0.7	5
Tear Initiation Strength	lb/in	1000	700-1200	1000 to 1500
Tear Propagation Strength	g/mil	8	20	12 to 25
CHEMICAL				
Resistance to:				
Strong Acids		Good	Good	Good
Strong Alkalis		Poor	Good	Poor
Grease and Oil		Good	Good	Good
Organic Solvents		Good	Good	Good
Water		Good	Good	Good
Sunlight		Good	Good	Fair
Fungus		Non-nutrient	Non-nutrient	Non-nutrient
Water Absorption (ASTM D570)	% (24 hours)	2.9	0.8	< 0.8
THERMAL				
Service Temperature (min/max)	degree C	-125/+200	-125/+200	-60/+105
Coefficient of Thermal Expansion (@22°C)	PPM/degree C	20	20	27
Change in Linear Dimensions (100° C, 30 min)	%	< 0.3	0.04 - 0.02	< 0.5
ELECTRICAL				
DIELECTRIC CONSTANT (ASTM D150) 1MHz		3.4	3.4	3
DISSIPATION FACTOR (ASTM D150) 1MHz		0.01	0.003	0.018
DIELECTRIC STRENGTH (ASTM D149) @ 1 mil thickness Volume Resistivity (ASTM D257)	V/mil ohm-cm	6000 1.0E+16	6000 1.0E+16	3400 1.0E+16

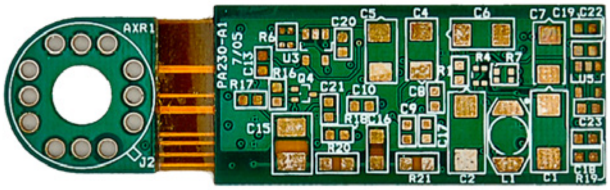
STIFFENERS

Stiffeners are typically used to reinforce selective areas of flexible circuitry for component support, durability and mounting. They can also be used for strain relief and heat dissipation. Bonding materials include pressure sensitive adhesive (PSA) and Temperature sensitive adhesive (TSA).

Stiffener Materials:

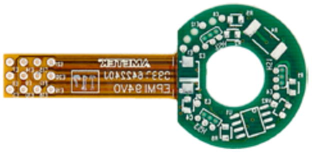
- Unclad FR4 / G10
- Polyimide

Stiffener thickness is determined by your actual needs. Typical stiffeners thickness' range from 0.005" to 0.062".



IMPEDANCE

Impedance is the single most important transmission line property used to determine the performance of a high-speed circuit. Impedance can be controlled with several different configurations including Characteristic, Differential, and Coplanar models.

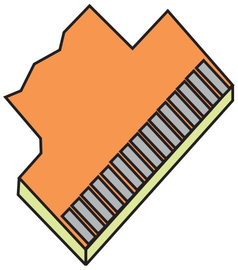


DOCUMENTATION REQUIREMENTS

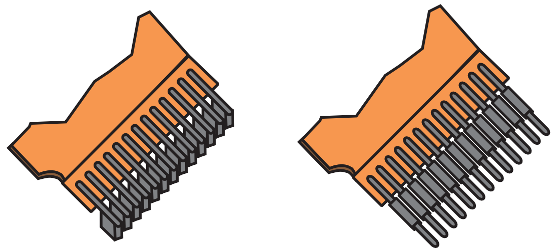
1. The FPCB shall be fabricated to IPC-6013, class (your requirement here) standards.
2. The FPCB shall be constructed to meet a minimum flammability rating of V-0 (if required).
3. The FPCB shall be RoHS compliant (if required).
4. The rigid material shall be GFN per IPC-4101-24 (if using epoxy materials).
5. The flexible copper clad materials per IPC 4204/ (your requirement here).
6. The covercoat material shall be per IPC 4203/ (your requirement here).
7. The maximum board thickness shall not exceed (your requirement here) and applies after all lamination and plating processes. This is measured over finished plated surfaces.
8. The flexible section thickness shall be (your requirement here, do not add this note if this thickness is not critical).
9. Minimum copper wall thickness of plated through holes to be per IPC 6013B.
10. Apply green LPI soldermask (if required) over bare copper on both sides, in the rigid sections only, of the board. All exposed metal will be (your surface finish requirement here).
11. Silkscreen both sides of the board (if required) using white or black (most common) non-conductive epox here.
12. Your electrical test requirements. IPC 9252A recommended.
13. A drill chart is required. This chart depicts your finished hole sizes, associated hole size tolerance and indicates if the hole is to be plated thru or non-plated. A finished hole size of +/- 0.003” is typical.
14. A dimensional drawing is required. All critical dimensions must be noted and the rigid to flex interfaces (this is where the rigid material stops and the flexible material begin) must be defined. Typical outline tolerances are +/- 0.010”.
15. A board construction and layer order is also required. This should show which layers are rigid materials and which layers are flexible material including copper weights.

