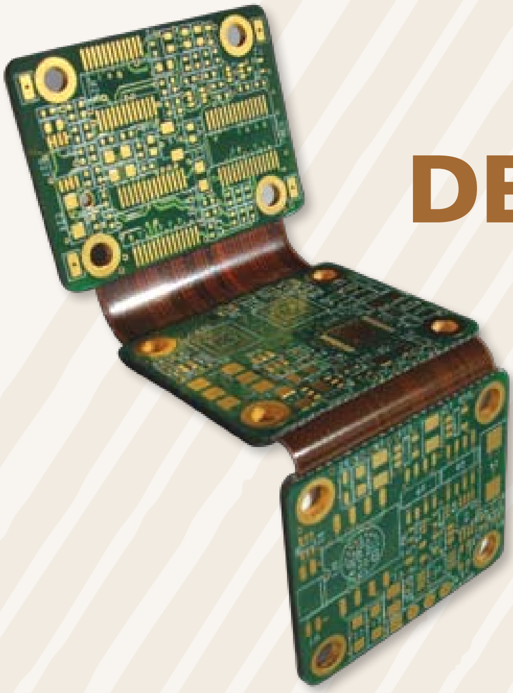
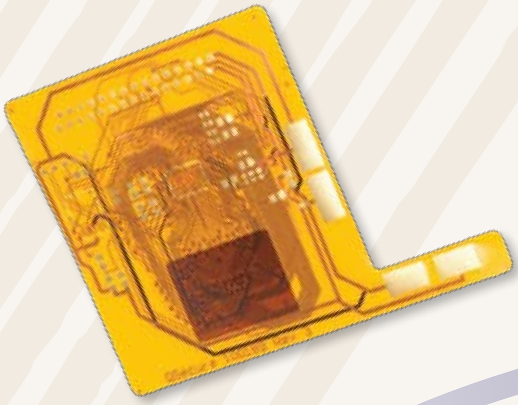
FLEXIBLE CIRCUIT Technologies®

Flexible Circuit and Heater DESIGN GUIDE





WE GO WHERE OTHERS WILL NOT™

ISO 9001:2008, ISO 14001:2004, ISO 13485, TS 16949, ROHS COMPLIANT, ITAR REGISTERED, UL REGISTERED, FDA REGISTERED, IPC MEMBER, BBB MEMBER

FLEXIBLE CIRCUIT Technologies®

TABLE OF CONTENTS

Application Documents

IPC-2152 Standard for Determining Current -Carrying Capacity in Printed Board Design.

IPC-2221 Generic Standard on Printed Board Design

IPC-2223 Sectional Design Standard for Flexible Printed Boards

IPC-6011 Generic Performance Standard for Printed Boards

IPC-6013 Qualification and Performance Specification for Flexible Printed Boards

IPC-4101 Specification for Base Materials for Rigid and Multilayer 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-SM-840 Qualification and Performance Specification of Permanent Solder Mask

*For more information on IPC specifications, contact IPC

3000 Lakeside Drive, Suite 309S
Bannockburn, IL 60015-1249 USA
Tel: 847.615.7100/FAX:847.615.7105
www.IPC.org

Flexible Circuit Technologies (FCT)
9850 51st Ave N. | Plymouth, MN 55442
763-545-3333 (main) | 888 -921-6167 (toll free)
www.flexiblecircuit.com | sales@flexiblecircuit.com

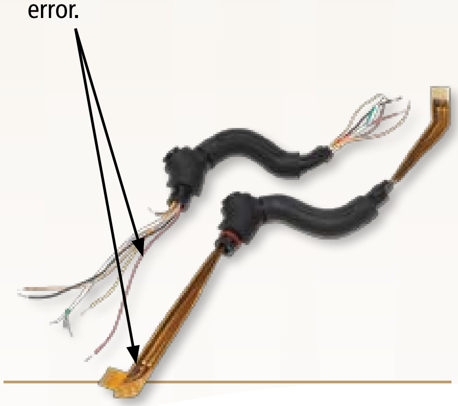
FORWARD

Flexible circuit designs share many of the same challenges as rigid PCB designs, but there are also many differences and additional challenges. The very nature of a flex circuit being able to bend and flex make it as much a mechanical device as an electrical one. This creates a special set of requirements unique to flexible circuitry. Understanding how these requirements interact will allow the PCB designer to create a flex circuit that balances the electrical and mechanical features into a reliable, cost effective interconnect solution. We hope you find this flex circuit design guide a useful tool throughout your design process. We also encourage you to call one of our knowledgeable, experienced Applications Engineers at any time during your design process. They stand ready to assist you at every step to ensure that your flex circuit design is a successful one.

WHY USE FLEX?

There are many reasons to use flexible circuitry as your interconnect choice. Among these are:

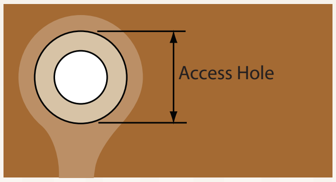
- **Low Mass**—Flexible circuits are only a fraction of the mass of discreet wiring making them ideal for high shock, high vibration applications.
- **High Wiring Density**—Because the conductors in a flexible circuit are photo-defined like a rigid PCB, flexible circuits are capable of very small conductors and therefore ultra-high wiring density. They can take up to 75% less space than a similar wiring harness.
- **The Ability to Bend and Flex**—Perhaps the single biggest reason for using flexible circuits is their ability to bend and flex to fit unique applications.
- **Ease of Assembly**—Every flex circuit is custom to its application, and if designed properly, should fit perfectly and consistently.
- **Termination Options**—Flexible circuits can accept PCB connectors, FFC connectors, and insulation displacement connectors. Plus, several options such as unsupported fingers than can only be done on flex.
- **No Wiring Errors**—Since the conductors on a flex circuit are photo-defined just like a rigid a PCB, there will never be a wiring error.



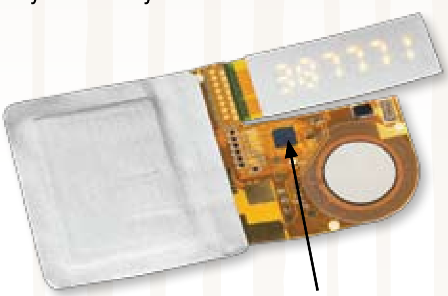
- **Lowest Total Cost of Ownership**—Using flexible circuitry as your interconnect solution gives your designers the freedom to eliminate costly features such as board to board connectors and jumper wires while streamlining assembly time which results in the lowest TCO.

TERMS AND DEFINITIONS

Access Hole: Opening in cover material to allow electrical connection to a conductor.



ACF: (**A**nisotropic **C**onductive **F**ilm) Adhesive films used to electrically and mechanically join conductive surfaces on a flex circuit. These films are available in both thermal setting and pressure sensitive versions and are only electrically conductive in the Z-axis.

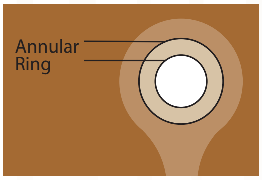


ACF Bonding for flip chip assembly.

Acrylic Adhesive: Thermo-setting film adhesive. The preferred adhesive for flex circuits manufactured in the US. See also Epoxy Adhesive.

Adhesive Squeeze Out: Adhesive that flows out on to a conductive surface during lamination.

Annular Ring: The ring of exposed copper or solder that surrounds a flex circuit's through holes.



Base Material: Copper clad flexible dielectrics, usually polyimide film, with or without adhesive.

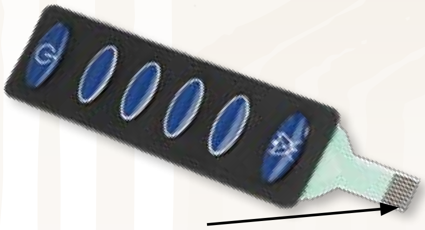
Bend Ratio: The ratio of bend radius to circuit thickness.

Bondply: A combination of insulating material with adhesive on both sides supplied as a film.

Circuit Class: Classes 1-3 based on inspection, testing, and performance requirements.

Circuit Type: Types 1-5 based on layer count, material selection, and vias.

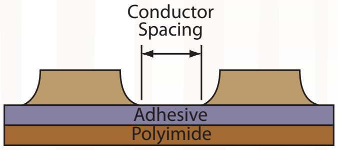
Conductive Ink: Conductive particles, usually silver or copper, suspended in an adhesive carrier, usually epoxy. Can be used to make conductive traces, or as a replacement for a copper shield. Typically more flexible than copper foil.



