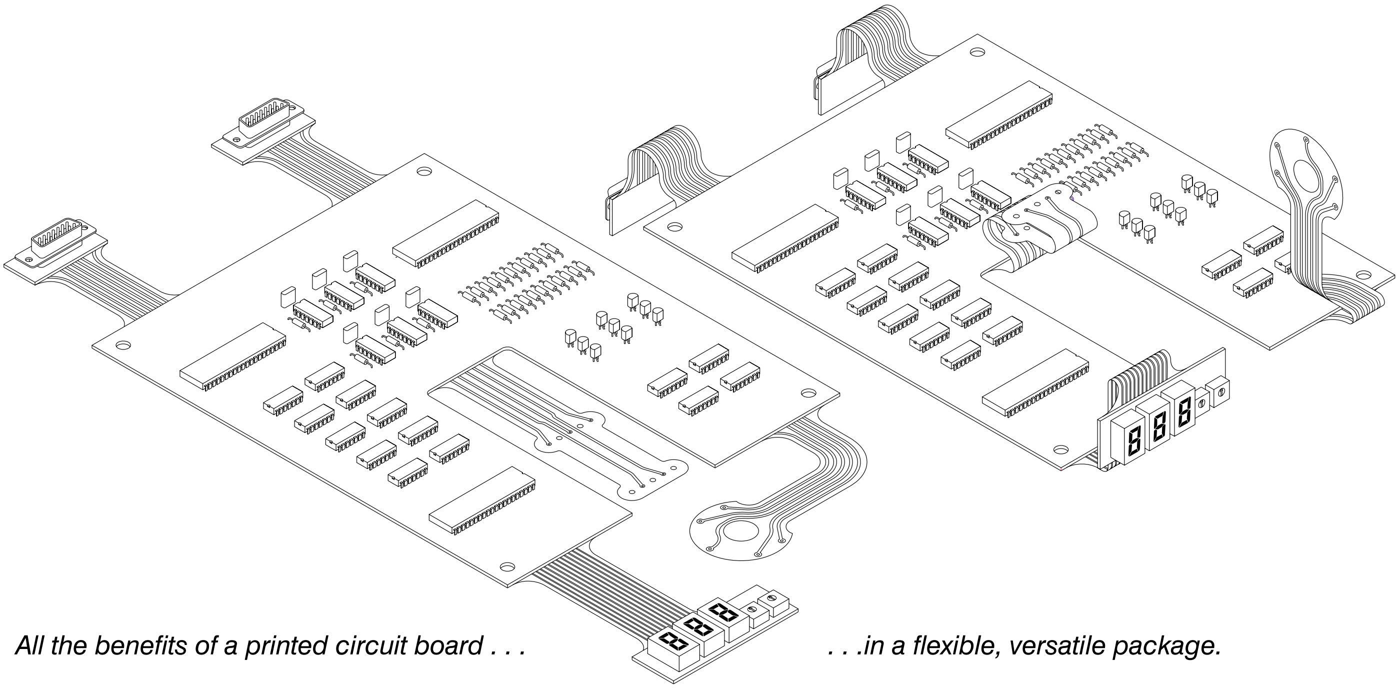
APPLICATION AID #24

Flex-Circuit Design Guide



All the benefits of a printed circuit board . . . *. . .in a flexible, versatile package.*

The purpose of this design guide is to enable you to design a high-quality, manufacturable flexible printed circuit. While using this guide, keep in mind that the design information provided is only a suggestion. Minco takes pride in manufacturing flex-circuits that are considered difficult to build. In most cases we can and do build above and beyond the 'standard' circuit, provided that the circuit design and type allow for it.

Minco is certified to MIL-P-50884, Type 1-5. If your circuit must meet MIL-P-50884 or IPC-6013, we urge you to read IPC-2223 and follow its recommendations.

You are encouraged to contact Minco with your questions and concerns.

Contents

Flex-circuits are a reliable alternative to conventional wiring. Not only do they improve connection reliability, they simplify assembly and improve component appearance. Flex-circuits only fit one way for fewer wiring errors during installation and servicing. This reduces rework and trouble-shooting time. By eliminating bulky wires, flex-circuits provide a cleaner, neater appearance.

Flex-circuits offer the same advantages of a printed circuit board: repeatability, reliability, and high density. However, the most important attribute that has made designers adopt flex-circuit technology is the capability of the flex-circuit to assume three-dimensional configurations. Flex-circuits can flex during installation and maintenance, or in use. With a little experimentation and imagination, a flex-circuit can save up to 75% of the space and/or weight of conventional wiring.

Furthermore, a flex-circuit offers dependability of price and of product. Recurring costs are lower than many wire harnesses, and since a flex-circuit is more resistant to shock and vibration than a PCB, repair and replacement costs are less.

Below are some of the ways that a flex-circuit can solve your design/packaging problems.

- Replace hardboard/ribbon cable assemblies.
- Replace multiple hardboards and connectors with a single flex-circuit or rigid-flex circuit.
- Control impedance with integral ground planes.
- Control EMI with solid or patterned shield layers.
- Surface mount components, then fold into tight areas.
- Feed electrical connections through gaps too narrow for round wires.
- Design circuit with exposed conductors for use with conductive elastomer keypads.
- Integrate heaters, temperature sensors, or wire-wound antenna coils into a flex-circuit (this is an exclusive offering by Minco).
- Specify pressure-sensitive conductive adhesive to attach circuit to cabinets or enclosures.
- Use flex-circuits as jumpers between hardboards.
- Replace tiny wires with easy-to-handle circuits.
- Buy circuits in panels for automated assembly – circuits can then be routed and clipped out singly.
- Use low thermally conductive flex materials and foils for electronic interface to super-cooled enclosures, or other environments with large internal/external temperature differences.

Specific applications

Military/Aerospace Field

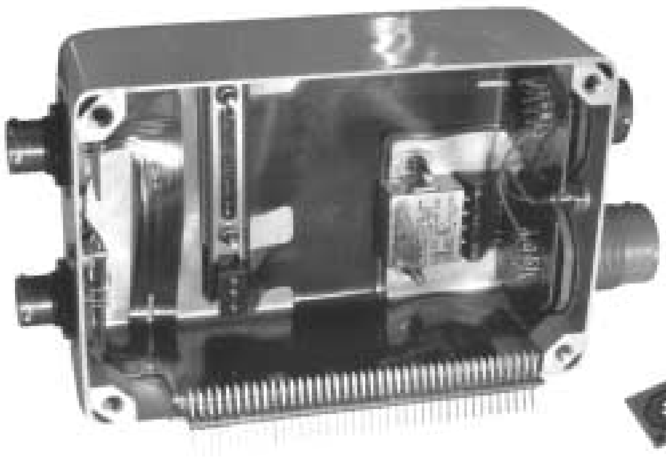
- Accelerometers
- Aircraft engine fuel controls
- Aircraft instrument plasma display connection
- Anti-tank missiles
- Avionics packages (altitude gauges, engine, fuel, etc.)
- Computer memory plane
- High-speed interconnects
- Dynamically tuned gyroscope
- Electronic countermeasure instrumentation
- Forward-looking infrared
- Laser gyroscope
- Parachute harness release
- Passenger address system
- Radar altimeters
- Satellites
- Space shuttle motor controls
- Torpedoes

Commercial Applications

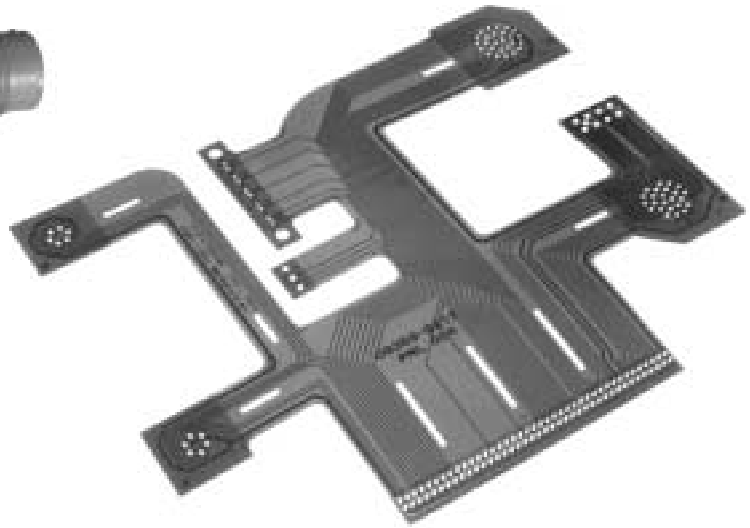
- Computer printheads
- Copy machine controls
- Disk drive head connection
- Exercise monitoring equipment
- Fiber optic switching modules
- High-speed computer printheads
- Injection molding equipment controls
- Laser communications
- Pick-up coils for drill pipe inspection
- Portable radios

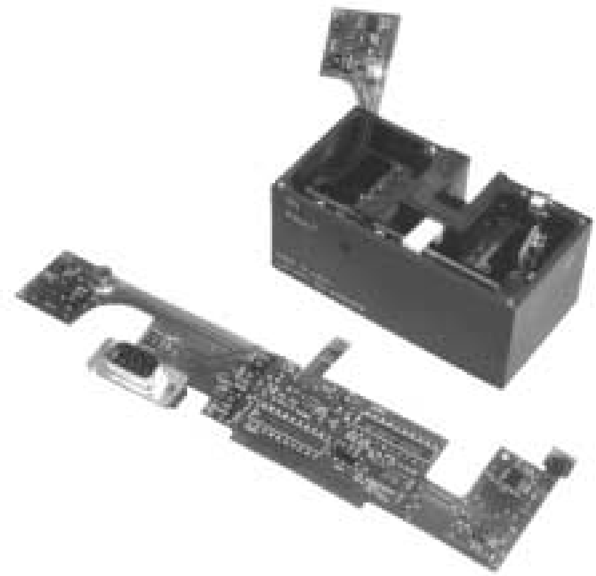
Medical Field

- Battery connections
- Blood analyzers
- Defibrillators (implantable and external)
- Hearing aids (in-the-ear and implantable)
- Implantable drug pumps
- Inductive bone growth stimulators
- Medical diagnostic equipment
- Nerve stimulators
- Nuclear magnetic resonance machines
- Pacemakers
- Pacemaker programming controls
- Shielding
- Ultrasonic probes



Aerospace field –
The flex-circuit below replaces round wiring in an avionics package (left). The pins at the front of the flex-circuit connect to the mother board.





Commercial application –
A flex-circuit with surface mount components folds to fit into a laser dimensioning device.

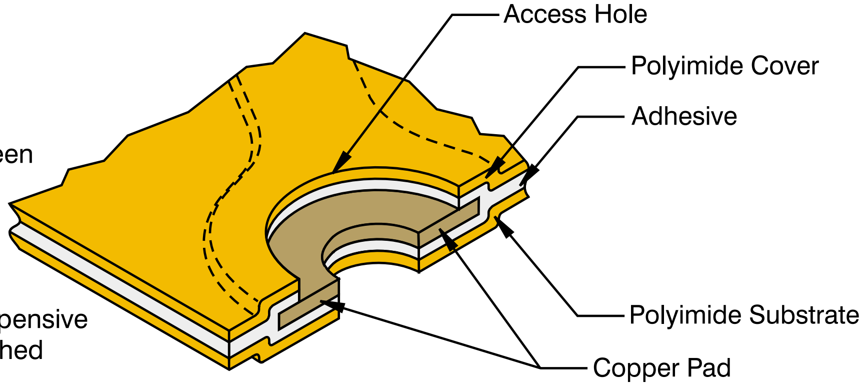
Medical field –
A double-layer flex-circuit and dual Flex-Coil provide interconnection and communication capabilities in an implantable cardiac pacemaker.

Circuit types

Minco manufactures all types of circuits, from simple to complex. The basic types are:

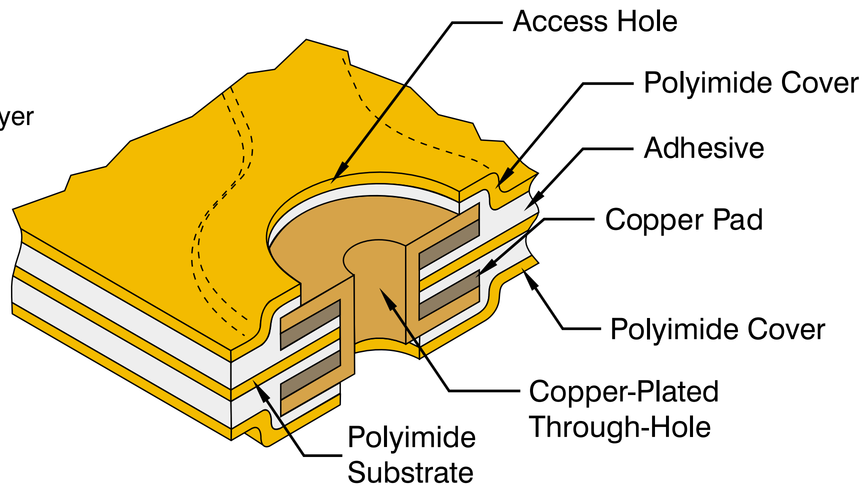
Single-layer

- Mil-P-50884 – Type 1
- One conductive layer, either bonded between two insulating layers or uncovered on one side.
- Access holes to conductors may be on either one or both sides. Access holes on both sides of a single-layer are more expensive since the substrate must be drilled or punched separately.
- Stiffeners, pins, connectors, components are optional.