VALUE ADDED OPTIONS

There are a variety of terminations for flex circuits and we can provide all of these as standard manufacturing process. Adding connectors and other minor component assembly is a common practice when producing flex circuits.

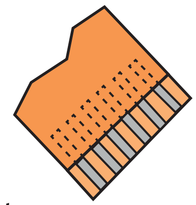


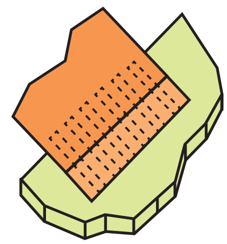
Supported Fingers

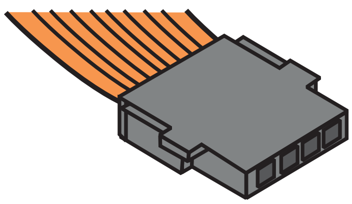


Crimped Sockets and Pins

Lap Joints







Connector

FLEX CIRCUIT CERTIFICATIONS

IPC Information

The following list contains the IPC specifications that you can reference in regards to specific materials, design, performance and assembly questions.

Performance:

- **IPC-6011** - Generic Performance Specifications for Printed Circuits
- **IPC-6012** - Qualification and Performance for Rigid Circuit Boards
- **IPC-6013** - Qualification and Performance for Flexible Circuits

Materials:

- **IPC-4202** - Flexible Base Dielectrics
- **IPC-4203** - Adhesive Coated Dielectric Films
- **IPC-4204** - Flexible Metal-Clad Dielectrics

Design:

- **IPC-FC-2221** - Generic Standard on Printed Circuit Board Design
- **IPC-FC-2222** - Rigid Circuit Boards
- **IPC-FC-2223** - Flexible Circuits

Circuits and Assembly (Quality Guidelines):

- **IPC-A-600** - Acceptability of Circuit Boards
- **IPC-A-610** - Acceptability of Printed Circuit Board Assemblies
- **IPC/EIA J-STD001** - Requirements for Soldered Electrical and Electronic Assemblies

STANDARD MANUFACTURING CAPABILITIES

Circuit Constructions:

- Single-Sided Flexible Circuits
- Double-Sided Flexible Circuits
- Multi-Layer Flexible Circuits
- Rigid-Flex Circuits

Circuit Sizes:

- **Single-Sided**
  Up to 22” by 28” (558.8mm by 711.2mm)
- **Double-Sided**
  Up to 16” by 22” (406.4mm by 588.8mm)

Multi-Layer:

- 12” by 24” (304.8mm by 609.6mm)

Drill Position:

- Tolerance of +/- 0.003” (0.076mm)

Line Width and Spacing:

- Minimum Line - 0.003” (0.076mm)
- Minimum Spacing - 0.003” (0.076mm)

Hole Size Capabilities:

- **Non-Plated Holes**
  min. of 0.006" (0.15mm) drilled hole size
  Tolerance +/- 0.002" (0.05mm)

- **Plated Thru Hole**
  min. of 0.006" (0.15mm) drilled hole size
  Tolerance +/- 0.003" (0.075mm)

Circuit/Blanking Considerations:

- **Soft Tooling**
  Outline dimensions +/- 0.01” (0.254mm)

  Radius of inside corners min. of 0.016” (0.40mm)

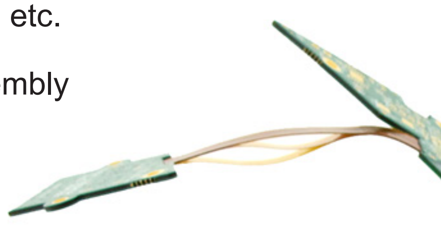
  Edge insulation 0.010” min. (0.254mm)
- **Hard Tooling**
  Outline dimensions +/- 0.001” (0.0254mm)