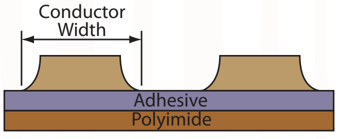
Membrane switch using printed conductive silver ink.

Conductor: The path that carries electrical current from one point to another.

Conductor Spacing: The width of space between conductor strands. A certain minimum conductor spacing must exist in order to prevent conductors from shorting together.



Conductor Width: The width of a conductor measured across its base.



Controlled Impedance: Combining material selection, circuit construction, and circuit feature sizes to yield a predetermined characteristic impedance. Impedance control requirements typically result in a thicker, less flexible circuit.

Cover: The insulating material covering the outer layers of a flexible circuit.

Covercoat: A liquid or semi-liquid insulating material used as a permanent cover over the outer conductive layers.

Coverlay: A combination of insulating material with adhesive on one side supplied as a film.

ENIG: Electroless Nickel Immersion Gold.

ENEPIG: Electroless Nickel Electroless Palladium Immersion Gold.

Epoxy Adhesive: Thermo-setting film adhesive. The preferred adhesive for flex circuits manufactured in Asia. See also Acrylic Adhesive.

Fillets: A flaring of a conductor as it connects to a pad. Used to minimize stress.

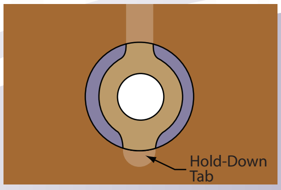
FR4: Common epoxy based hardboard material used to make stiffeners (no copper cladding), or base material in a rigid flex circuit (copper clad).

Gerber: The most common PCB electronic data exchange format. This format is preferred over the other formats such as ODB++, and DXF.

Hardboard: Resin impregnated glass cloth, most commonly epoxy or polyimide resin, with or without copper cladding.

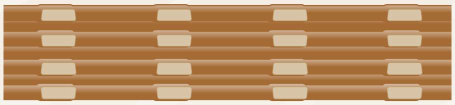
HASL: Hot Air Solder Level.

Hold Down Tabs: An extension of copper on a conductor pad that aids the pad in gripping to the base substrate. Hold down tabs are also referred to as "anchoring spurs".

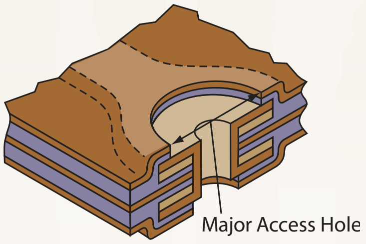


TERMS AND DEFINITIONS

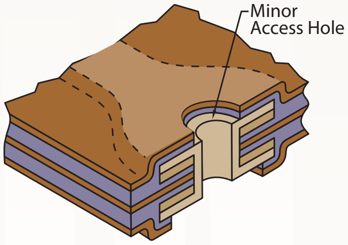
I-Beam Effect: Stacking conductors on top of each other on multiple layers, resulting in a thicker and stiffer circuit. Generally considered poor design practice and should be avoided.



Major Access Hole: An access hole that is large enough to expose a major portion of a conductor pad, which is usually plated with a final finish or coated with solder.



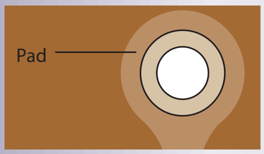
Minor Access Hole: An access hole that exposes only a very small portion of a conductor pad, used on holes where a solder pad is not needed or desired. The cover hole must still be larger than the through hole to allow for normal registration tolerances.



Neutral Bend Axis: Imaginary planar region of flex that does not experience any tension or compression forces when the circuit is bent or folded.

Pad: A conductive land, usually round, and placed over holes drilled for electrical connection.

PIC: Photo Imageable Cover (cover coat).



POP: Pads Only Plating. Refers to a process where copper is plated only in through holes and on pads. Used to reduce thickness and increase flexibility. Also referred to as selective or button plating.

Polyester: Low temp, low cost insulating material.

Polyimide: High temp insulating material available in film, hard board, or B-stage adhesive. Polyimide film is the most common insulating material used in flex circuitry.

Prepreg: Uncured resin impregnated glass cloth used as an adhesive in rigid flex circuits. Resin can be any of a number of types including epoxy, polyimide, BT, etc.

PSA: Pressure Sensitive Adhesive.

PTH: Plated Through Hole.

Punch and Die: A very expensive steel tool used for punching covers, adhesives, and final circuit outlines that is capable of tens of thousands of punches between sharpenings. Also capable of extreme accuracy.

Rigid Flex: A circuit containing rigid PCB boards connected by integral flexible areas where the flexible materials and circuitry run through both rigid and flex areas.

Silkscreen: A processs for applying legend, marking, LPI solder mask, and silver ink conductors.

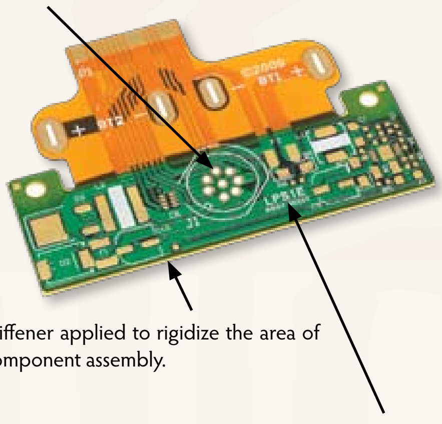
Strain Relief: Usually refers to a bead of semi-rigid adhesive applied along a rigid/flex interface, but can also refer to any of a number of features that can reduce, or eliminate, stress concentration features.

SMOBC: Solder Mask Over Bare Copper.

SRD: Steel Rule Die, an inexpensive tool used to punch covers, adhesives, final circuit outlines, etc. Constructed from a long blade that is formed to a desired shape and then pressed into a laser cut plywood base. Capable of hundreds or a few thousand punches. Capable of moderate accuracy.

Stiffener: A rigid sheet material, usually Epoxy/glass construction or thick polyimide film (.005"), used to rigidize areas of the flex circuit that should not flex.

Plated through holes connect circuit layers and are used for through hole component assembly.



Stiffener applied to rigidize the area of component assembly.

Silkscreen nomenclature can be useful to add identifying markings to circuits.

Termination: The method used to bring electrical signals to/from the flexible circuit. Most commonly connectors, pins, or access holes.

Via: A plated through hole used to interconnect multiple layers of circuitry.

FLEX CIRCUIT CLASSES / TYPES

Circuit Classes:

Flex circuits fall into 3 classes (1-3 per IPC-6013) based on the level of inspection and testing required, and also by the performance requirements of the finished product.

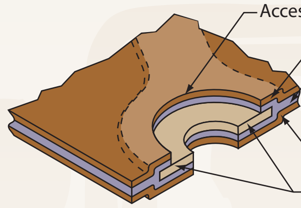
- Class 1 circuits have the minimum inspection, testing and performance requirements. These circuits are the least expensive and are typically used in applications such as disposable electronics (e.g. musical greeting cards) and RFID tags.
- Class 2 circuits have moderate inspection, testing, and performance requirements. Class 2 circuits are more expensive than class 1 and are typically found in applications such as cameras, medical diagnostic equipment, and cell phones.
- Class 3 circuits have the highest level of inspection, testing and performance requirements. Class 3 circuits are the most expensive of the 3 classes, and are typically found in applications that involve the taking or maintaining of life. Applications would include implantable cardiac devices and military/aerospace electronics.

Circuit Types:

Flex circuit type is determined by the number of conductive layers, construction/materials, and the presence or absence of plated through holes. The common flex circuit types (1-4) are illustrated at right. A fifth flex circuit type (type 5) is very uncommon and is not shown. Type 5 circuits are two or more layers without plated through holes.