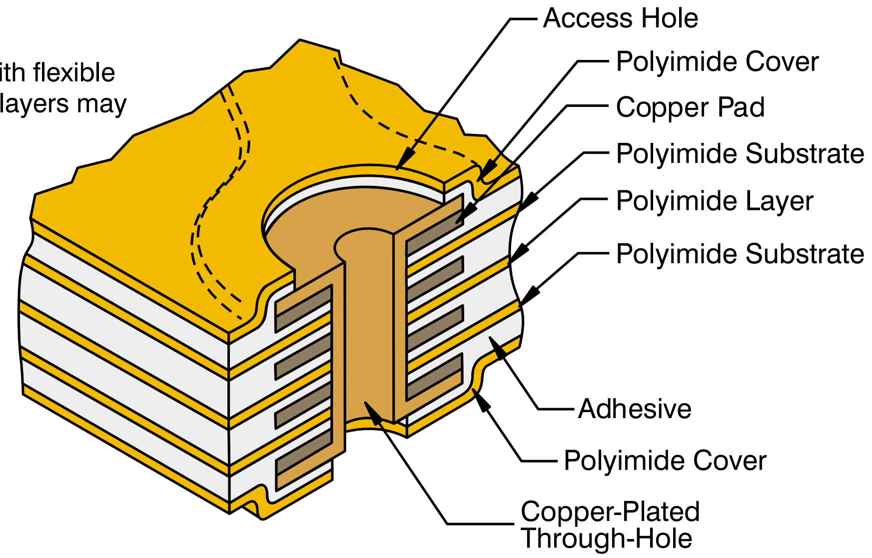
Double-layer

- Mil-P-50884 – Type 2
- Two conductive layers with an insulating layer between, plus cover layers on outer layers.
- Plated through-holes provide connection.
- Access holes or exposed pads without covers may be on either or both sides; vias can also be covered on both sides.
- Stiffeners, pins, connectors, components are optional.



Multilayer

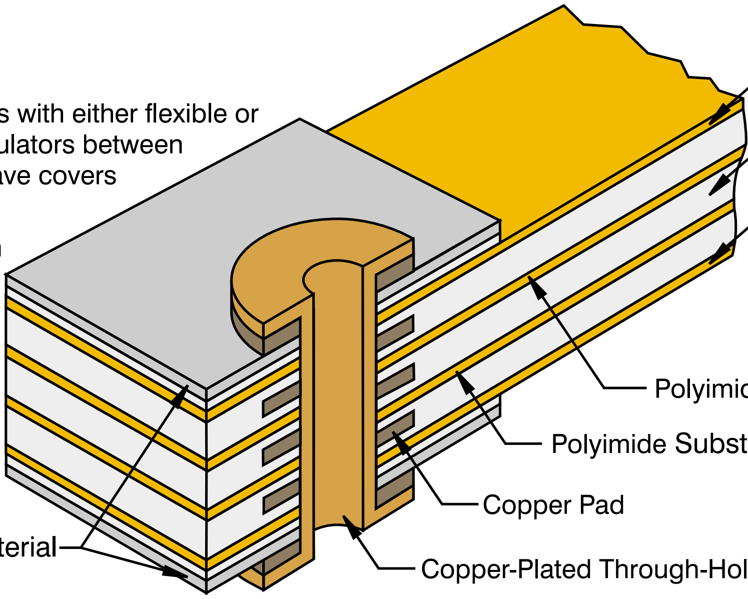
- Mil-P-50884 – Type 3
- Three or more flexible conductive layers with flexible insulating layers between each one; outer layers may have covers or exposed pads.
- Plated through-holes for connection.
- Access holes or exposed pads without covers may be on either or both sides.
- Blind or buried vias are possible.
- Stiffeners, pins, connectors, components are optional.



*Adhesiveless base material also available

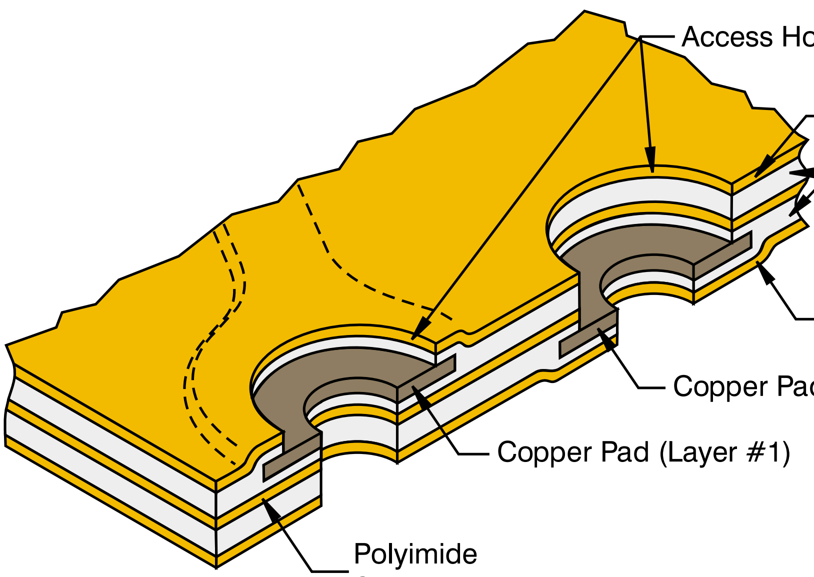
Rigid-Flex

- Mil-P-50884 – Type 4
- Two or more conductive layers with either flexible or rigid insulating material as insulators between each one; outer layers may have covers or exposed pads.
- Rigid-flex is differentiated from multilayer circuits with stiffeners by having conductors on the rigid layers. Plated through-holes extend through both rigid and flexible layers (with the exception of blind and buried vias). Rigid-flex also costs more.



- Access holes or exposed pads without covers may be on either or both sides; vias or interconnects can be fully covered for maximum insulation.
- Stiffeners, pins, connectors, components, heat sinks, and mounting brackets are optional.
- We also manufacture 'flush' rigid-flex, where the top surface of contact areas is level with adjacent adhesive/insulation.
- Minco is capable of sequentially laminated, drilled, and plated circuits, which allows for more flexibility in designing the circuit.

Note: The Special Considerations for Rigid-Flex section on page 20 provides additional information for designing a rigid-flex circuit.



Multilayer, no plated through-holes

- MIL-P-50884 – Type 5
- Two or more conductive layers with insulating layers between each one; outer layers may have covers or exposed pads.
- Through-holes are not plated.
- Access holes or exposed pads without covers may be on either or both sides.
- Stiffeners, pins, and connectors are optional.

*Adhesiveless base material also available

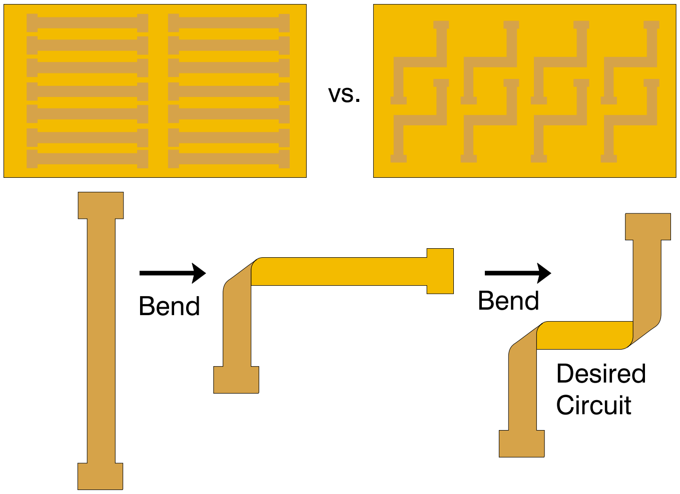
Design differences – flex-circuit vs. hardboards

Designing a flex-circuit is only one step away from designing a hardboard. The most important design difference to keep in mind is the three-dimensionality of a flex-circuit. Creative bending and flexing can save space and layers. Other important differences:

- Flex-circuits both require and permit looser tolerances than hardboards.
- Because arms can flex, design them slightly longer than required.

Design Tips to Minimize Circuit Cost

- Always consider how circuits will be 'nested' on a panel.



Efficient nesting = Yield of 14
vs.
Inefficient nesting = Yield of 8

- Keep circuits small; consider using a set of smaller circuits instead of one large circuit.
- Follow recommended tolerances whenever possible.
- Design unbonded areas only where they are necessary.
- If circuits have only a few layers, stiffeners can be far less expensive than designing a rigid-flex circuit.
- Specify 0.001" of adhesive on the cover material per 1 oz. of copper (including plated copper).
- Building circuits with exposed pads and no cover layers is sometimes less expensive.

Step-by-step approach to designing a flex-circuit

1. Read the available literature that is applicable for the circuit you desire. If the circuit is intended for a military/aerospace application, review IPC-6013 and IPC-2223 or MIL-P-50884. Copies are available from Minco.
   Applicable sections in this design guide:
   - Manufacturing of a Flex-Circuit
   - Reference Documents
2. Define the circuit parameters according to the package that uses the circuit. It may be helpful to cut out a paper template to represent the actual circuit. Experiment with bending and forming the template in order to achieve maximum efficiency. Design a circuit for maximum 'nesting' in order to fit as many circuits as possible on a panel.
   Applicable sections in this design guide:
   - Design Differences: Flex-Circuit vs. Hardboard
   - How to Improve Flexibility and Bend Radius
3. Determine the wiring locations and the conductor paths. This step will determine the number of conductor layers.
   Applicable sections in this design guide:
   - Circuit Types
   - Minco Circuit Codes
   - Cost Impact of Layer Count
   - Special Considerations for Rigid-Flex
4. Calculate the conductor width and spacing according to the current capacity and voltage.
   Applicable sections in this design guide:
   - Tolerances
   - Nomograph
   - Electrical
5. Decide what materials to use.
   Applicable sections in this design guide:
   - Materials
6. Choose the methods of termination and through-hole size(s). Evaluate the bend areas and methods of termination to determine if stiffeners are needed.
   Applicable sections in this design guide:
   - Terminations
   - Holes
7. Determine what testing you require. Avoid over-specification in order to avoid additional cost.
   Applicable sections in this design guide:
   - Minco Test Capabilities

Minco circuit codes

Circuit Code	MIL-P-50884 Type	Description
E0C	-	Stiffeners, insulators, wires only. No circuitry.
E1A	1	Single layer - access on one side.
E1B	1	Single layer - major access holes on two sides.
E1C	1	Single layer - access on one side, with stiffeners, pins, or connectors.
E1D	1	Single layer - access on both sides, with stiffeners, pins, or connectors.
E2A	5	Two layers - no plated through-holes.
E2B	2	Two layers - with plated through-holes.
E2C	2	Two layers - with plated through-holes and stiffeners, pins, or connectors.
E2D	5	Two layers - no plated through-holes, with stiffeners, pins, or connectors.
EM*x*A*	5	Multilayer flex-circuit -no plated through-holes.
EM*x*B	3	Multilayer flex-circuit -with plated through holes.
EM*x*C	3	Multilayer flex-circuit -with plated through-holes and stiffeners, pins, or connectors.
EM*x*D	5	Multilayer flex-circuit - no plated through-holes, with stiffeners, pins, or connectors.
ER*x*A	5	Rigid-flex - no plated through-holes.
ER*x*B	4	Rigid-flex - with plated through-holes.
ER*x*C	4	Rigid-flex - with plated through-holes and stiffeners, pins, and connectors.
ER*x*D	5	Rigid-flex - no plated through-holes, with stiffeners, pins, or connectors.

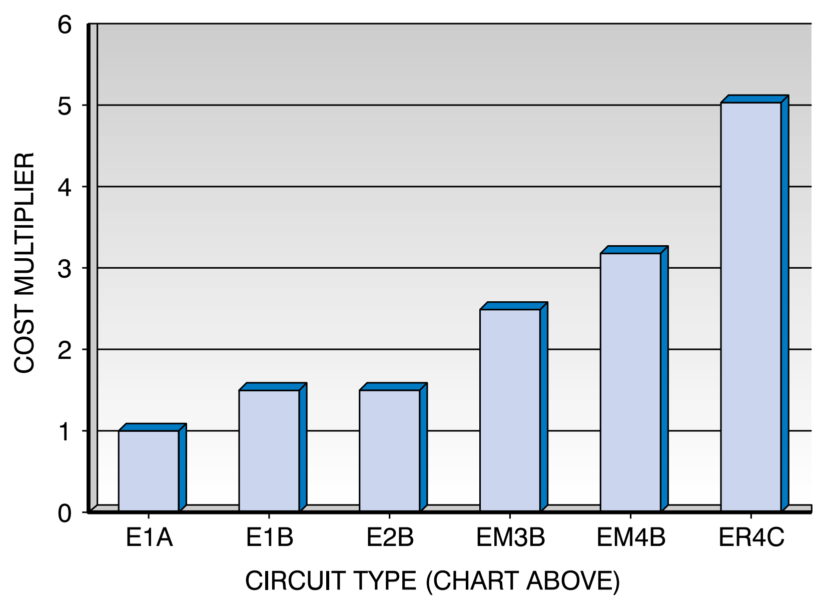
* Layer count indicated by *x*. Example: EM3B1887, ER6C2896

Note: Coils are represented with a 'C' instead of an 'E'.