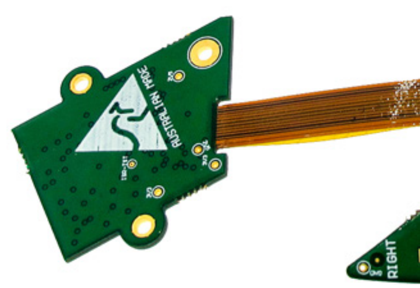
  Edge insulation 0.006” min. (0.152mm)
- **Laser Cut**
  Outline dimensions +/- 0.003” (0.075mm)

  Edge insulation 0.004” min. (0.1mm)

Added Value Capabilities:

- Component / Connector Assembly
- SMT, Thru Hole, BGA, etc.
- RoHS Compliant Assembly
- Precision Stenciling
- Electrical Testing





STANDARD MATERIALS

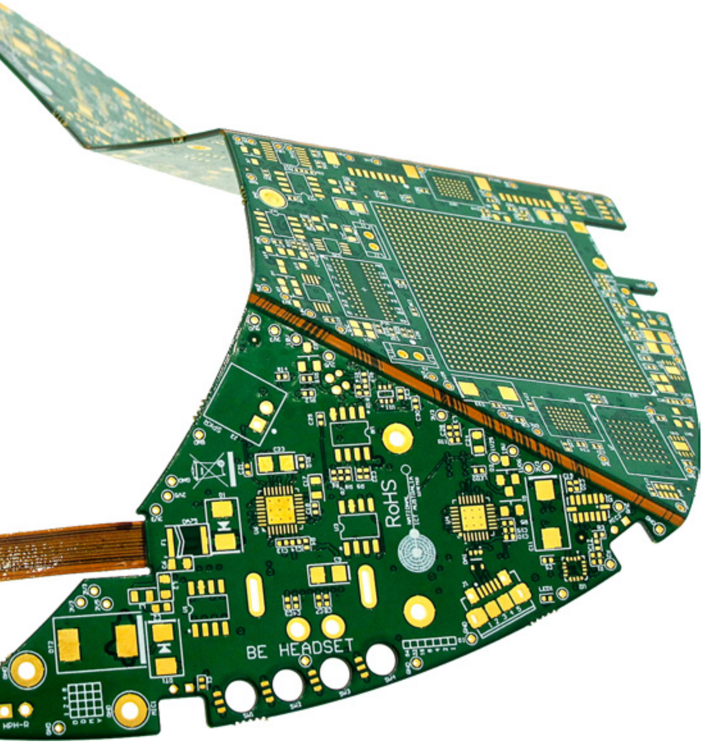
Base Materials:

- **Polyimide**
  0.5 mil to 5 mils (0.012mm - 0.127mm)
- **Polyester**
  2 mil to 15 mils (0.050mm - 0.38mm)
- **Adhesiveless Materials**
  0.5 mil to 6 mils (0.0127mm - 0.152mm)
- **Flame Retardant**
  Laminates and Coverlay

Base Copper:

- 0.33 oz. - 0.00046" (0.012mm) ED Copper
- 0.5 oz. - 0.0007" (0.018mm) RA Copper
- 1 oz. - 0.0014" (0.036mm) RA Copper
- 2 oz. - 0.0028" (0.071mm) RA Copper

RA = Rolled Annealed
ED = Electro-Deposited



Solder Mask:

- **Polyimide Coverlay**
  0.5 mil to 1 mils (0.012mm - 0.0254mm)
- **Polyester Coverlay**
  1.5 mil to 3 mils (0.076mm - 0.228mm)
- **Flexible Photo-imageable Solder Mask**
  Surface Mount and dense applications

Surface Finish:

- ENIG (Electroless Nickel Immersion Gold)
  (RoHS Compliant)
- Tin Plating (RoHS Compliant)
  Immersion (Electroless) and Electrolytic
- Silver (RoHS Compliant)
  Immersion
- HASL (Hot Air Solder Leveled)
- Hard Gold - Electrolytic

Added Value Capabilities:

- FR4-drilled, routed and scored
- PSA (Pressure Sensitive Adhesives)
- Polyimide & FR-4 Stiffeners
- Epoxy Strain Reliefs
- Conductive Silver Ink Shield Layers

Certifications:

- ISO 9001: 2008 Certified
- QS 9000 Compliant
- RoHS Compliant
- IPC Member - Product is manufactured in accordance with the requirements of IPC-6013
- UL certified

COMMON MATERIAL STACKUPS

1 Layer Flex with Optional FR-4 Stiffeners and PSA

	0.001 in	Coverlay
	0.001 in	Coverlay Adhesive
Layer 1	0.0014 in	Copper (1 Oz.)
	0.001 in	Adhesive
	0.001 in	Polyimide
	0.002 in	Adhesive / PSA Adhesive 0.002 in / Adhesive
	0.031 in	FR-4 Stiffener / FR-4 Stiffener

Coverlay through Polyimide: Flex Thickness. All layers: Part Thickness.