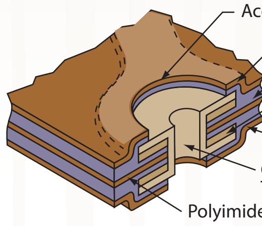
IPC 6013 Type 1

- Single conductive layer
- Insulating material one or both sides
- Access to conductors on one or both sides



IPC-6013 Type 2

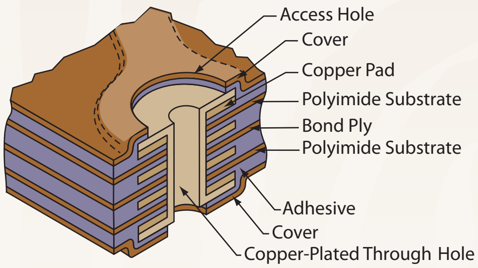
- Two conductive layers with flexible insulating film between them
- Plated interconnect holes
- Insulating cover material on one or both sides
- Access to conductors one or both sides



FLEX CIRCUIT CLASSES / TYPES

IPC 6013 Type 3

- Three or more conductive layers
- Flexible insulating material between layers
- Plated interconnect holes
- Insulating cover material one or both sides
- Access to conductors one or both sides

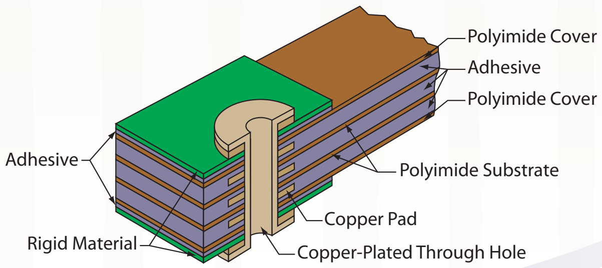


Type 1-Single Layer Flex

Dual access is accomplished on a single sided flex by laser skiving openings on the bottom side of the flex.

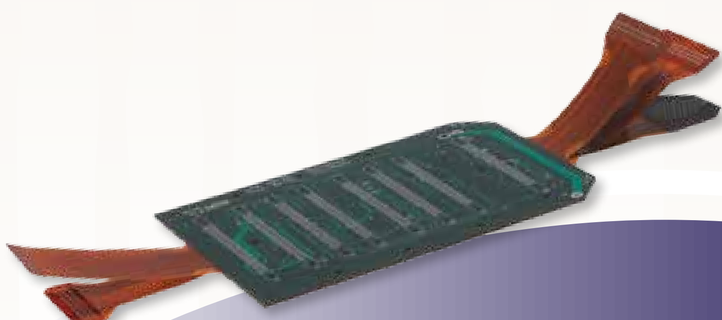
IPC 6013 Type 4

- Two or more conductive layers
- Insulating material may be rigid or flexible
- Plated interconnect holes through flex and rigid materials
- Access to conductors one or both sides through cover material or SMOBC



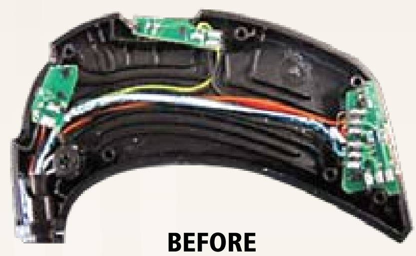
Type 4-Rigid Flex

Rigid flex circuits combine rigid FR-4 areas for dense component population inter-connected with flexible polyimide areas which can be bent to accommodate overall packaging needs.



FLEX CIRCUIT MECHANICAL DESIGN STEPS

Objective: Provide low profile interconnect for 3 small PCBs



BEFORE

AFTER

Step 1: Layout circuit footprint on CAD and create a paper doll.

Step 2: Check paper doll for fit.

Step 3: Create polyester mock-up.

Step 4: Check polyester mock-up for form and fit.

Step 5: Contact flex vendor to make mechanical mock up. This mock up should accurately reflect the actual materials of the final circuit.

Final step: Flex circuit

Review electrical schematic/net list to estimate approximate layer count. Account for all signal and plane layers. Also, refer to the Conductor Width Nomograph (page 11) for any conductors with high current requirements. Multiply the number of layers by .0055" to get the approximate overall thickness of the circuit (if your circuit has controlled impedance requirements, this multiplier may be larger).

Review mechanical requirements/solid model to determine minimum bend radii. Determine and evaluate bend ratio.

Determine flex termination method(s).

Create a "paper doll" of the proposed flex circuit outline. The first paper doll outline can be created with just a ruler and a pencil, but subsequent iterations should then be transferred to a CAD program so that you can keep track of your modifications. Place the paper doll in the assembly to check form and fit. Don't forget to account for the termination hardware. Make modifications as required to optimize fit.

Keep the assembler in mind during the fit check. If the paper doll tears during installation, it may signal possible assembly problems. A flex circuit that is difficult to install will add time (cost) to the assembly, and can be a reliability risk due to possible damage to the circuit during the installation.

Re-create the paper doll using .010" polyester sheet material. You can usually use a standard copy machine to print the circuit outline. Cut the model out and check for form and fit and modify as necessary. The polyester is a bit stiffer than paper and will better represent the mechanical properties of the flex circuit.

Obtain a mechanical sample from your flex circuit vendor. This sample will be constructed from the same materials as the final flex, but will not have any etched circuitry (only solid copper). All component holes and circuit outline features should be represented. Connectors can be glued in place with epoxy to give a true sense of the final fit. This will be the final opportunity to tweak your design prior to ordering actual circuits. If you have followed the steps above, the mechanical sample should require few, if any, modifications.

STANDARD MATERIALS

Flexible Circuit Technologies can work with a wide variety of flex circuit materials to give you the electrical and mechanical performance you require. However, to get the lowest possible cost for your flexible circuit, it is advisable to design your circuit using standard materials whenever possible. Using uncommon materials in your design can add significantly to both the cost and the lead time of your circuit.

Conductive Material:

- Copper Foil ¼ oz (9 um), 1/3 oz., ½ oz., 1 oz., 2 oz.
- Constantan
- Cupro-nickel
- Inconel
- Silver Filled Epoxy
- Carbon
- Aluminum

Insulating Material:

- Polyimide Film .001", .002", .003"
- Polyester Film
- PEN
- PET
- Solder Mask
- PIC

Adhesive:

- Epoxy .001", .002"
- Modified Acrylic .001", .002"
- Prepreg
- PSA
- Adhesiveless

Stiffener Material:

- Glass Reinforced FR4 (epoxy)
- Polyimide Film (non-reinforced) .005"

Final Finishes:

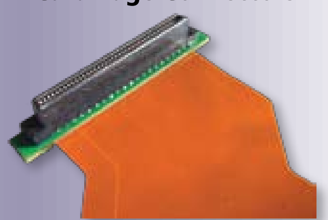
- ENIG
- ENEPIG
- Hard Nickel/Gold
- HASL
- Immersion Tin
- Tin Plate
- Organic (OSP)

TERMINATIONS

Virtually any connector or component that can be mounted on a rigid PCB can also be mounted on a flex circuit. In addition, flex circuitry offers many other options including unsupported fingers and insulation displacement connectors. Contact your FCT Applications Engineer to discuss which termination option will work best for your application.