Cost impact of layer count

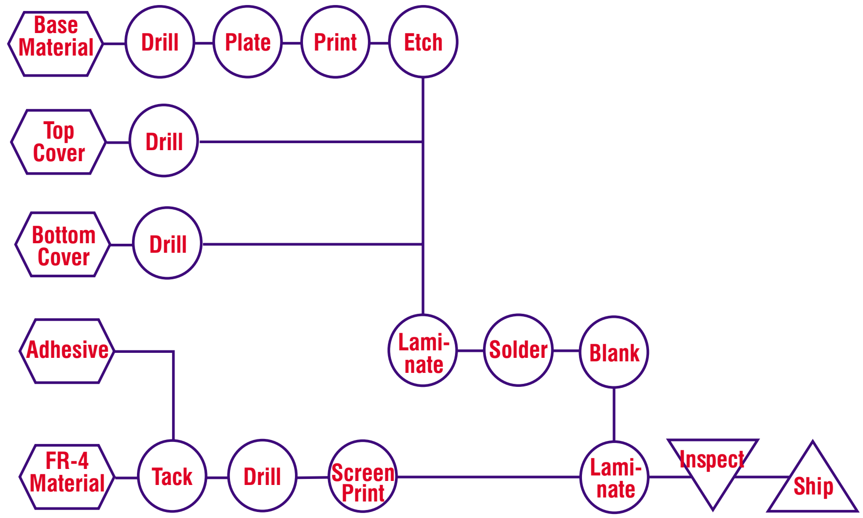
The information for the chart to the right was taken from a sampling of circuits built with Minco's standard materials. This chart is not intended to be used as a price guide. However, it does show that circuit cost generally rises with layer count.

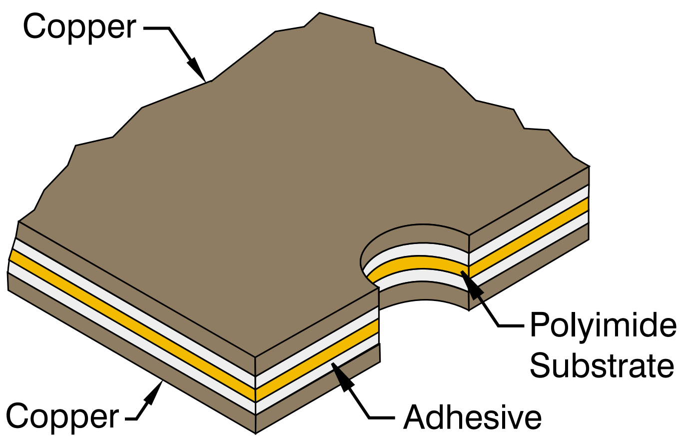
It is in your best interest to experiment with options in order to save money. For example, consider using two circuits to do the job of one. Two double-layer circuits could potentially be less expensive than one four-layer multilayer circuit. Circuits can also be folded in order to save space and layers.



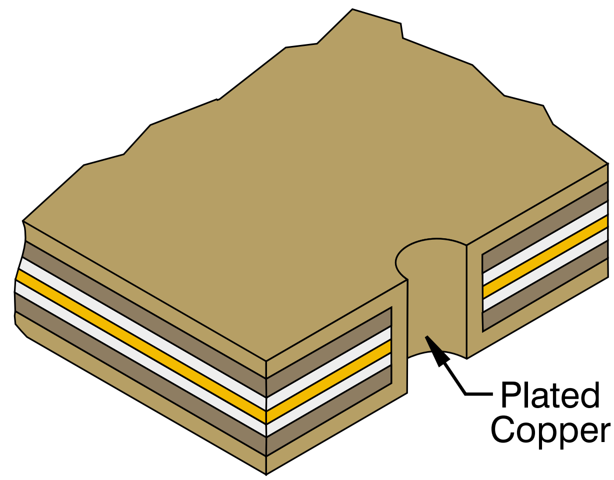
Manufacturing a flex-circuit

Building a flex-circuit generally involves the same steps from circuit to circuit. However, certain circuit designs can add cost. For example, a single-layer circuit with access holes on both sides is more expensive than a single-layer with access holes on one side because the double-sided access hole circuit must have its substrate drilled separately. The adjacent flow chart and the illustrations below identify some cost driver issues, such as access holes, plated through-holes, etc. The flow chart shows the manufacturing process for a standard double-layer circuit with a stiffener.

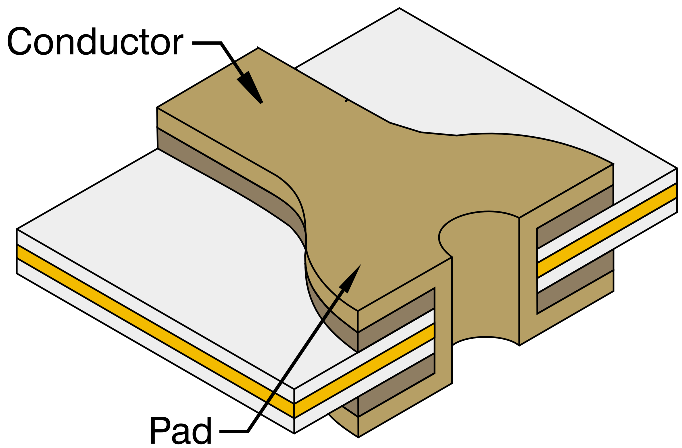




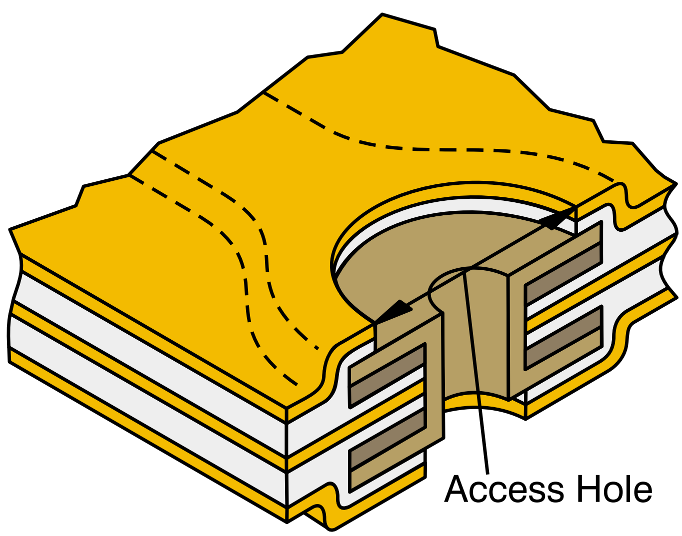
1. Double-sided material is drilled



2. Through-holes are copper-plated



3. Copper is etched to create conductors and pads



4. Polyimide covers are laminated over etched copper

Materials

This table lists the standard materials and material thicknesses that Minco has available. Minco's standard materials are in boldface. If the material or thickness you require is not listed, consult Minco.

Material Function	Material Type	Sizes/Thicknesses Available
Flexible Insulator	**Kapton*** and other polyimides †	0.0005", 0.001", 0.002", 0.003", 0.005"
Rigid Substrate (Rigid-Flex)	**FR-4**	Variety of thicknesses between 0.003" and 0.125"
	Polyimide Glass	Variety of thicknesses between 0.003" and 0.125"
Conductor	**Copper**	¼ oz., ⅓ oz., ½ oz., 1 oz., 2 oz., 3 oz., 5 oz., 7 oz., 10 oz.
	Different hardnesses of copper	Half-hard, **rolled-annealed**, electrodeposited
	Beryllium copper	0.003": half-hard and quarter-hard 0.004": half-hard
	Cupro-nickel (70/30 alloy)	0.000625", 0.0009", 0.0013", 0.0019", 0.0023"
	Nickel	0.002", 0.003", 0.005"
	Silver Epoxy	‡
Adhesive	**Modified acrylic**	0.0005", 0.001", 0.002", 0.003", 0.004"
	Modified epoxy	0.0005", 0.001", 0.002", 0.003", 0.004"
	Phenolic Butyral	0.001", 0.002"
	Pressure-sensitive adhesive (PSA)	0.001", 0.002", 0.005"
	Preimpregnated material: FR-4, polyimide	0.002" -0.008"
Stiffener	Copper, Aluminum, & other metals	Variety of thicknesses available.
	Polyimide Glass	See "Rigid Substrate - Polyimide Glass" above.
	FR-4	Variety of thicknesses between 0.005" and 0.125"
	Polyimide	See "Flexible Insulator -Kapton" above.

* Dielectric is 2,000 volts/0.001".

† Other polyimides are available for special applications.

‡ Material is applied as an alternative to standard copper layers.

Kapton is a registered trademark of DuPont for polyimide. Kapton/modified acrylic has a temperature rating of -65 to 150°C, although circuits will discolor after a long term exposure at 150°C. For special applications, Minco can use an adhesive that will withstand temperatures of 150°C continuous, and 200°C short-term.

Tolerances

You are not limited to the tolerances listed in this section. Tighter tolerances than those listed can be attained, but often at a higher cost. Accordingly, more relaxed tolerances will cost less. Even with relaxed tolerances, a flex-circuit will still have a uniformity that is impossible to achieve with conventional wiring. Flex-circuits have more 'give' than the typical hardboard, so it is not always necessary to specify tight tolerances.

Circuit Dimension in Inches†	Outline Dimensions (Profile Tolerance)			Hole-to-Border Dimensions			Cluster to Cluster§
	Hand Trim	SRD	Punch and Die	Hand Trim	SRD	Punch and Die	
1	±0.020	±0.015	±0.003	±0.015	±0.015	±0.007	±0.003
5	±0.025	±0.020	±0.007	±0.020	±0.020	±0.012	±0.007
10	±0.030	±0.025	±0.012	±0.025	±0.025	±0.017	±0.012
15	±0.035	±0.030	±0.017	±0.030	±0.030	±0.022	±0.017
20	±0.040	±0.035	±0.022	±0.035	±0.035	±0.027	±0.022

†Round circuit to next highest increment.

§Represents from a group of holes to a group of holes. Holes *within* a group will have a tolerance of ±0.003.

Note: Dimensional tolerances are given in inches. See Glossary for description of profile tolerance.

Trimming

Each trimming method has advantages and disadvantages. Generally, hand-trimming is only performed for 'best effort' or prototype circuits. Laser trimming provides hard tooling tolerances for small quantities of circuits. It is also useful for complex cutouts not achievable with other methods.

Steel rule dies are best for intermediate quantities and tolerances. Punch-and-dies are recommended for tight tolerances, complex circuits, and/or a high quantity of circuits. For more specific information on steel rule dies and punch-and-dies, see the Glossary at the end of this guide. Other tooling options are available to provide near punch and die tolerances for medium volume quantities.

Solder Thickness

MIL-P-50884 does not require a maximum solder thickness, but Minco imposes a 0.0003" minimum on pads, measured at the crest. A 0.0001" minimum is required on through-hole walls, also measured at the crest. Optional plated solder has a ±0.0005" tolerance before reflow.

Conductor Width and Spacing

See the nomograph on pages 12-13 for calculating proper conductor width and spacing. Minco can hold a 0.004" minimum conductor width/spacing (0.002" minimum at higher cost) on 1 oz. copper and 0.005" minimum conductor width/spacing on 2 oz. copper (for thicker copper, consult Minco). For best producibility, design circuit conductors at least five times wider than they are thick.

Tolerances for conductor width depend on whether the copper is plated or unplated:

Copper Thickness	Plated Copper	Unplated Copper
½ oz.	±0.004"	±0.001"
1, 2, 3 oz.	±0.004"	±0.003"

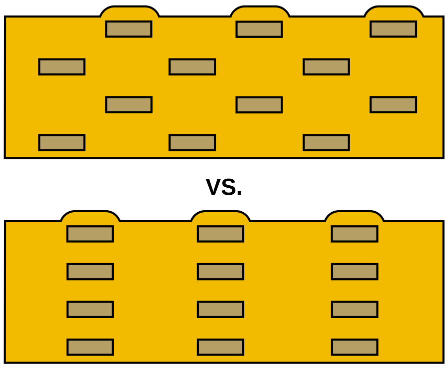
Maximum Circuit Size

- If all conductor widths are greater than 0.030": 24" × 72"
- Fine line circuits: 24" × 36"
- Circuits more than two layers: 16" × 22"

How to improve flexibility and bend radius

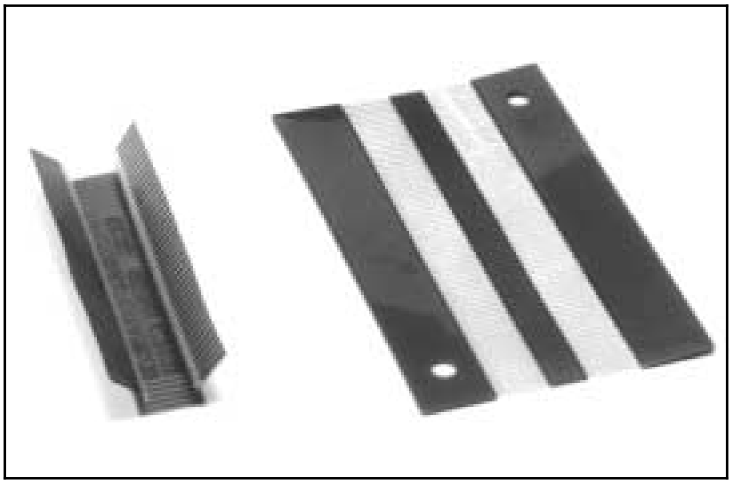
Single-layer circuits are the best choice for dynamic (flex-in-use) applications. Generally, double-layer or more circuits are best suited to static applications where the circuit flexes only during installation. The minimum allowable bend radius depends on a number of factors, and is best defined in IPC-2223. Approximate circuit thickness is slightly less than the sum of the insulator, adhesive, and foil layers. Some ideas to increase flexibility are:

- Circuits with two layers or more can be selectively plated to improve dynamic flexibility.
- It is best to keep the number of bends to a minimum.
- Stagger conductors to avoid to the I-beam effect, and route conductors perpendicular to a bend.
- Do not place pads or through-holes in bend areas.