Total Part Thickness: 0.038 in — Spec: 0.038 in, +/- 0.004 in

Flex Thickness: 0.005 in — Spec: 0.005 in, +/- 0.002 in

2 Layer Flex with ZIF Contact Fingers

	0.001 in	Coverlay
	0.001 in	Coverlay Adhesive
Layer 1	0.0007 in	Copper (1/2 Oz.)
	0.001 in	Adhesive
	0.001 in	Polyimide
	0.001 in	Adhesive
Layer 2	0.0007 in	Copper (1/2 Oz.)
	0.001 in	Coverlay Adhesive
	0.001 in	Coverlay
	0.002 in	Adhesive / Adhesive
	0.004 in	Polyimide Stiffener / Polyimide Stiffener

Coverlay through Coverlay: Flex Thickness. Layer 1 Copper through Polyimide Stiffener: ZIF Contact Thickness.

Flex Thickness: 0.008 in — Spec: 0.008 in, +/- 0.002 in

ZIF ContactThickness: 0.012 in — Spec: 0.012 in, +/- 0.002 in

4 Layer Rigid Flex (2 Flex Layers)

	0.0005 in	Soldmask (LPI)	
Layer 1	0.0024 in	Base Copper 0.0014 in + Plating 0.001 in min.	
	0.022 in	FR-4 (170 tg min.)	
	0.0054 in	Prepreg (2 x 1080, no flow)	Coverlay: 0.001 in; Coverlay Adhesive: 0.001 in
Layer 2	0.0007 in	Copper (1/2 Oz.)	Copper (1/2 Oz.)
	0.001 in	Polyimide Core (Adhesiveless)	Polyimide Core (Adhesiveless)
Layer 3	0.0007 in	Copper (1/2 Oz.)	Copper (1/2 Oz.)
	0.0054 in	Prepreg (2 x 1080, no flow)	Coverlay Adhesive: 0.001 in; Coverlay: 0.001 in
	0.022 in	FR-4 (170 tg min.)	
Layer 4	0.0024 in	Base Copper 0.0014 in + Plating 0.001 in min.	
	0.0005 in	Soldmask (LPI)	

Flex Thickness (flex area layers). Rigid Thickness (all layers). Epoxy Strain Relief (Optional)

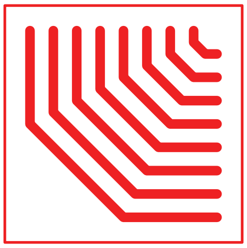
Rigd Thickness: 0.063 in — Spec: 0.062 in, +/- 0.006 in

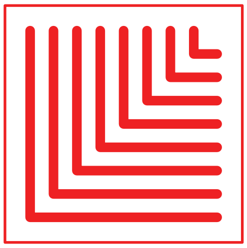
Flex Thickness: 0.006 in — Spec: 0.006 in, +/- 0.002 in

GERBER LAYOUT & DESIGN RECOMMENDATIONS

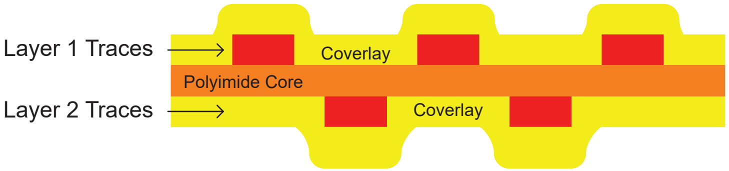
Radiused Corners within Flex Bend Areas
Reduces / eliminates stress concentrators, improves reliability


Preferred

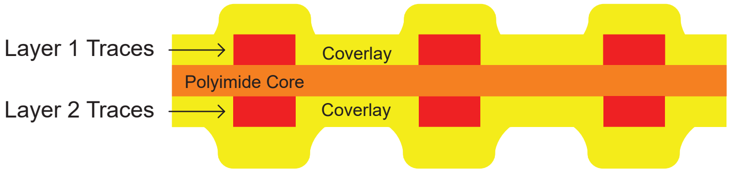

Acceptable


Not Allowed

Staggered Layer to Layer Trace Positioning
Eliminates "I-Beam" effect, improves flexibility and reliability