ZIF Connector Terminal

Card Edge Connectors

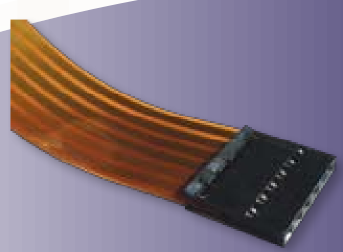


SMT Connector

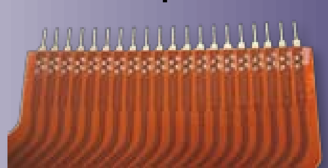
Right Angle Connector

Circular Connector

Insulation Displacement Connectors



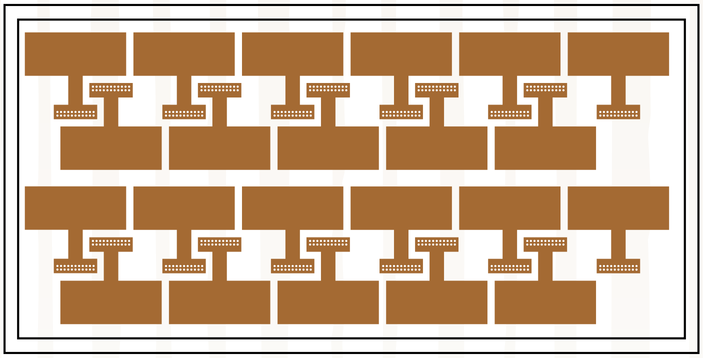
Crimp Pins

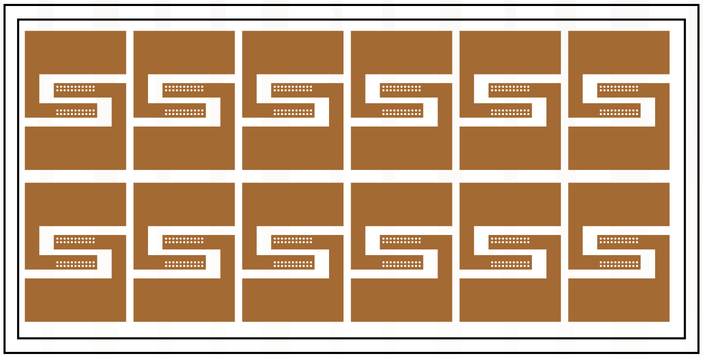


COST DRIVERS

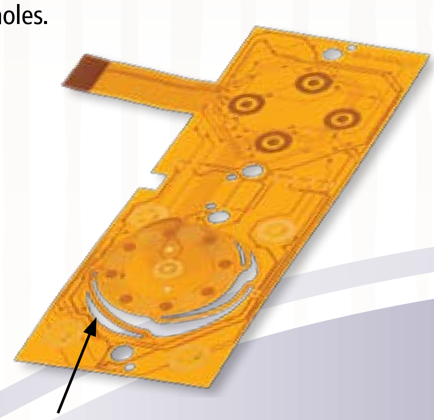
Every designer is looking for ways to decrease costs without sacrificing performance. IPC research has shown that PCB designers drive over 75% of the circuit cost based on the decisions they make. It is imperative that the flex designer understand what features add value and what features add only cost. Designers should never sacrifice reliability to save costs, but at the same time, many flex circuits are over specified resulting in additional costs that add no additional value. Here is a list of the features that drive the majority of your circuit cost:

- **Layer Count**—As the number of layers increase, so does the cost. More layers will require additional materials and processing time. Processing high layer count flex or rigid flex can also be very technically challenging which may result in reduced yields.
- **Circuit Size and Shape**—Most flexible circuits are constructed in panel form. The greater the panel area a circuit occupies, the greater the cost. There are instances where even a small change in outline can result in a large cost decrease. The illustration below shows how a slight modification to the flex shape allows for a better nesting of the flexes on the panel, resulting in two more circuits per panel.





- **Circuit Type (i.e. type 3 vs type 4)**—Rigid flex circuits are typically more expensive than multilayer flex with stiffeners. Scrutinize your design to determine if your application requires a rigid flex construction, or if a multilayer with stiffeners will work. If in doubt, call your flexible circuit manufacturer and ask.
- **Circuit Class (i.e. class 3 vs class 2)**—Class 3 circuits require additional testing, inspection, and construction requirements which make them more expensive. Review the requirements of your application to determine the proper class for your flex circuit.
- **Drawings Overly or Too Tightly Dimensioned**—It is important to remember that you are purchasing a flexible circuit, not a machined part. The materials used to manufacture flexible circuits both permit and require looser tolerances than rigid PCBs. Each dimension placed on a drawing will have to be verified, so ask yourself, "is this dimension adding value, or just cost?". All non-critical dimensions on your flex circuit drawing should be designated as reference.
- **Dissimilar Layer Counts in PTH Areas**—All areas that have plated through holes should have the same layer count and construction.
- **Multiple Final Finishes**—While multiple final finishes can certainly be accomplished, it usually requires a series of hand masking operations that will add cost.
- **Small Features**—Because of the inherent dimensional instability of flex circuit materials, small circuit features (i.e. via pads) can cause processing difficulties and reduced yields. There are instances where it would be less expensive to add additional layers with larger features, than to design with very small features. For this reason, it is advisable to contact FCT early in the design stage for guidance.
- **Blind and Buried Vias**—These are significantly more expensive than through holes.

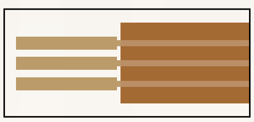


Laser cutting can eliminate the time and cost of tooling during prototyping. It can also be an effective way to cut out unique shapes within a flex circuit.

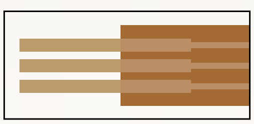
FLEX CIRCUIT DESIGN GUIDELINES

Overview

Scrutinize your design for stress concentration features. Stress concentration features are the predominant single cause for mechanical failures in flexible circuits (i.e. cracked/broken conductors, torn insulating material, etc.). To avoid stress concentration points, the construction of the circuit should not change in, or immediately adjacent to, the bend area. In a bend area, there should be no change of conductor width or thickness or direction, no termination of plating or coatings, no openings in covers or outer insulating materials, and no holes of any kind in a bend zone. If you would like your flex circuit design evaluated for stress concentration features free of charge by an FCT Applications Engineer, call 888-921-6167 or 763-545-3333, or submit your design at www.flexiblecircuit.com



Poor Design-Fingers stop short of cover



Good Design-Fingers extend under cover

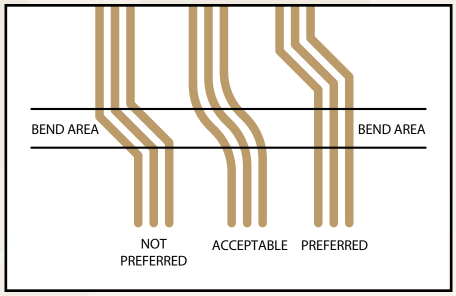
Bend Ratio

Determine and evaluate the minimum bend ratio of your design. This will be your single best indicator of whether your flex circuit may experience problems in service. Bend ratio is bend radius: circuit thickness.

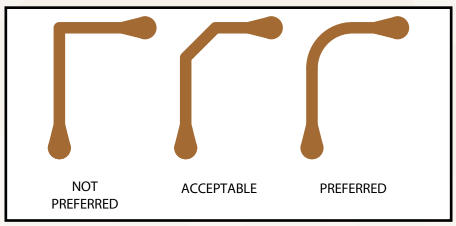
Preferred Bend Ratios Are:	
10:1	Single Layer
10:1	Double Layer
20:1	Multiple Layer

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.



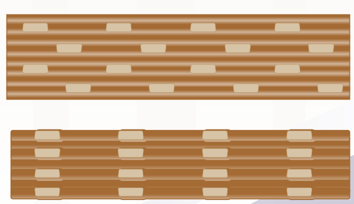
Conductors should always change directions with soft curves rather than sharp corners. When curves are not an option, 2- 45 degree corners are preferred over 1-90 degree corner.



- When possible route small conductors on the inside of a tight bend. Small conductors (<.007") will tolerate compression better than stretching. Placing these conductors on the inside of a bend will reduce or eliminate tension forces.

- Do not stack conductors on top of each other on multiple layers creating an "I beam" effect. Stacking conductors will essentially increase the overall circuit thickness thereby decreasing flexibility and the circuit's ability to bend reliably.

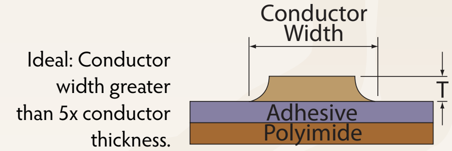
Good



Bad

Conductors

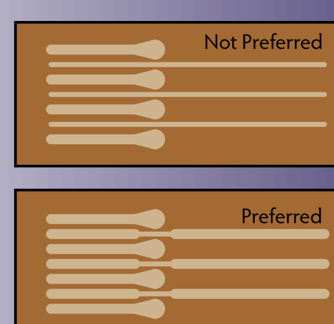
Flexible circuit conductors are manufactured using a photo-etch process which starts with a full sheet of copper. Conductors are formed by masking the desired conductive paths, and then chemically removing all unwanted copper, leaving the desired circuit patterns. As the etchant dissolves the unmasked copper, it also attacks the edges of the conductors resulting in what is referred to as "under-cut".



As copper foil thickness increases, so does the amount of undercut. This makes it very difficult for the flex manufacturer to create very small conductors on very thick foil. There are also variations in the etching process (primarily etchant strength which varies with the amount of copper in the solution). For this reason, the designer must factor in a processing allowance for strand width (and spacing). For optimum etch yields, conductor widths should be at least 5 times greater than the thickness.

Copper Foil Thickness	Plated Copper	Non-plated Copper
¼ oz	+/- .001" (25 um)	+/- .001" (25 um)
½ oz	+/- .001" (25 um)	+/- .001" (25um)
1 oz	+/- .002" (50um)	+/- .001" (25um)
2 oz	+/- .003" (75um)	+/- .002" (50 um)

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.

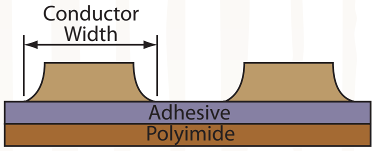


CONDUCTOR WIDTH NOMOGRAPH

The conductor width nomograph on the opposite page will assist you in determining the conductor widths necessary to carry various current loads. The nomograph was reprinted from IPC-2152. Refer to IPC-2152 for a more in-depth analysis of various features and variables that affect current carrying capacity of copper conductors.