Staggered conductors vs. I-beam effect

- Factory forming is an option you may want to consider. Most constructions can be factory formed, depending on geometry. Because circuits are flexible, formed circuits will relax in time. Form tolerances apply *only* to the part in the constrained position. Consult Minco for more information on factory forming.

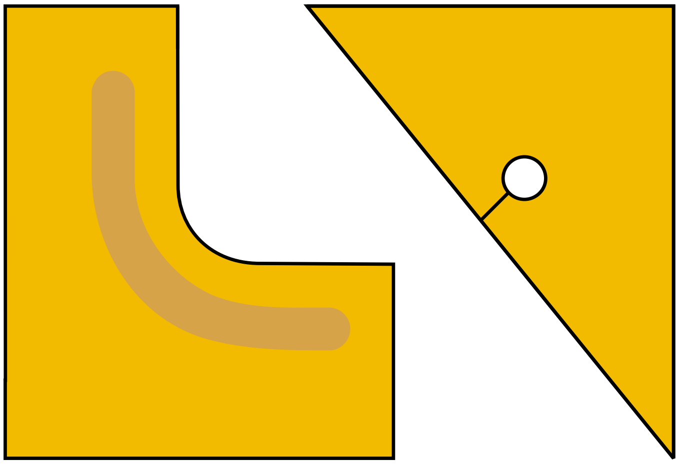


Factory formed circuit

- Do not place potting, discontinuities in the cover, discontinuities in the plating, or other stress concentrating features near any bend locations.
- Unbonded layers in a relatively thick multilayer or rigid-flex circuit are an option in order to improve flexibility, but this may be more expensive.
- Minco can provide an epoxy or RTV fillet to reinforce the edges between the flexible and rigid areas of a circuit.

Tear Stops

Polyimide has a high initial tear strength, but once a tear starts, it propagates easily. All inside corners must be radiused. The larger the inside radius, the greater the tear strength. If tearing is a concern, polyimide-insulated circuits can be designed with tear stops at the inner corners, for corners of 120° or less. Minco can incorporate internal or external polyimide or Teflon tear stops. Polyimide or Teflon tear stops will add to circuit cost.



Conductor width nomograph

The nomograph on the facing page will help you determine the maximum allowable current capacity, in amperes, of a flex-circuit conductor. Taken from MIL-STD-2118, the nomograph shows current for various conductor thicknesses, widths, and temperature rises.

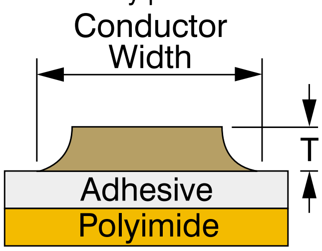
Using the Nomograph

1. Locate the width of the conductor at the left side of the bottom chart.
2. Go right to the line marked with the conductor thickness. Trace down to the bottom of the chart to find cross-sectional area of the conductor.
3. Go up to the curve in the top chart showing temperature rise. This is the difference between ambient temperature and the temperature of the current-carrying conductor. Conductor temperature should not exceed 105°C. For example, if the ambient temperature might reach 80°C, the temperature rise above ambient of the conductor should be less than 25°C (105°C - 80°C). In this case use the 20°C curve.
4. From the temperature rise curve, go left to find the maximum current.

Reverse the order of these steps to calculate required conductor width for a given current.

Conductor Aspect Ratio

For best producibility, design conductors to be at least five times as wide as they are thick. For example, with 2 oz. copper (0.0028") make conductors 0.0140" or wider. Unfortunately, with ever-increasing demands for circuit density, this is rarely possible.

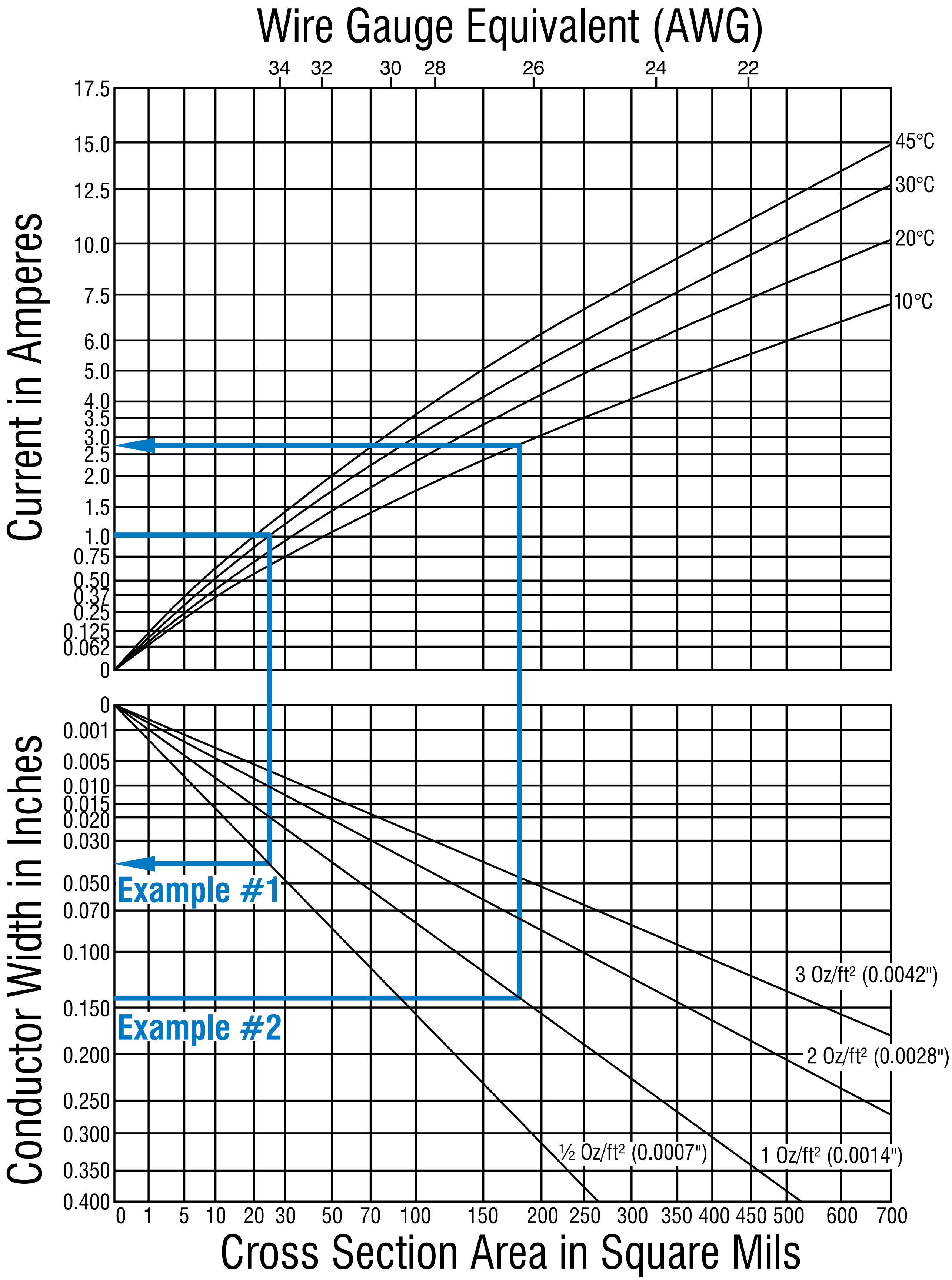


Conductor width should be 5x greater than the conductor thickness (T).

Assumptions

1. The nomograph is valid only for conductors with a polyimide cover layer, not exposed conductors.
2. The conductor thickness includes copper plating. Be aware that plating may add 0.0005" to 0.0014" of thickness. Selectively plated circuits do not have significant plating over conductors. The nomograph does not apply for plated metals other than copper.
3. Derate current by 15% for conductor thicknesses greater than 0.0042" (3 oz./sq. ft.).
4. The temperature rise curves only recognize heat generated by the conductor itself. Heat from power dissipating components or nearby conductors on other layers is not included.
5. It is assumed that conductor coverage is relatively small; i.e. there is enough free space between conductors for sideways heat dissipation. Groups of closely spaced parallel conductors *on the same layer* can be treated as one large conductor. Add all the cross sectional areas together and all the currents together to find temperature rise.
6. Current ratings are for still air environments. Forced air cooling will increase the rating; operating circuits in a vacuum will greatly decrease it.

Contact Minco for assistance in cases where the nomograph does not apply. Also contact us if you have difficulty designing sufficient current capacity into the space available. We can suggest ideas to increase current capacity.



Example #1: A current of 1 amp with ½ oz. copper and 30°C temperature rise will require a conductor width of 0.040".

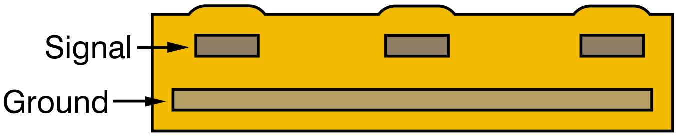
Example #2: A conductor with width 0.140", etched from 1 oz. copper (0.0014") will produce a temperature rise of 10°C at 2.7 amps.

Controlling Impedance and Electrical Noise

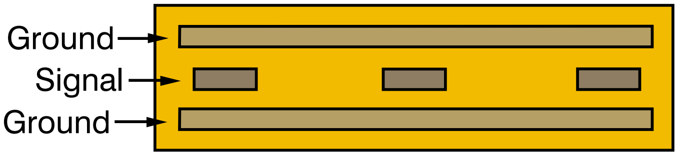
Predictable electrical characteristics make flex-circuits an ideal choice for high-speed signal transmission. Uniform spacing between conductors and grounds, continuous shield layers, and repeatable geometries are features that help to control impedance and reduce crosstalk. And with flex-circuits, you can eliminate connectors and other transitions that contribute to signal attenuation.

Flex-circuit design options to control impedance and EMI include:

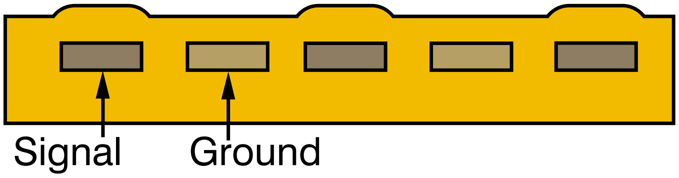
- Microstrip – a single ground plane beneath the signal lines.



- Stripline – dual ground layers above and below the signal lines.



- Alternating ground and signal lines in the same plane.



- Rigid-flex/stiffened flex-circuits with uninterrupted ground layers
- Silver epoxy coating. Silver epoxy is applied to the outside of circuits and electrically connected to other layers via access holes in the cover coat. Silver epoxy shielding can be less costly and is more flexible than copper.

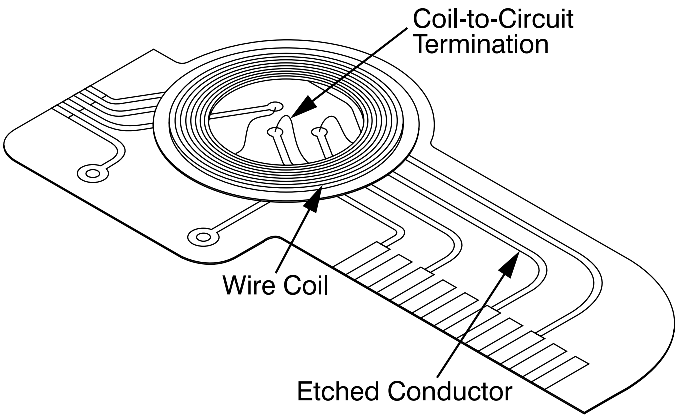
Minco can provide tight tolerance on line width, spacing, and distance to ground layers in order to meet your required impedance value. Actual impedance will also depend on the circuit's shape after installation.

Contact Minco for advice on designing circuits to specific electric characteristics.