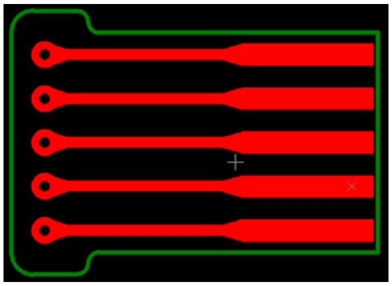


Preferred

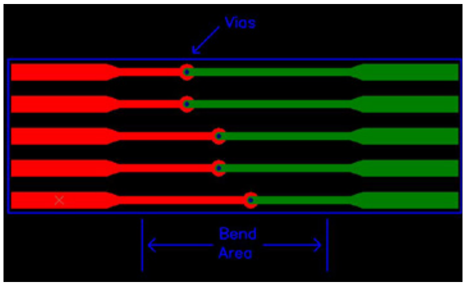


Not Recommended

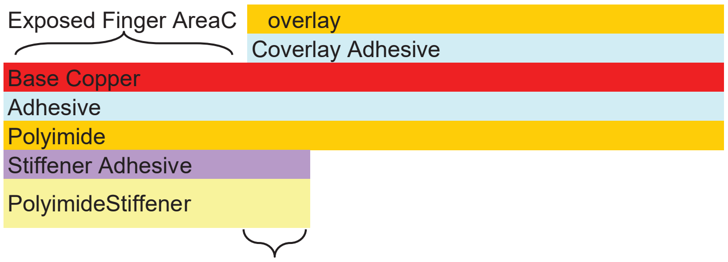
Fillets & Teardrops
Eliminates stress concentrations and improves reliability



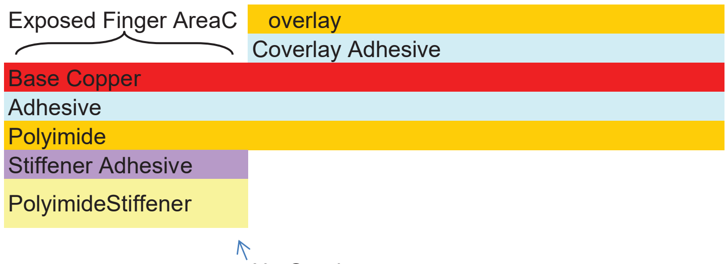
Vias within Bend Areas
Not recommended, significant stress concentrator, May lead to breakage



Stiffener & Coverlay Terminations
Prevents creation of significant stress concentrator



Required



Not Allowed

FLEX & RIGID-FLEX CIRCUITS GLOSSARY

Access Hole:
An opening in a layer of dielectric material that provides access to a land on a conductive layer of the flexible circuit.

Adhesions (pressure sensitive tape):
The bond produced by contact between pressure-sensitive adhesive and a surface.

Adhesive:
A substance, typically an epoxy or acrylic glue, used to adhere coverlays, stiffeners, etc to a flex circuit.

Array:
A group of circuits arranged in rows and or columns on a panel for component assembly purposes.

Artwork:
An accurately-scaled configuration that is used to produce the conductor pattern.

Base Material:
The insulative material upon which a conductive pattern may be formed. The base material may be rigid or flexible.

Bend Ratio:
A relationship between the thickness of material bent to the radius over which it is bent.

Bending Resistance:
The ability of a material to withstand repeated bending to specified parameters without producing cracks or breaks in excess of the specification allowance.

Bonding Layer:
An adhesive layer used in bonding together plies of dielectric and conductive materials during lamination.

Cap Lamination:
A process of making multilayer printed boards with surface layers of metal-clad laminates bonded in a single operation.

Conductor Layer:
The total conductive pattern formed on one side of a single layer of base material.

Coverfilm (Coverlayer):
A film of dielectric material with adhesive which is bonded over the etched conductor runs to insulate them.

Dielectric Strength:
The maximum voltage that a dielectric can withstand under specified conditions without resulting in a voltage breakdown, usually expressed as volts per unit of dimension.

FLEX & RIGID-FLEX CIRCUITS GLOSSARY

Dynamic Flex:
A flexible circuit designed to move during operation.

Edge Spacing:
The distance of an etched feature from the edges of a printed circuit board.

Flexible Multilayer Printed Board (Type 3):
Multilayered printed board made of only flexible materials. Different areas of the multilayer printed board may have different numbers of layers and thickness.