Using The Nomograph

1. Find the current matching your requirements on the left side of the upper chart.
2. Move to the right until you intersect the curve that corresponds with the maximum temperature rise allowed. Keep in mind that the temperature rise is from system operating temperature (not necessarily room temp).
3. From the intersection point of the current/temp rise, move straight down to the lower graph to where you intersect the copper weight of the conductors.
4. Move to the left to determine conductor width.



Example #1:If design requires 2 amps @ 10°c temp rise, conductor would be either

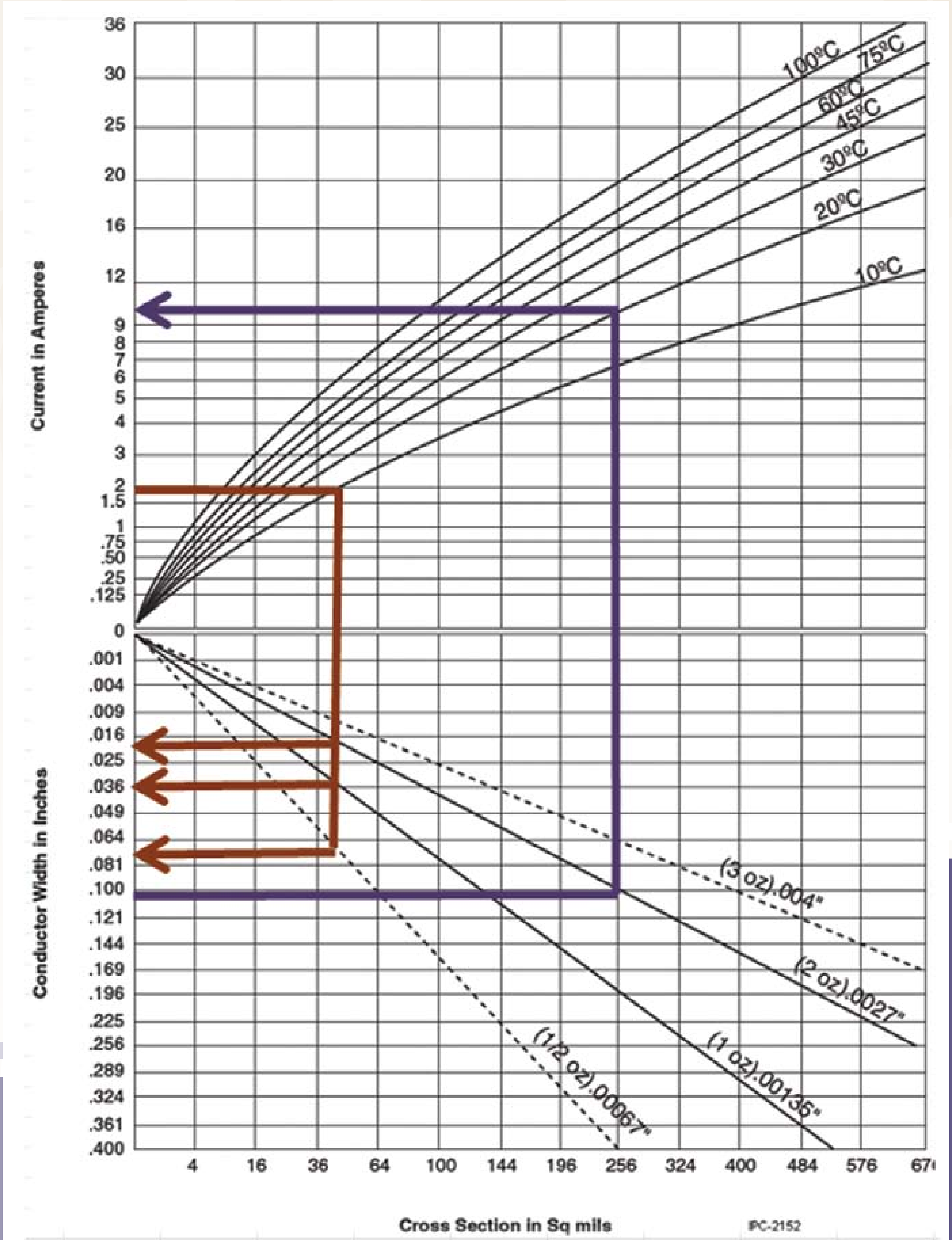
a) .018" wide on 2oz Cu, or

b) .036" wide on a 1oz Cu, or

c) .072" wide on 1/2oz Cu

Example #2: How much current can a .100" wide conductor on 1oz Cu carry with 20° (max temperature rise) @ 10 Amps.

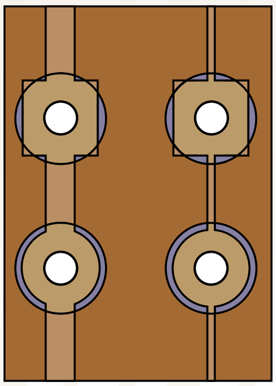
CONDUCTOR WIDTH NOMOGRAPH



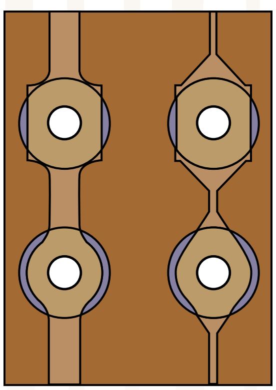
FLEX CIRCUIT DESIGN GUIDELINES

PAD 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.

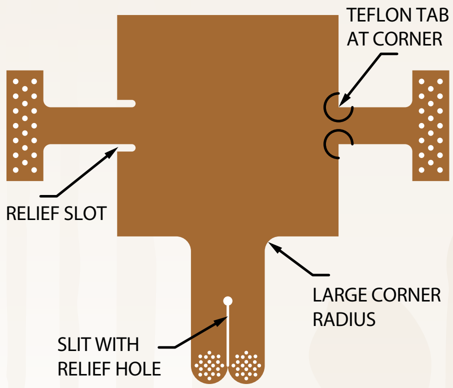


Before Fillets



After Fillets

Tear Relief



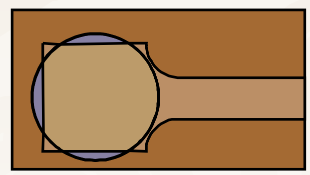
This illustration shows the most common and effective methods of eliminating tears in a flexible circuit. Copper tear stops are not shown because they have been shown to have limited value in keeping a tear from starting or propagating.

Vias

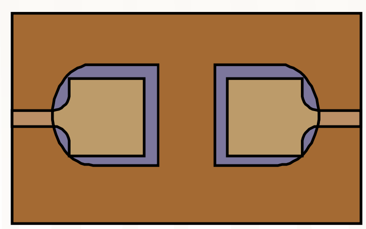
FCT can provide circuits with through hole, blind, or buried vias. Through holes can connect all layers at a via point. Blind vias connect outer layers to adjacent layers, but do not extend through the circuit. Buried vias connect internal layers but do not extend to the outer layers. Blind and buried vias will increase the cost of the circuit, but can increase usable PCB real estate on non-drilled layers.

SMT Access Openings

The two most common cover materials are polyimide film and flexible soldermask. The methods for creating access openings in the 2 materials are very different and carry very different design requirements. Access openings in polyimide film are created by drilling, routing, or punching, which limits the size and shape of the openings to what can be done with a round bit or a tool. For this reason, SMT access openings in polyimide film are either round or oval. Also, gang access of multiple SMT pads is a common design practice on flex circuits.



Flexible soldermask, like regular PCB soldermask, is photo-defined so any shape opening is possible. Soldermask openings should be made slightly larger than the SMT pads to ensure that the mask does not get on the pads if there is any misregistration in the printing process.



FLEX CIRCUIT DESIGN GUIDELINES

Controlled Impedance and Signal Integrity

The speed at which electronic devices are operating is continually increasing. The result is that the characteristic impedance of all parts of the electronic assembly, including any flex or rigid PCBs in the system, need to have matching impedance. Impedance mismatches will cause signal reflections and degradation at each mismatch point, which in turn results in erroneous signals and ultimately device malfunction. The characteristic impedance of a flex can be determined prior to manufacturing using an impedance calculator. An FCT engineer can assist you with these calculations, or you can buy or download an impedance calculator. A number of factors will affect the characteristic impedance of a flex PCB. The main contributors are:

- The dielectric constant of the insulation materials used to construct the circuit.
- The width of the traces carrying the signal.
- The distance of the signal traces from the reference plane layer(s).
- The thickness of the traces carrying the signals.
- The distance between signal traces in differential impedance applications.