Flex-Coils™

Flex-Coils are flex-circuits containing integral wire coils for use as antennas or inductors. There are three basic types of Flex-Coils:

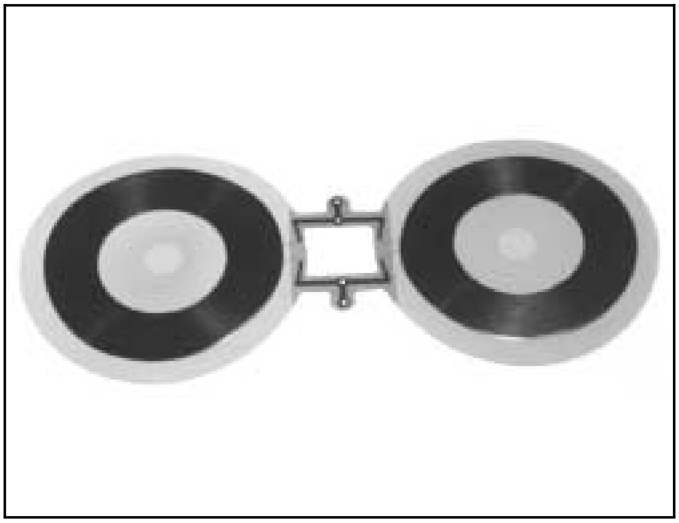
- Simple, flat coils with wire leads
- Coils laminated inside flex-circuits
- 'Rim' coils that are built up in the Z-axis.



Flex-Coil

Flex-Coils have the same advantages that a flex-circuit does. Wiring errors are reduced because the coil is oriented in the same spot every time, which provides repeatable signals. Flex-Coils are rugged and easy to assemble, and their design usually guarantees a reduced package size. A Flex-Coil can terminate in any manner that a flex-circuit can, or to a wire lead. Heavy wire leads are available.

See Bulletin FC-1, **Flex-Coils**, for more information on Flex-Coil capabilities, design considerations, and the information required for a quote or build.



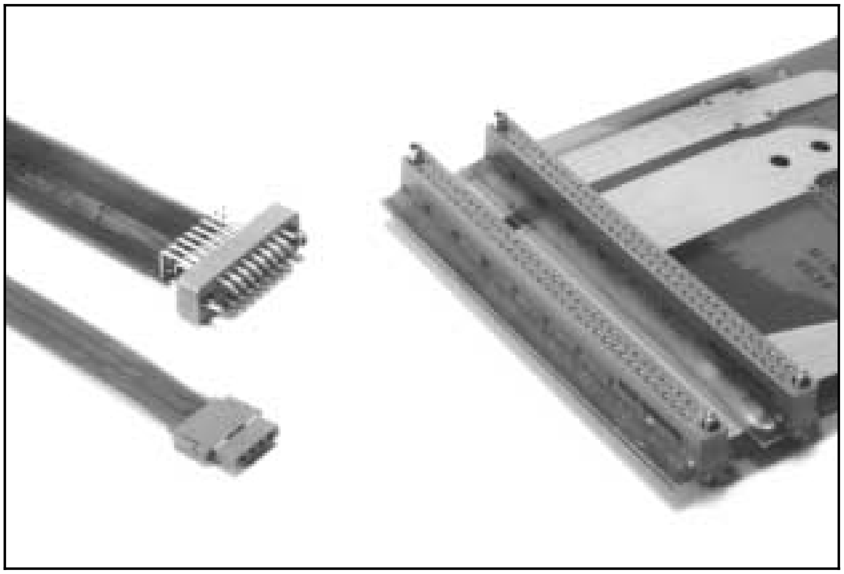
Minco Flex-Coil™

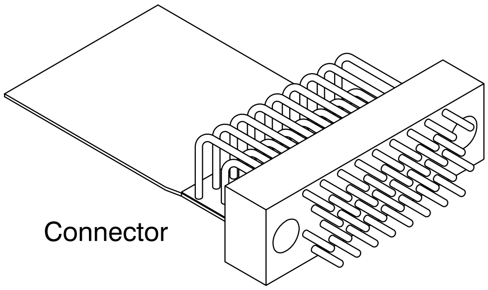
Terminations

There are a variety of terminations for a flex-circuit, and a variety of methods for applying these terminations.

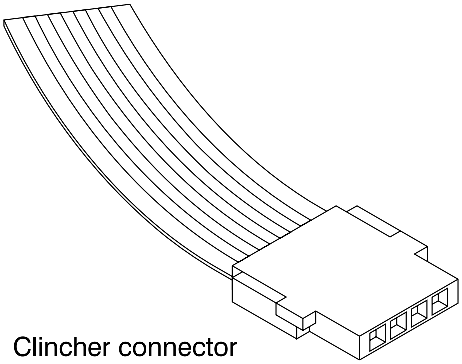
- Connectors are usually customer selected, and can be attached by hand-soldering, wave-soldering, or crimping in the case of insulation displacement connectors (IDC's). DuPont Clincher™ connectors are a good option for many applications. Connectors can be potted after attachment or conformally coated for protection and insulation with epoxy, polyurethane, or RTV.

Connector Type	Centerline Distance
Clincher Connector	0.100" min.
Other types: pin center-to-center	0.050" min.





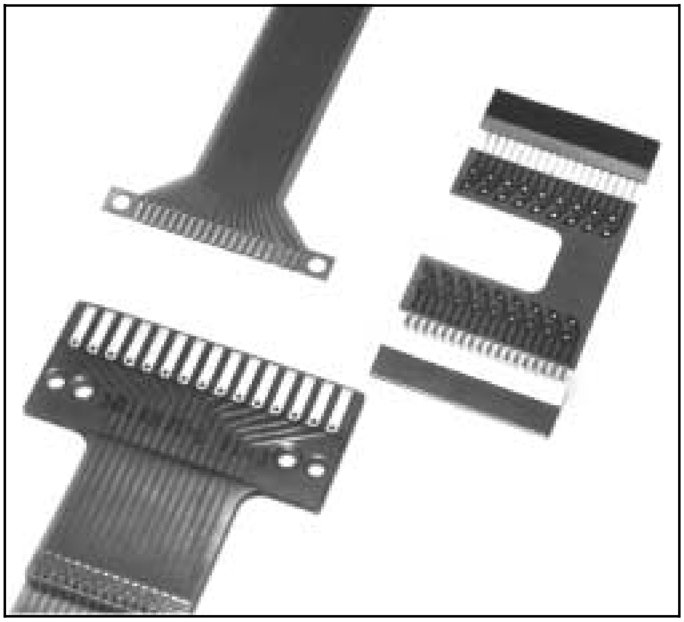
Connector

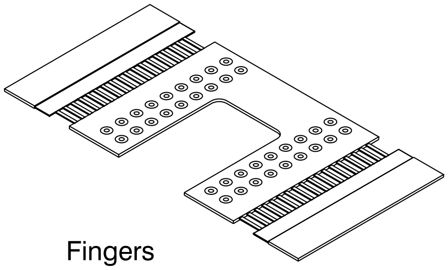


Clincher connector

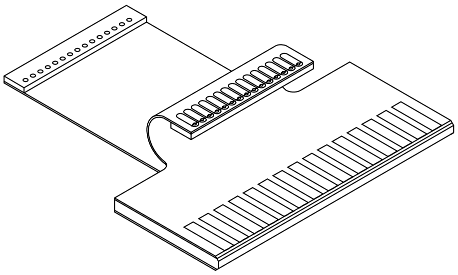
- Fingers can be supported or unsupported.

Finger Type	Centerline Distance
Supported	0.006" min.
Unsupported	0.020" min.



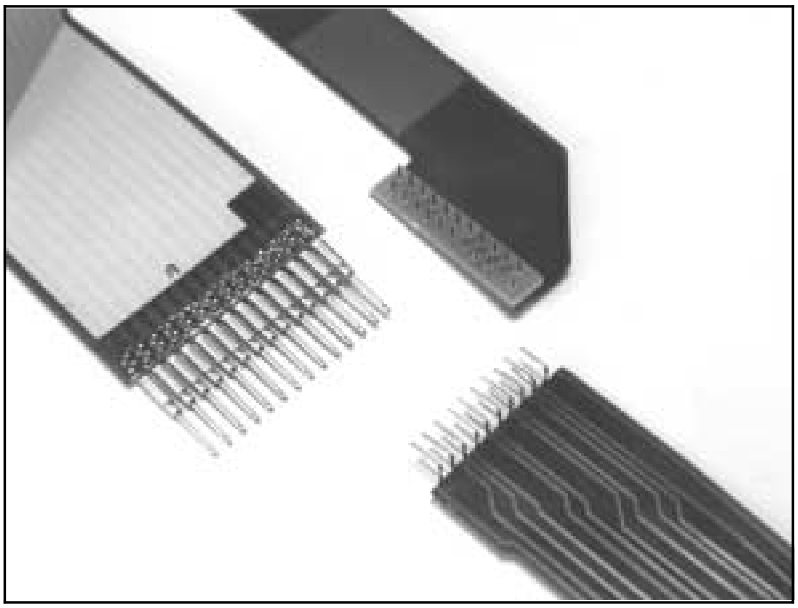


Fingers

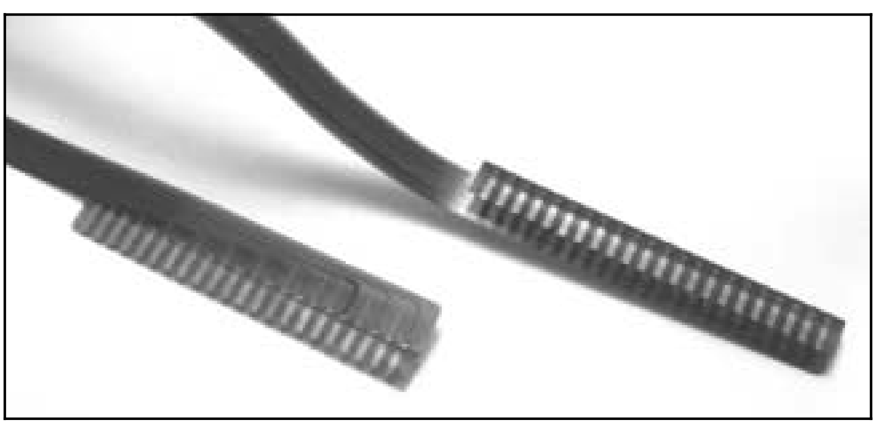


Pins

- Socket pins are pressed in place and then soldered. Soldered pins offer the lowest cost in pin terminals. Pins can be swaged to the circuit and soldered after the swaging procedure. Or, pins can be swaged to a FR-4 stiffener and then soldered. Swaged/soldered pins are moderately priced and have good mechanical strength.
- Brazed pins and ribbon, applied with a welding technique (melting point 618°C), are also available, and can provide a weldable surface for subsequent assembly.
- End pins that are in line with conductors can be brazed, soldered, or crimped to conductors. Pins can be bent to form a staggered arrangement.

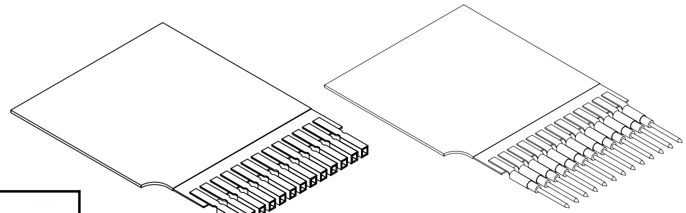


- Flex-circuits can interface to hardboards via soldered lap joints, lap joints applied with an anisotropic adhesive (conductive in the Z-axis only), compression dots using a raised metal dot, or ‘zebra’ strips. Compression methods usually require a stiffener behind the contact area.

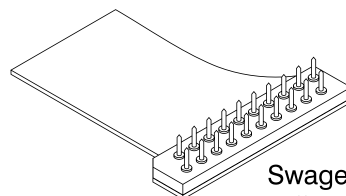


Pins can be inserted separately or ganged in a header. Minco recommends using a FR-4 or polyimide stiffener in pin areas to improve mechanical strength and simplify assembly.

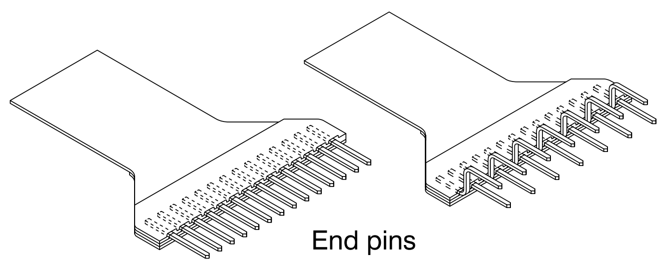
Pin Type	Centerline Distance
Swaged	0.100" typical, 0.085" min.
Brazed	0.100" typical, 0.035" min.