Flexible Printed Board (Circuit):
A printed board made only of flexible materials. Typically Polyimide or Polyester based.

Insulation Resistance:
The electrical resistance of an insulting material that is determined under specific conditions between any pair of contacts, conductors or grounding devices in various combinations.

Polyimide:
The synthetic polymer that has more than two imide radicals in the main chain.

Prepreg:
A sheet of material that has been impregnated with a resin cured to an intermediate stage (i.e. B-Staged Resin).

Sequential Lamination:
The process of manufacturing multilayer printed circuit boards in which multiple double-sided layer stacks with interconnecting holes between conductive patterns on both sides are laminated or combined, after which additional layers are attached to the partially completed board stack up.

Static Flex (Flex-To-Install):
A flexible printed board designed to be bent for installation purposes only (not in operation).

Steel Rule Die:
A piece of tooling made from a hardwood base with hand-formed steel rule placed into a laser burned path which is used to profile portions of or the entire final shape of a flex circuit.

Stiffener Board:
A material fastened to the surface of a flexible printed board to increase its mechanical strength.

Thermal Cure:
A chemical reaction using heat energy that hardens organic substances such as adhesives and coating materials.

Window (in the coverlayer):
An opening in the dielectric of a flexible printed board that exposes conductors.

ASSEMBLY PRE-BAKE SPECIFICATIONS

The need for an assembly pre-bake cycle is an industry accepted standard and a byproduct of the hydroscopic properties of flexible circuit materials. This property is independent of which material manufacturer is used. Entrapped moisture may flash to steam during assembly and out-gas with considerable negative results.

Applying a Pre-Bake cycle and eliminating retained moisture will improve assembly yields, product reliability and prevent delamination and blistering.

Flex Circuit & Component Preparation
Source - "Flexible Circuit Technology, Third Edition" by Joseph Fjelstad

As the principle elements of a flex circuit assembly, the flex circuit and components should be properly prepared. One thing that most plastic components and flex circuits have in common is that they take up moisture.

This means that they must be protected from humid environments. Otherwise, they must be pre-baked to prevent explosive outgassing of trapped moisture and the creation of defective conditions such as cracked components or blistering and delamination of the coverlayer.

DuPont Pyralux® Flex materials Baking Recommendations Prior to Reflow
Source - From the Pyralux® Technical Manual page 5.23

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.***

AMERICA'S OLDEST. A HISTORY OF INNOVATION.

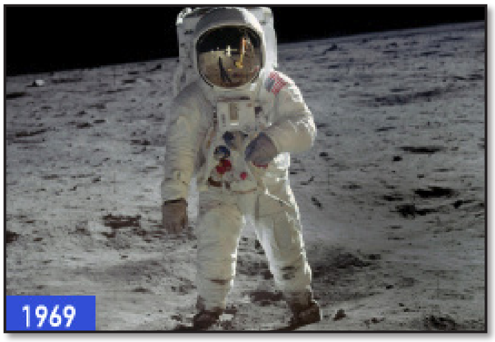
Since 1952 Epec has been connected to the development of the PCB and the electronics industry.

Epec was formed through the merger of Electralab and Printed Electronics Corp (EPEC), who were proudly two of the five founding members of the IPC, the 2,900 member trade association supporting the $1.5 trillion global electronics industry.

From pioneering innovation in the PCB industry with R&D, training and setting professional core values, the legacy of Epec has now passed to a new generation of very bright young people, and continues the great tradition of imagination.


1952


1957


1969

CERTIFICATIONS

We are very proud of the high quality products we manufacture. Over the years, we have received an impressive collection of quality awards from customers both large and small.

ISO9001 Certified

ITAR — Compliant International Traffic in Arms Regulations

RoHS3

CONTACT US

Our knowledgeable staff has many years of experience in the industry. We welcome the opportunity to put our skills to work for you! Please contact us with any questions or requests.

North American Headquarters
176 Samuel Barnet Boulevard
New Bedford, MA 02745

Manufacturing Center
2310 Tall Pines Drive, Suite 240-W
Largo, FL 33771

Contact Us By Phone:
Toll Free: (888) 995-5171
Office: (508) 995-5171

Contact Us By Email:
Email: sales@epectec.com

Stay Connected with Epec Engineered Technologies
Join our social community and keep in touch with all our latest
technology investments, current news, upcoming events, and promotions.

Visit Our Website @ www.epectec.com