The most common impedance requirements range from 50-75 ohms (single ended) or 100-110 ohms differential. Achieving these impedance values in flex circuitry requires the use of thicker dielectric materials than are normally used, resulting in an overall thicker and stiffer circuit.

The bottom shows the construction with higher impedance requirements. The added thickness of the controlled impedance part will make the circuit less flexible.

Plane Layers and Shielding

Reference plane layers and external shielding play a key role in both impedance control and signal integrity. FCT can add plane layers using:

- Additional etched copper layers
- Screened conductive epoxies or inks
- Laminated conductive films

Shielded flex circuits reduce noise and control impedance of signal lines. Shielding can be solid, patterned or cross hatched and can be on one or both sides.

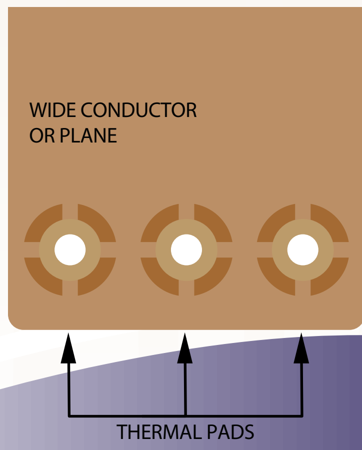
Copper plane layers are the standard for internal planes that require connection through plated vias. Copper planes will cause a flex to hold a pre-form better, while screened epoxies and inks and laminated conductive films will produce a more flexible circuit. Your FCT Applications Engineer can guide you in selecting the best shielding option for your design.

Stiffeners

It is wise to rigidize SMT, connector, and other termination areas on your flex circuit with mechanical stiffeners. FCT can add stiffeners of various thickness made from epoxy glass laminate (FR4) or polyimide film. In SMT applications, stiffeners should be applied to the side opposite the SMT components. On through hole connectors and other through hole applications, stiffeners should be applied to the same side as the connector or through hole component. Stiffeners applied to connector areas will require holes that match the connector footprint. Holes in the stiffener should be sized at least .015" larger than the access hole in the circuit.

Thermal Pads

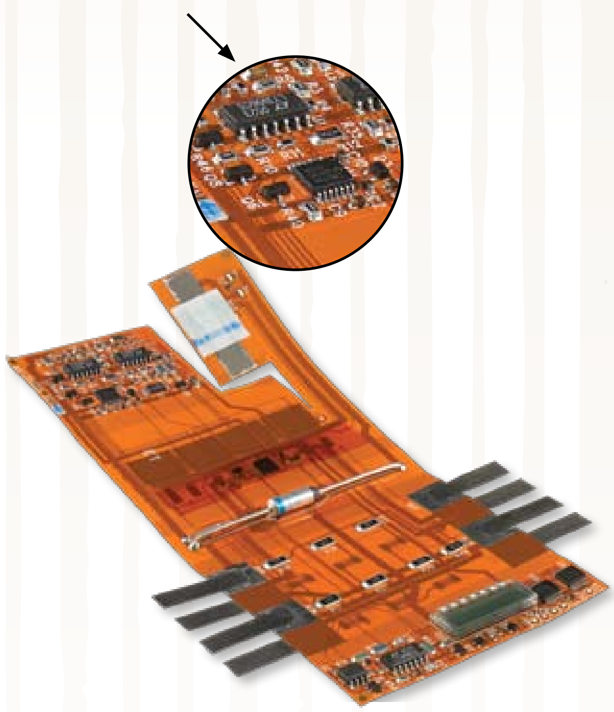
Thermal pads should be used on any solder pad that is surrounded by a large amount of copper. Large areas of copper will sink heat away from a non-thermal pad and make it very difficult to solder.



HANDLING / ASSEMBLY GUIDELINES

- Thoroughly bake flex circuits prior to assembly. The materials used in flexible circuit manufacturing are very hygroscopic. In the right humidity conditions, a flex circuit can near saturation in less than an hour. It is imperative that this moisture is removed prior to the circuit being subjected to elevated temperatures. Moisture is typically removed through an extended baking process (2-6 hours dependent upon circuit thickness and construction) at temperatures between 225F and 275F). After baking, the flex circuits should be processed immediately. If it is not feasible to process the circuits immediately after they are baked, they should be stored in a sealed dry box with desiccant, or in a nitrogen chamber until they can be processed (which should be 24 hours or less).

Component Assembly - we offer through hole and surface mount capabilities, as well as circuit testing, and electrostatic protective packaging.



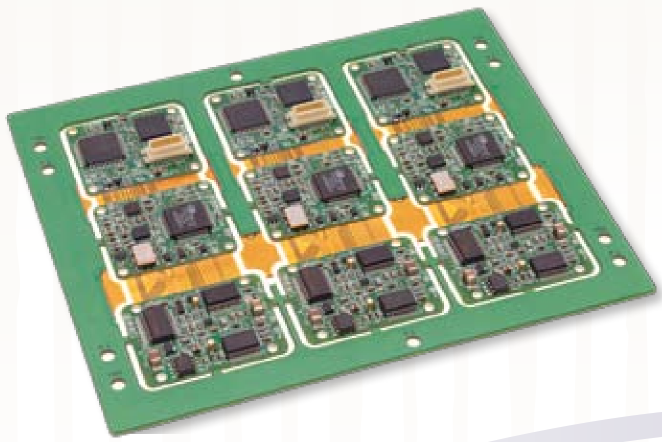
- Flex circuits should be formed at the very end of the assembly process. After a circuit is formed, it should not be subjected to any elevated temperature. Elevated temperatures will cause the circuit materials to soften and the bend will relax.

- Any bend in a flex circuit that exceeds 10:1 bend ratio on single and double sided circuits, or 20:1 on multilayers, should be formed only once. Once the part has been formed, it should not be opened and reformed, or exercised in any way. Bends with tight ratios will permanently stretch the copper conductors on the outside of the bend. If the circuit is flattened, the copper will not recompress. Rather, the copper will ripple. Reforming or exercising the bend will make the conductors alternately ripple and flatten causing the copper to become brittle. Brittle conductors will ultimately lead to cracks and failures.

- Make sure that your reflow temperature profile is matched to flex circuit materials. Due to their low mass and relatively low temperature ratings, flex materials cannot withstand, nor do they require, the elevated temps and durations of standard rigid PCB profiles.

- Utilize a carrier or transport system for your flex circuit during the assembly process. Flex circuit materials are not as durable as rigid PCB materials, and are more prone to damage due to careless handling. FCT can provide custom shipping trays that can also be used as carrier trays during the assembly process.

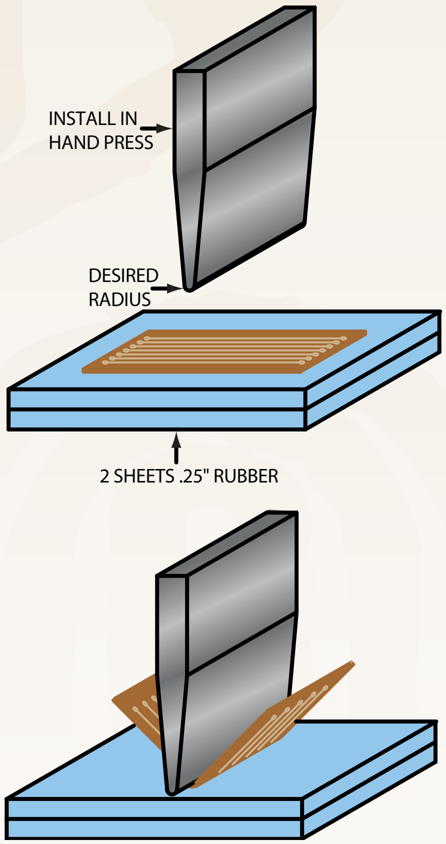
Circuits can be provided in panel form to allow for subsequent SMT assembly. Circuits are held in panel with breakout tabs that allow for easy depanelization after assembly.