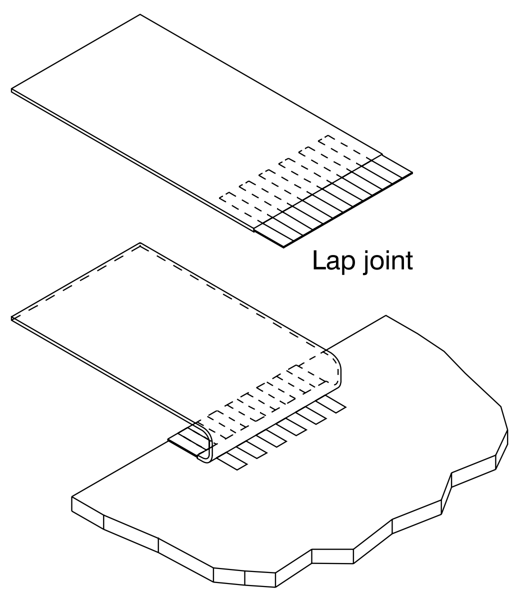
Crimped pins and sockets



Swaged pins with stiffener



End pins

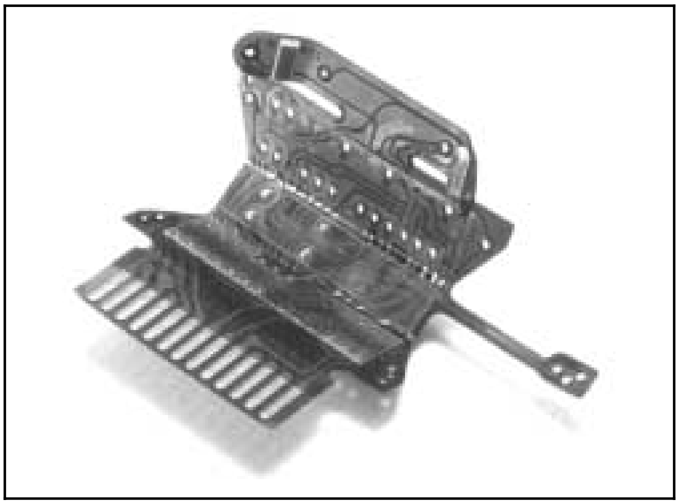


Lap joint

Stiffeners

Benefits of using a stiffener:

- Stiffeners are an inexpensive option for rigidizing pin areas, surface mount areas, or hole patterns for component mounting. Surface mount areas do not always require a stiffener, depending on component size, but a stiffener is recommended and will add very little to cost or bulk.
- Stiffeners can also be utilized to force a bend line in select areas. If this is the case, Minco can provide epoxy or silicone fillets for the edges of a FR-4 stiffener where flexing occurs.



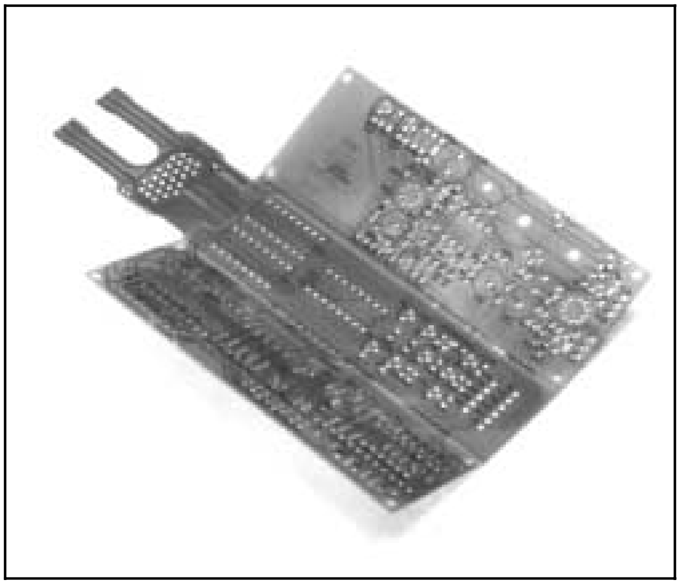
Small stiffeners used to force a bend line.

- Stiffeners reinforce solder joints and increase abrasion resistance.
- Stiffeners can also provide easier handling for soldering and automated pick-and-place. Circuits can be soldered while still in the stiffener and the customer can clip the circuit free after wave-soldering and circuit testing.
- Stiffeners make parts easier to handle during assembly.
- Stiffeners can be silk-screened with component mounting locations for rapid assembly.

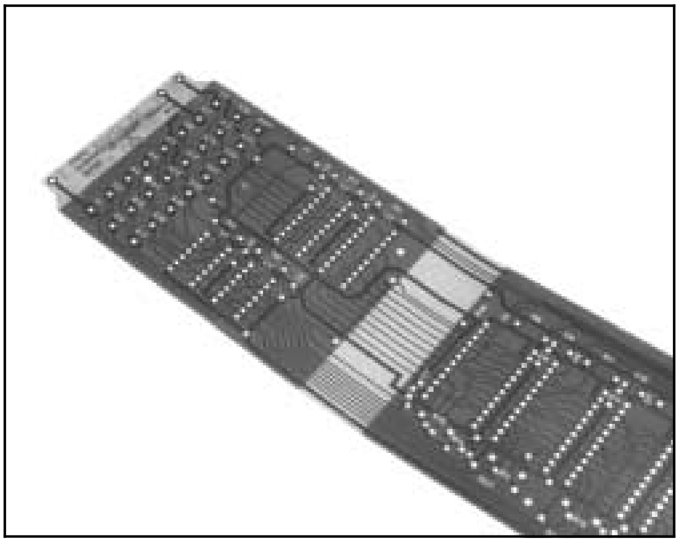
Stiffeners are commonly FR-4 or polyimide material, and are usually applied with modified acrylic adhesive, although PSA may be less expensive for small stiffeners. Keep in mind that PSA has less resistance than acrylic to high temperatures or solvents.

Standard FR-4 material thicknesses range from 0.003" to 0.125". Thicknesses over 0.031" are more expensive. In addition, the smaller a stiffener is dimensionally, the more difficult it is to work with.

Typical thickness for polyimide stiffeners is 0.005", but 0.001", 0.002", and 0.003" are also available. Polyimide stiffeners are less expensive than FR-4 stiffeners because they are punched on a die instead of routed with a drill bit. Because the polyimide stiffener lay-up procedure is performed with pins, registration is better. The polyimide stiffeners are trimmed along with the cover at the final blanking procedure, which guarantees perfect outside alignment.



FR-4 stiffeners

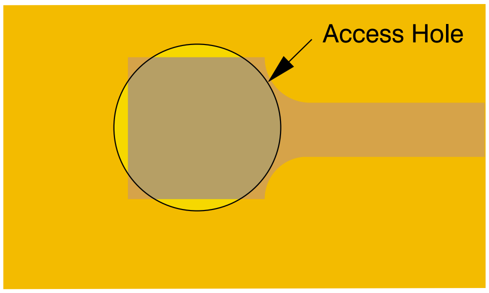


Polyimide stiffeners

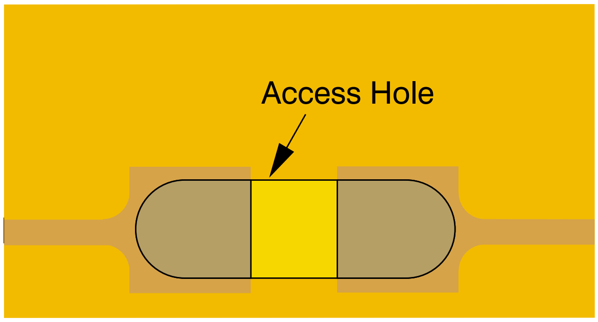
Surface Mount

Minco can provide flex-circuits with areas that are specifically designed for surface mount. Because covers are drilled, not silk-screened, round access holes are easier to provide. Square pads with round access holes are a good compromise. Square access holes will add to cost because the pad access area would have to be punched out with a punch-and-die. Below are some ideas for configuring pads for surface mount.

Photoimageable coverlay materials are also available, and can provide intricate openings for dense surface mount patterns.



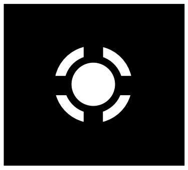
Surface mount area with round access hole



Surface mount area with slotted access hole

Soldering Tips

- Since polyimide absorbs moisture, circuits must be baked (1 hour @ 250°F) before soldering.
- Pads that are located in large conductor areas, such as ground planes, voltage planes, or heat sinks, should be provided with relief areas, as shown below. This limits heat dissipation and therefore makes 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.



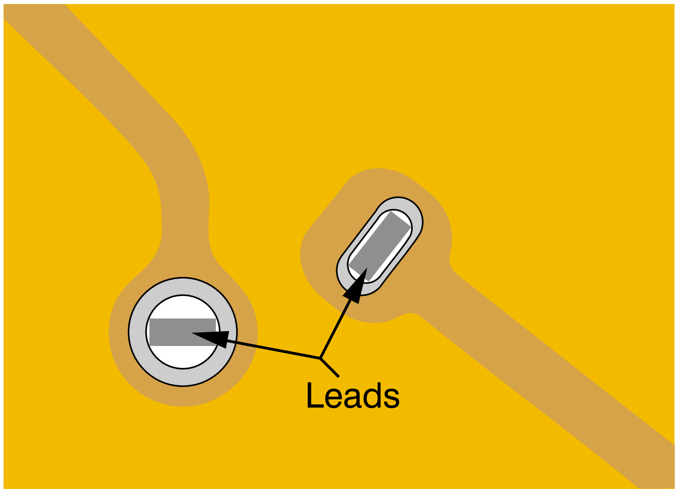
Solder relief area

- Minco can solder connectors or components as an added service. Soldering to IPC J-STD's is optional.
- Minco can supply circuits in panel form for easier component assembly.

Holes

Minco can drill through-holes as small as 0.0039". A 0.020" through-hole size is typical. Expect a finished hole tolerance of ± 0.003". For all circuits, the finished through-hole size should be about 0.003" to 0.010" in diameter larger than the component lead. This depends on the number of leads per component and the positional tolerance of the component leads.

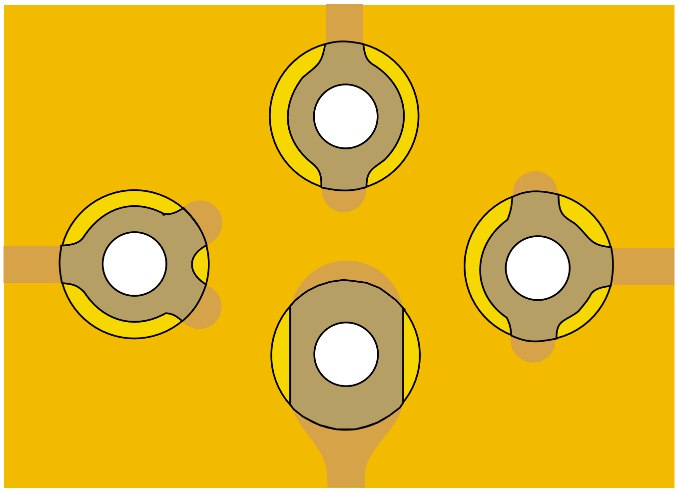
It is best to specify round, instead of slotted, through-holes. This will reduce drilling time and cost.



Round through-holes vs. slotted through-holes

Pads

Whenever possible, pads should be larger than the access holes and/or hold-down tabs (also called 'anchoring spurs') should be used. If space is an issue, hold-down tabs save more space than making the pads larger than the access holes. Hold-down tabs are especially important for single-layer circuits because a single-layer does not have the added strength of a plated through-hole. There are a variety of hold-down tab designs available.



Hold-down tabs

Pads, access holes, and annular ring general requirements

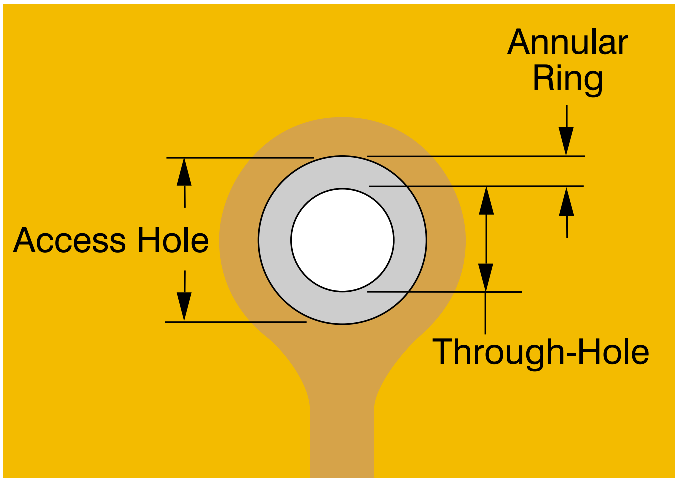
Feature	Single-Layer	Double-Layer	Multilayer
Pad*	0.060" + t	0.030" + t	Outer pad: 0.030" + t Inner pad: 0.025" + t
Major access hole*	0.050" + t†	0.030" + t	0.030" + t
Minor access hole‡	0.015" + t	0.015" + t	0.015" + t

t = nominal through-hole diameter

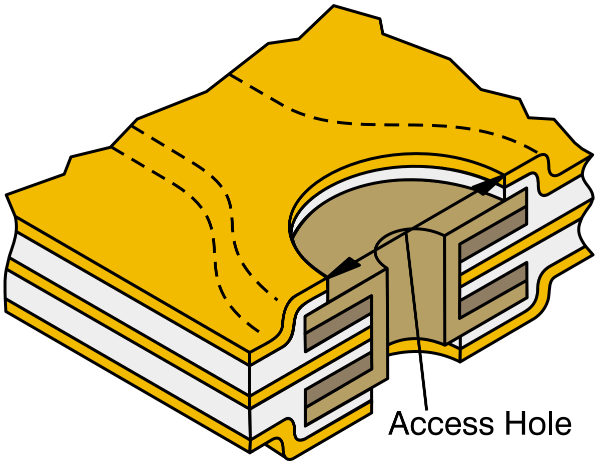
*Pad and major access hole design requirements are based on typical annular ring requirements of 0.015" minimum for a single-layer, 0.005" minimum for a double-layer or more, and 0.002" minimum for multilayer innerlayers.