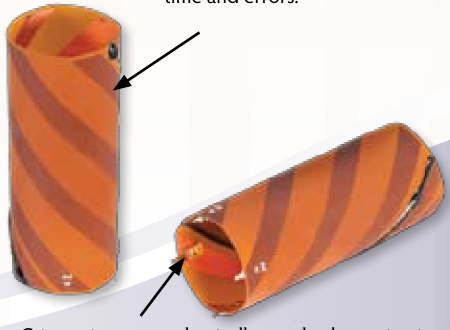
CIRCUIT FORMING GUIDELINES

Probably the single biggest reason for using a flexible circuit for your interconnect needs is that it gives you the ability to form and shape the circuit to fit in your application. However, simply using flex materials does not guarantee that the circuit can be formed to any shape. In many cases, a custom forming tool is required to ensure that the circuit can be repeatably and reliably formed.

- If your required bend ratio is less than 10:1 for single or double sided circuits, or less than 20:1 for multi-layers, you will want to create custom forming tooling. Depending upon the complexity of the bending and forming, this tooling can be constructed from plastic or metal. An experienced FCT engineer is available to assist you in designing your forming tools.
- Circuits are best formed cold. Flex circuits become very fragile when they get hot, so it is advisable to form your circuits cold whenever possible.
- If a bend is relaxing too much after cold forming, heat can be added to the process to make the bends hold their shape better. The circuit should still be loaded into the forming fixture cold (room temperature), and then the entire assembly should be placed in an oven for just long enough to bring the forming tooling up to temperature. The best oven temp is the lowest temperature that works for your application. The assembly should then be removed and allowed to cool back to room temperature before the circuit is removed from the tooling.



Pre-formed circuits reduce subsequent assembly time and errors.



Crimp pins are mechanically attached to a circuit to allow for soldered connections.

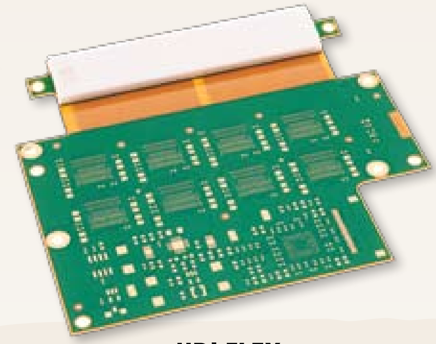
FCT CAPABILITIES

Flexible Circuit Technologies

We know how difficult it can be to find a supplier that is experienced in a wide variety of industries and flexible enough to take on any technical challenge. What makes FCT different? On the front end, our engineers have a wealth of experience in unique applications and a desire to solve problems that others will not. We have domestic and international production capabilities to bring design to reality, and if necessary we can add a dedicated manufacturing line to meet your unique product needs. Finally, with our inventory stocking program, your products can be built in quantities required to effectively meet your business objectives, while being delivered in quantities and time frames desired by your production facility. Like our motto says "We Go Where Others Will Not!"

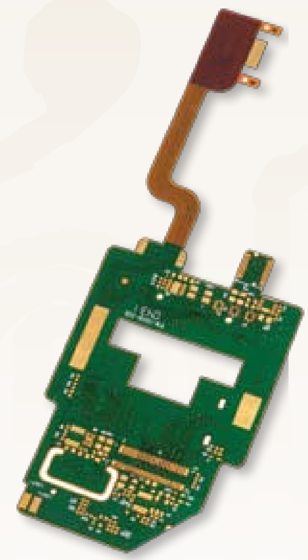
Single, double, multilayer flex circuits, as well as rigid flex circuits, can be designed with dozens of different conductors, adhesives, insulation layers , finishes, connectors and more. The combinations are nearly endless and are limited only by the designer's imagination.

MULTILAYER FLEX

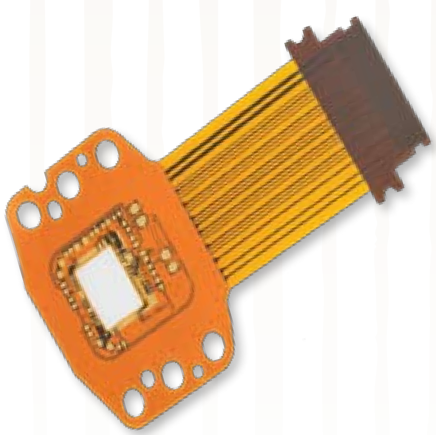


HDI FLEX

HEAVY COPPER FLEX



RIGID FLEX



SINGLE/DOUBLE LAYER FLEX

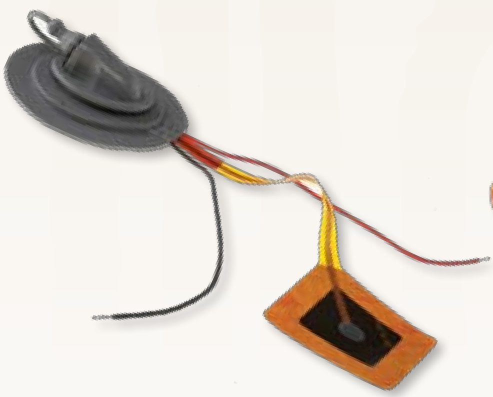
UNBONDED FLEX

MEMBRANE SWITCH

FCT CAPABILITIES

GRAPHIC OVERLAY

FLEXIBLE COIL



OVER MOLDING

FLEXIBLE HEATER

FLAT FLEXIBLE CABLE

COMPONENT ASSEMBLY

Services that also demonstrate we will go where others will not.

It's not just our design and manufacturing capacity that makes us different; our technical, engineering, procurement and customer services give us a competitive advantage.

- **Engineering and Design Support**—Applications and design engineering staff with decades of experience in flexible circuitry.
- **Domestic and International Production Capabilities**—Three manufacturing facilities in Asia and USA.
- **Value Added Assembly**—Reduce your vendor count, production delays and quality issues by having us do your sourcing, assembly and testing. From a single component to complex box build, we can handle your needs.
- **Inventory Stocking**—Pull and push inventory to meet your needs. Order in high volume but let us manage your inventories with JIT deliveries.
- **Prototypes, High or Low Volume**—Many manufacturers have minimum custom orders. We don't. Order 10 or a million.

HIGH DENSITY INTERCONNECT (HDI)

As electronic devices continue to shrink, PCB real estate in these devices becomes more densely populated. In many designs, there is just not enough room for all of the required SMT components, and also for all of the through hole interconnects between layers. In many cases, the answer to this problem is HDI (High Density Interconnect) technology.

What is HDI?

HDI combines several (or all) of these features:

- Very small traces and spacing (typically < 0.08mm/0.08mm).
- Very small via pads (typically <0.4mm).
- Very small interconnect vias (typically <0.15mm).
- Blind, buried and/or filled vias on one or more layers.

HDI technology allows the designer to eliminate many of the usual through hole vias that are used to connect layers, and move those interconnects to internal layers of the circuit. This will free valuable space on the outer layers that can now be used for SMT components.

In order to manufacture ultra high wiring density flex circuits with features this small, state of the art equipment and processes are required. Sequential lamination processes combined with laser direct imaging technology (LDI) is required to overcome the inherent dimensional instability of the flex circuit materials. Mechanical and Laser drills with optical targeting, and high aspect ratio plating lines are necessary to ensure well placed and reliably plated vias. AOI (Automatic Optical Inspection) is also required to accurately identify internal and external etching flaws. All of this state of the art processing equipment makes FCT uniquely qualified to tackle even the most demanding high density flex designs.

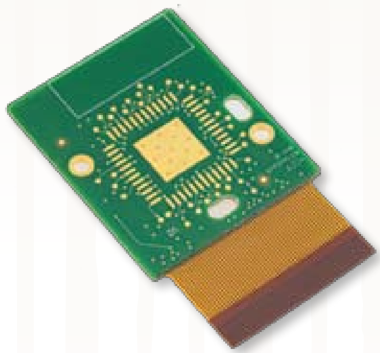
FCT feature sizes for HDI designs:

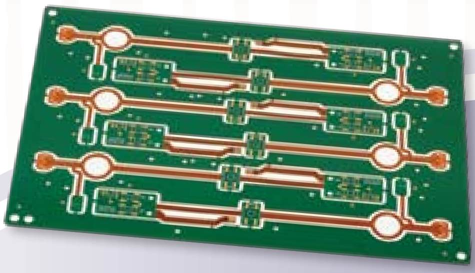
- Minimum trace: 0.05mm
- Minimum space: 0.05mm
- Minimum pad size: via size plus 0.15mm
- Minimum through hole drill 0.1mm
- Minimum laser drill 0.08mm

RIGID FLEX DESIGN GUIDELINES

Since rigid flex circuits are a hybrid of rigid and flexible PCBs, there are special guidelines that apply to this type of construction.