†If hold-down tabs are used, then 0.030" is allowable.

‡Customer must allow tangency (see the Glossary for a definition of tangency).

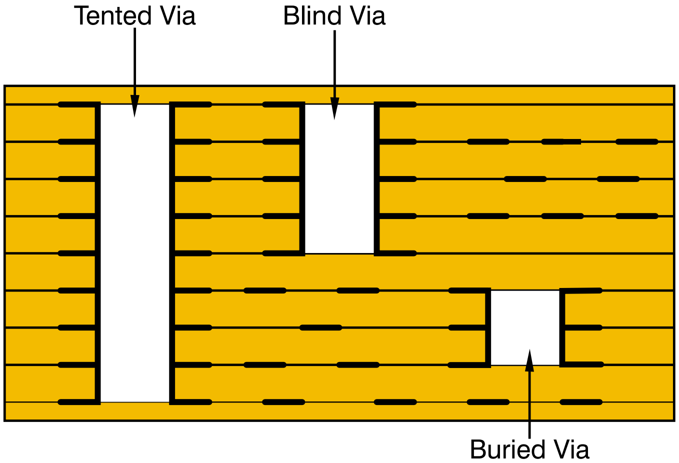


For more information on access holes (major and minor) annular ring, pads, and through-holes, see the Glossary on pages 24-27.



Vias

Minco can provide circuits with covers that have no access holes exposing the vias (called 'tented vias'). Minco can also provide blind and buried vias in multilayer and rigid-flex circuits. Blind vias connect the top or bottom conductor layer to adjoining layers, but the via does not extend through all layers. A buried via only connects internal layers and is not exposed in the finished circuit. Blind and buried vias increase circuit cost, but they free up space for additional conductors on the non-drilled layers.



Stiffener Holes

Stiffener holes should be a minimum of 0.015" in diameter larger than the access holes. It is better if the access hole underneath the stiffener hole is a minor access hole in order to increase the stiffener web between holes and to prevent potential solder wicking between the stiffener and the circuit. The customer must allow tangency. Round stiffener holes are less expensive than slotted stiffener holes and, as mentioned before, thinner stiffener material (less than 0.031") is less expensive to process.

Marking

Minco can meet all marking requirements. Etched marking, stamped ink marking, and screen marking are available for flex-circuits. Etched marking does not add to cost, since it is produced in the standard etching process. We recommend against etched marking of revision levels, as a revision change will then require an artwork change. The revision level should either be omitted or ink marked on the outside of the circuit. For ink marking, unless a specific ink is specified, a one-part permanent black epoxy is used to mark the circuits.

If desired, Minco can mark stiffeners and covers with component mounting locations.

Special considerations for rigid-flex

- Before designing a rigid-flex circuit, make certain that it is truly what you need. If the circuit only has a few layers, stiffeners are a cheaper option than rigid-flex.
- It is most cost effective to build a rigid-flex with an even number of layers. All rigid portions of the circuit should have the same number and stack-up of layers.
- For realistic yields, the thickness of the circuit should be no more than five times the hole diameter, though Minco prefers a 3 to 1 through-hole aspect ratio.
- Minco builds circuits of 16-18 layers, but costs increase significantly above 10-12 layers.
- Expect a trim tolerance similar to that of a steel rule die from hole-to-border and border-to-border. Hole placement within a cluster of holes and from cluster-to-cluster will have a ± 0.005" tolerance.
- Minimum interior border radius of 0.047" is standard, but smaller radii are available.
- Utilizing unbonded layers can increase flexibility where there are many flex layers, but this option can be more expensive. Specify unbonded layers only in areas of the circuit that will bend.
- Minco can provide an epoxy or RTV fillet on stiffener edges that will bend or flex.
- For rigid-flex circuits, it is less expensive to have plated through-holes in the rigid portions only.
- Minco can provide blind and buried vias in rigid-flex circuits.

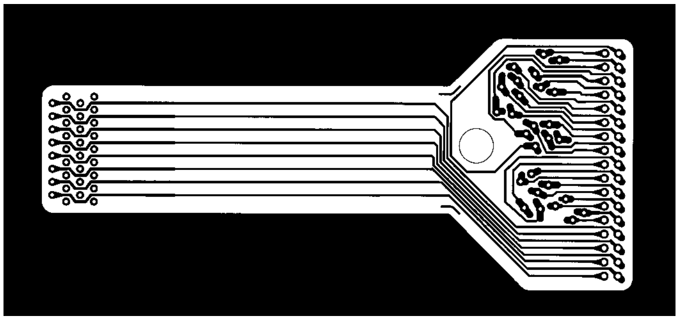
Designing CAD artworks

This section provides the information necessary for designing CAD artwork that will meet the tolerance and quality requirements for a flex-circuit. A correctly designed artwork will prevent unnecessary and costly delays in the initial shipment.

Most CAD artwork is customer supplied. Minco can generate CAD artworks at additional cost. To generate an artwork, Minco needs:

- Outline dimensions and tolerances. See page 10.
- Location and size of conductor pads. See the table on page 19.
- Minimum conductor widths, minimum spaces between conductors, and conductor thickness. These will depend on current carrying capacity, dielectric, and the flexibility requirements of the circuit. See the nomograph on pages 12-13.
- Conductor routings. Drawings of conductor paths are preferred, but a point-to-point connection list or schematic is acceptable.
- Locations where the circuit will be bent, if any, and required flexibility at these locations (i.e. bending for installation or a dynamic application).

We prefer that you supply CAD generated data as a minimum. Photoplotted film of each circuit, shield, and silkscreen marking layer can speed the checking process and reduce the possibility of errors. If you cannot furnish CAD data, we will digitize the physical artwork at additional cost.



CAD-generated artwork

Minco can accept CAD data in the following forms:

- Gerber photoplotter code: Embedded aperture code (RS-274-X) format, or with separate, detailed aperture list.
- AutoCAD DXF
- HPGL format
- AutoCAD DWG

Other forms may be acceptable–contact Minco for details.

Minco accepts transfer media in the following forms, which are listed in order of preference:

- E-mail: Ask Minco Sales Engineer for address
- FTP: Ask Minco Sales Engineer for address
- Floppy (3½"), Zip disk, or CD: IBM-PC format

Guidelines for all formats:

- Single entity draws for conductors are required.
- Single pad flashes are required.
- Minimum entities for conductor to pad transitions ('fillets') are preferred.

Guidelines for DXF:

- Place artwork data, part outline, hole centers, soldermask, coverlay, silkscreen marking, etc. on separate, individual CAD system layers.
- Polygons or zero width line draws for irregular pad shapes and shield area outlines are preferred instead of filling in these shapes.
- Supply arcs and circles. Do not convert arcs or circles into segmented lines.
- Avoid supplying only conductor outlines, as it increases set-up cost. If you do supply conductor outlines, include a supporting CAD system layer with proper line width conductors and pads.

Guidelines for Gerber:

- Incremental or absolute – absolute is preferred.
- 2.3 or 2.4 data. We prefer 2.4, which specifies 2 digits to the left of the decimal point and 4 to the right.
- The decimal point may or may not be omitted – we prefer that it be omitted.
- Leading/trailing zeros may be present or omitted – we prefer that the leading zeros be omitted.
- We prefer character code 7-bit ASCII, but 8-bit ASCII, EBCDIC, EIA, or BCD is acceptable.
- Commands may be terminated with an asterisk (*) or a dollar sign ($) – we prefer the asterisk.
- Each coordinate is followed by:
  - D01 – Draw line
  - D02 – Move without drawing
  - D03 – Flash a pad

Example of a preferred format:

```
D72*
X29750Y43800D03*
```

In addition:

- Line draws require 'Round' apertures be used.
- Pad flashes may use 'Round,' 'Square,' 'Oval,' and 'Rectangular' apertures. Any other special shapes must be accompanied by a detailed drawing of the aperture required.
- When conductors change direction, chamfer or radius corners. All outside conductor corners will have a radius equal to one half the conductor width.
- Large copper planes or shielded areas should have as few fill lines as possible. Do not supply files with 'redundant' fill patterns (i.e. overlapping vertical and horizontal fill lines). If Gerber conversion allows, specify the following fill apertures to be used:
  - Fill #1 – 0.005
  - Fill #2 – 0.010
  - Fill #3 – 0.020
  - Fill #4 – 0.050
  - Fill #5 – 0.100

When sending your photoplotter code, please include:

- The format of the data
- An aperture wheel listing (when required)
- The number of files supplied and their correspondences to the circuit layers.

Below is an example of the information to include with your photoplotter code. For best service, please provide all the following file information:

```
Data Type — Gerber
Character code — 7-bit ASCII
Format — 2.4 absolute
Decimal point — Suppressed
End of block character — *
Zero suppress — Leading
Aperture listing:

    Position    DCode    Aperture
    1           D10      .006 Line
    2           D11      .010 Square
    3           D12      .060 x .040 Oval
    4           D13      .065 x .045 Rect.
    5           D14      .055 Round
    .           .        .
    .           .        .
    .           .        .
    23          D72      .095 Round
    24          D73      Target

6 files on disk:
    COMP.GBR:  Artwork Layer 1
    SOLDER.GBR:  Artwork Layer 2
    SS1.GBR:  Silk Screen Layer 1
    SS2.GBR:  Silk Screen Layer 2
    DRILL.DRL:  Drill Pattern
    APERTURE.LST:  Gerber Wheel List
```

Artwork checklist

Minco strongly encourages you to use the artwork checklist that is provided. If you answer 'yes' to all the criteria, your artwork will probably not need adjustment. Note: Depending on the size/complexity of the circuit, the criteria may differ.

Dimensions

- Does the border match the print dimensionally?
- Does the artwork match the print dimensionally?

Etched Marking (if any)

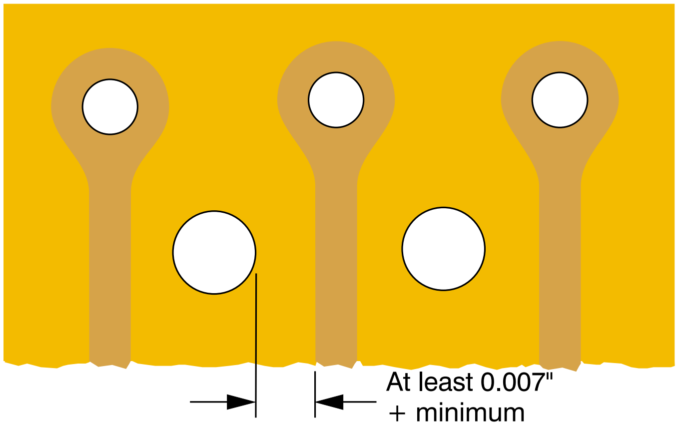
- Are all features of the letters and symbols at least 0.010" wide?
- Are letters and symbols sufficiently clear of conductors and borders?

Border and Cutouts

- Is there a trim border (part outline) on at least one layer?
- Do nominal borders allow for tolerances in the table on page 10?

Non-Wiring Hole Clearance

- Are non-wiring holes at least 0.007" (more preferred) over the specified minimum distance from conductors and borders?



Non-wiring hole at least 0.007" + minimum from conductor.

Conductor Width

Is artwork conductor:

- at least 0.001" over the specified minimum for unplated ½ oz. copper?
- at least 0.004" over the specified minimum for unplated 1 oz. or thicker copper?
- at least 0.005" over the specified minimum for plated copper?

Conductor Spacing

Is artwork conductor space:

- at least 0.001" over the specified minimum for all unplated copper?
- at least 0.002" over the specified minimum for all plated copper?

Conductor Routing

- Are conductors perpendicular to bend lines?
- Have you avoided the I-beam effect? (See page 17 or Glossary for a definition of I-beam effect.)

Pads and Annular Rings

- Are there pads on all layers for all plated through-holes?
- Are non-exposed pads (via pads) 0.020" larger than the through-holes (not applicable for micro vias)?
- Are center locations provided for all drilled holes and/or slots?
- Are the annular rings on all holes at least 0.007" larger than the specified minimum?
- Are access holes in cover layers at least 0.007" (more preferred) over the specified minimum distance from conductors and borders?
- Are all pads filleted?

Providing information for a quote

Information required for ballpark quote:

- Quantities desired
- Number of layers
- General size of the circuit
- Features such as stiffeners, pins, connectors, etc.

Information required for firm quote:

- Quantities required
- Complete physical shape
- Materials
  - Conductors
  - Insulator
  - Stiffener
  - Other
- Number of layers
- Plating requirements
- Applicable specifications
- Unusual areas of the circuit that Minco should be aware of, such as unbonded or cut-away areas
- Tolerances clearly outlined (geometric profile preferred)
- Other requirements–conductor spacing, conductor width, border, etc.
- Special marking or packaging requirements
- Testing requirements–type, percent to be tested, and frequency. Is IPC-6013 or MIL-P-50884 testing required?
- Additional components that Minco is expected to supply/assemble (inform Minco of preferred suppliers if the components are unusual).