- On rigid flex circuits, ensure that all plated through holes are in a rigid area (no PTHs in flex areas).
- Specify adhesiveless flex materials and "cut-back" or "bikini" cover construction for rigid flex designs. Acrylic adhesive is the "Achilles heel" of a plated through hole in a rigid flex circuit. Eliminating acrylic adhesive from the plated through hole area will greatly increase the reliability of the PTHs.
- Rigid sections connected by flex should be a minimum of .375" apart and preferably .5" or more.
- Utilize "unbonded" construction to increase flexibility (see illustration on page 16). When using unbonded construction on impedance controlled circuits, you must ensure that signal and reference plane layers are not unbonded from each other. When the circuit is bent, the unbonded areas will buckle, which will cause an impedance mismatch if the signal and reference plane layers are not bonded together.
- When specifying a carrier panel or "pallet" for component installation, contact your manufacturer to make sure that the carrier panel fits efficiently on their processing panel. Failure to do this can result in a major cost increase.





REQUIREMENTS FOR FLEX QUOTE / FABRICATION

Budgetary Quote:

Flexible Circuit Technologies can give you a budgetary quote with a minimal amount of information. We would need the approximate layer count, part size and shape, circuit type (i.e. type 1-4), and how many circuits you want. If you have additional information at this stage of your design, include that as well. Keep in mind that FCT is eager to assist you at this stage of your flex circuit design. Many potential design and performance problems can be avoided by including your flex manufacturer during the design stage.

Firm Quote:

A higher level of documentation is required for a firm quote. We will need your desired order quantity, plus a drawing showing part size and shape, materials used, drilled hole sizes and locations (unless shown in accompanying Gerber files), and notes that specify all critical features of your circuit. If you have CAD files at this stage of your design, please include those as well to ensure that you receive the best possible price for your circuit. At this point in your design, you should have pretty good idea of how the circuit will be formed, and any environmental concerns such as shock, vibration, or elevated temperatures. Sharing this information with your FCT Applications Engineer will allow him to evaluate the circuit construction and final configuration to determine if your design is sound.

To Fabricate Your Circuit:

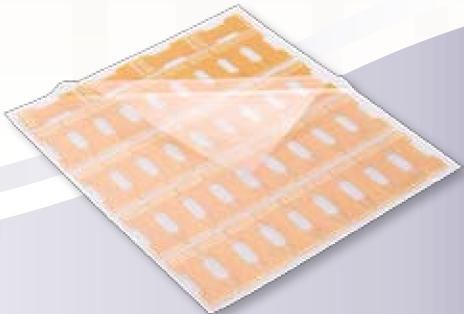
In order to fabricate your circuit, FCT will need to have a drawing, and CAD data that define all features of the circuit including all conductor layers, border outline, drill layer, conductor access openings, SMOBC (if required), and marking. This can be supplied in several formats, but the most common (and preferred) would be Gerber or ODB++. Drawings are usually transmitted in DWG or DXF format.

SHIPPING OPTIONS

There are many shipping options available to ensure that your circuits arrive at your facility in perfect condition. Many of these options can also be used as protective carriers on your production floor to reduce or eliminate damage due to handling.

Bulk Bag—This is the least expensive method and is best for bare, unformed circuits with no stiffeners or polyimide stiffeners.

Low Tack—Low cost and works well for all bare, unformed circuits including circuits with FR4 stiffeners. Available in ESD safe material.



Custom Shipping Trays—Moderate cost and offers the best protection. Best option for circuits that are formed or populated.

FLEXIBLE HEATERS

Flexible circuits can be made with resistive metals rather than copper, resulting in flexible heaters. Flexible heaters offer a low mass, ultra-thin heating solution that provides uniform heating with fast warm up. FCT can apply the heater to your device, or supply the heater with a pressure sensitive adhesive backing to be installed at your facility.

Specifying your heater:

Metal Foil

Metal foil type and thickness are driven by the overall resistance requirements of the heater and by the total area over which the resistance must be spread. The most common metals used for flexible heaters are:

- **Cupro-Nickel 715**—This alloy is 70% copper and 30% nickel and has a very flat TCR (Temperature Coefficient of Resistance). This alloy is typically used in applications that don't require a high resistance density. It is possible to solder and copper plate to Cupro-Nickel.
- **Constantan**—Constantan is a variation of Cupro-Nickel with 55 percent copper and 45 percent nickel. Constantan is also typically used in flex circuit applications such as strain gauges and thermocouples. Constantan also has a very low TCR.
- **Inconel**—There are several alloys of Inconel, but all are predominantly nickel, with chromium as a second element. Iron, Molybdenum, Niobium, Cobalt and other metals are used to create the different Inconel alloys. Inconel 600 is the most commonly used Inconel alloy for flexible heaters. The high resistivity makes this foil ideal for applications that require a high resistance packed into in a small area.
- **Aluminum**—Aluminum foil is generally chosen as a heater element material in order to save money. The resistivity is roughly double that of copper, and like many other pure metals it has a high TCR. It may be necessary to have control circuitry that can adjust to the changing resistance of the heater. Aluminum etches very quickly which makes it difficult for the manufacturer to keep tight resistance control.

Flexible Heaters are thin bendable elements that can be shaped to fit your unique heating equipment needs

Metal Type	Resistivity	Low TCR
Copper (reference)	.661417 uohm-inch	
Cupro-Nickel	16.22047 uohm-inch	X
Constantan	19.63495 uohm inch	X
Inconel	40.6 uohm-inch	X
Aluminum	1.10236 uomh-inch	

*Other metal foil types also available. Please contact FCT applications engineer for assistance in selecting the appropriate resistive foil for your application.

Insulation Type

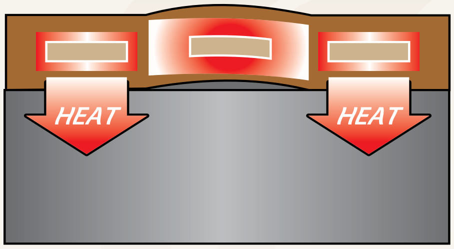
Insulation choices are driven mainly by the temperature requirements of the heater. The most common insulations are:

- **Polyimide Film With Acrylic Adhesive**—The most common flexible heater insulation. It has a wide temperature range and extremely good dielectric properties (.001" polyimide has a dielectric strength rating of 7700 volts). The high dielectric strength of polyimide film allows the use of film thicknesses as low as .001". This results in extremely fast response time and quick transfer of heat from the heating element to the object being heated (usually referred to as the heat sink). This material has excellent adhesion and moderate chemical resistance.
- **Polyimide Film With Epoxy Adhesive**—Very similar in properties to polyimide film with acrylic adhesive, but with better chemical resistance.
- **Polyester Film**—Lower cost than polyimide film resulting in lower overall heater cost.

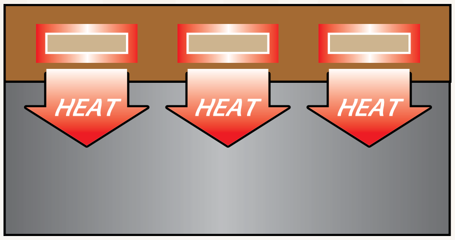
Heat Sink Adhesives

There are several adhesives that can be used to bond flexible heaters to a heat sink. FCT can use thermo-setting acrylic or epoxy film adhesives to bond the heaters for you and supply a turn-key assembly. Flexible heaters can also be supplied with a wide range of pressure sensitive adhesives with release paper backing so that you can install your heater yourself. It is very important to mount the heater such that there are no air pockets between the heater and the heat sink. Air pockets can reduce heat transfer and create hot spots.

Air pockets between the heater and heat sink will cause localized hot spots which can result in premature heater failure.



Proper installation provides good heat transfer to the heat sink.



Insulation Chart

	Temperature Rating	Chemical Resistance
Polyimide/Acrylic	-200 C to 150 C	Fair
Polyimide/Epoxy	-200 C to 150 C	Good
Polyester/Acrylic	-40 C to 105 C	Fair

FLEXIBLE CIRCUIT TECHNOLOGIES | www.FlexibleCircuit.com

U.S. OPERATIONS | 9850 51st Ave. N. | Plymouth, MN 55442, USA
Sales@FlexibleCircuit.com | Toll-free: 888-921-6167 | Phone: 763-545-3333 | Fax: 763-545-4444

SHENZHEN OPERATIONS | F3 Building 6, Liantang Industrial Area | Dan Zhutou,
Long Gang District Shenzhen, CHINA 518004

XIAMEN OPERATIONS | Xiamen Haicang Zone Cheung Road 198 | Xiamen, CHINA 361006