Information required for manufacturing:

- All the information that is required for a firm quote *Plus:*
- Drawing
- Artwork for conductors, coupons, and screened marking, if applicable (unless Minco is to generate)
- More detailed information as far as circuit dimensions and requirements

Supplying drawings:

The 'perfect drawing' will provide the following information:

- Cross-section diagram (i.e. material stack-up)
- Outline drawing of circuit
- Material listing
- Specifications
- Hole chart
- Feature chart (i.e. minimum conductor width and spacing, and any other minimum spacing requirements)
- Dimensional tolerances
- Special plating requirements
- Marking requirements
- Testing requirements
- Special packaging requirements

Reference documents

IPC Specifications*

- IPC-2221, Generic Standard on Printed Board Design
- IPC-2223, Sectional Design Standard for Flexible Printed Boards
- IPC-6013, Qualification and Performance Specification for Flexible Printed Wiring

Military

- MIL-P-50884, Printed Wiring, Flexible and Rigid-Flex
- MIL-STD-2118, Flexible and Rigid-Flex Printed Wiring for Electronic Equipment, Design for

Minco Documents

- E.I. 206, General Specifications for Flexible Etched Circuitry
- Minco's standard workmanship requirements.
- Bulletin FC-301, Flex-Circuits
- General brochure with applications and basic design information.
- Bulletin FC-1, Flex-Coils™

*For more IPC specifications, contact:
IPC
2215 Sanders Road
Northbrook, IL 60062-6135 USA
Tel: 847-509-9700 / FAX: 847-509-9798
URL: http://www.ipc.org

Minco test capabilities

When specifying testing, consider your needs carefully, as over-specification can greatly raise the cost of a circuit. Minco encourages electrical testing. It is required on all multilayer, rigid-flex, and factory-formed circuits that are fabricated to MIL-P-50884, and certain classes of IPC-6013.

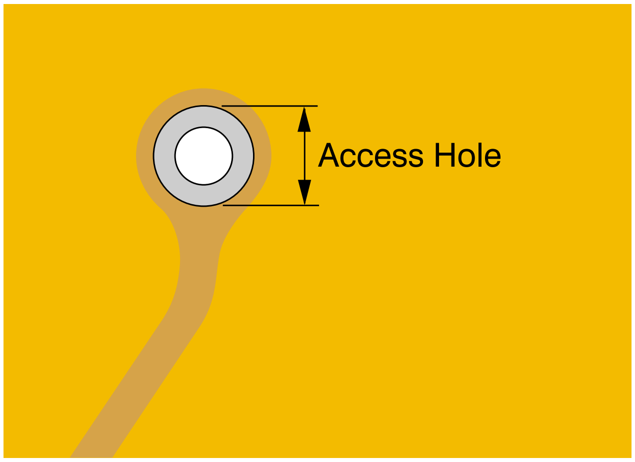
See the table to the right for information on Minco's test capabilities.

Minco Can Test For. . .	Range of Operation
IPC-6013 and MIL-P-50884 conformance	N.A.
Complete dimensions	Resolution: 4 decimal places Accuracy: 0.001" per foot.
Electrical continuity	1Ω to 10KΩ; suggest 5Ω Stimulus: 0.01V to 5.0V
Insulation resistance	10KΩ to 100MΩ at 10V to 250VDC Suggest 100 MΩ at 100VDC
Thermal shock	-70 to 200°C
Moisture resistance	Up to 98% relative humidity
Plating thickness	Down to 0.000001"
Flexibility	0 to 999,999 flexes
Microsections	Viewed at up to 1000X
Dielectric withstanding voltage	0 to 100 micro amps Suggest 20 micro amps, 10 second duration Stimulus: 6500 volts maximum, AC or DC

Glossary

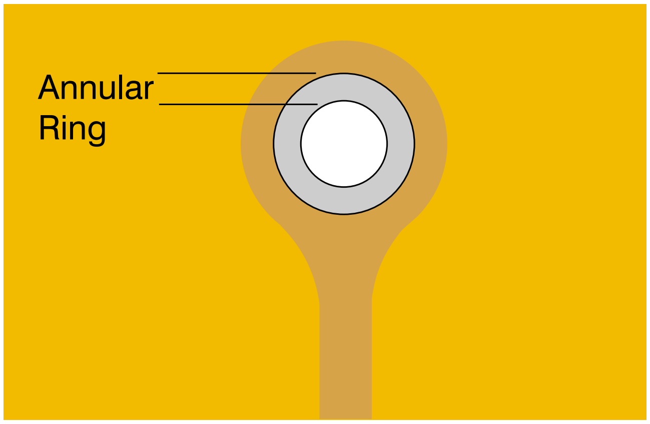
Access Hole

A hole in the cover layer of a circuit that allows electrical access to a flex-circuit's conductor pads and through-holes.



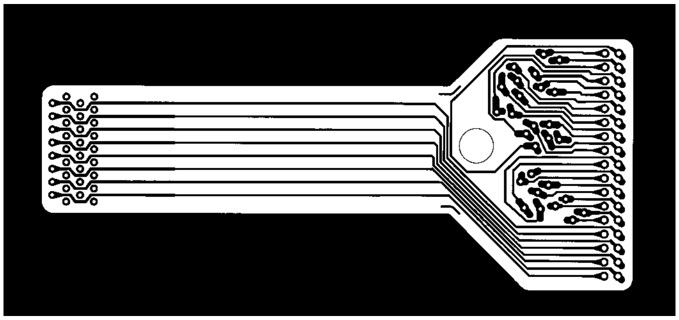
Annular Ring

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



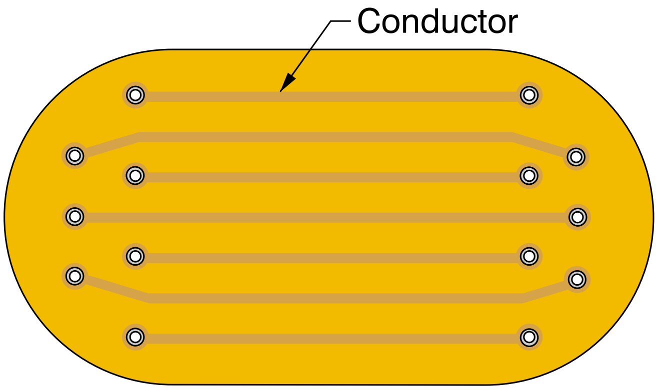
Artwork

The original pattern of conductor strands for a flexible circuit.



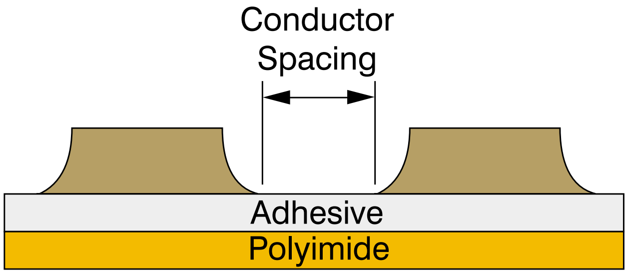
Conductor

The path that carries an electrical current from one point to another. Minco's flex-circuit conductors are commonly found in the form of copper strands.



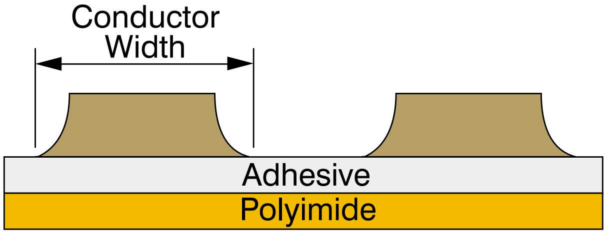
Conductor Spacing

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.



Cover

Insulator material that is laminated to an etched element. Covers can be located on the inner or outer layers of a circuit. Internal cover layers are found in the unbonded regions of a circuit.

Dynamic Application

The use of a flex-circuit in an environment that requires flexing in use.

Flex-Circuit

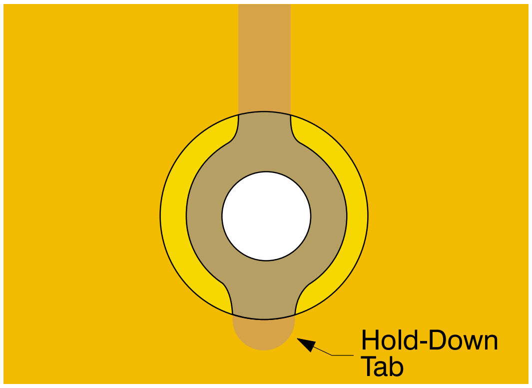
Flexible printed circuits made from etched foil conductor strands. The conductor strands are laminated between insulation layers. Flex-circuits can vary in complexity from the simplest single-layer circuit to a complex multilayer.

Flex-Coil™

Flex-circuits with internal or attached wire-wound coils.

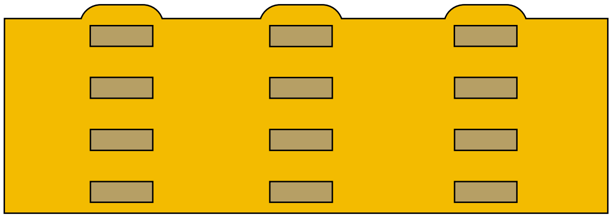
Hold-Down Tabs

An extension of foil on a conductor pad that aids the pad in gripping to the substrate insulation. Hold-down tabs are also referred to as 'anchoring spurs.'



I-Beam Effect

The tendency of a flex-circuit to have reduced flexibility and fractured conductor strands if the conductor strands are layered directly over each other, instead of being staggered from layer to layer.



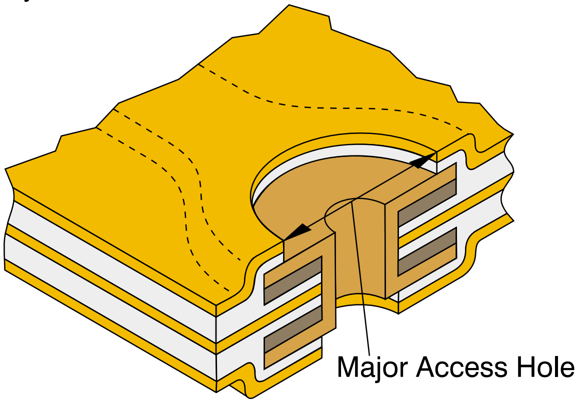
I-beam effect

Impedance

The measurement in ohms of the apparent resistance of an AC circuit. Impedance depends on several factors: DC resistance, capacitance, inductance of the line, the width of the conductor strands, and the conductor spacing relative to ground and insulating layers.

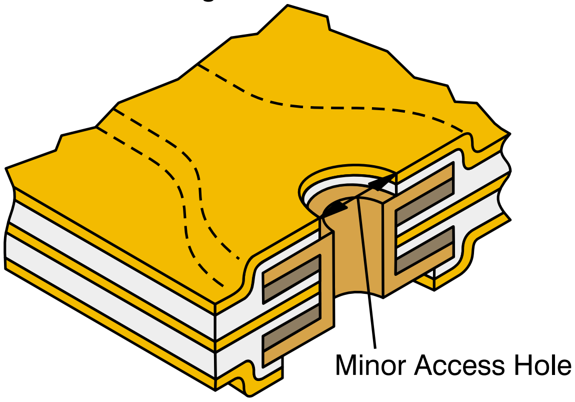
Major Access Hole

An access hole (see 'Access Hole') that is large enough to expose a major portion of a conductor pad, which is usually coated with solder.



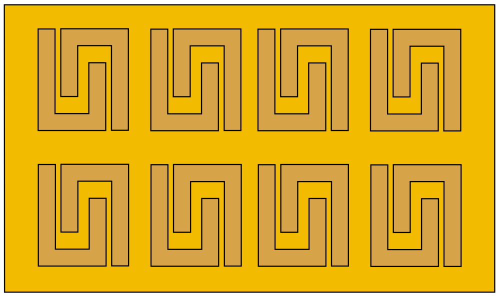
Minor Access Hole

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



Nesting

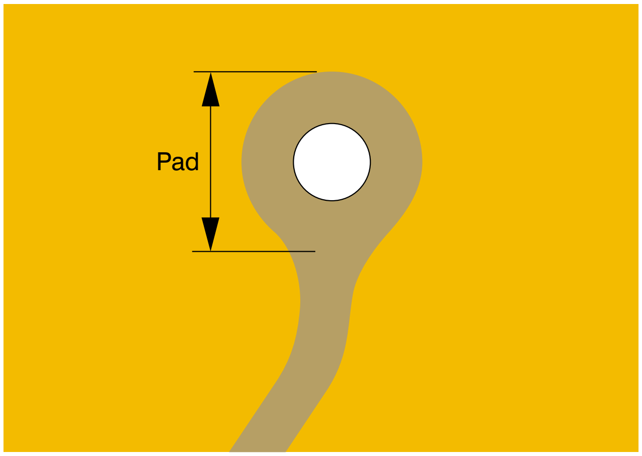
Designing circuits so that they lay closely together on a panel during production. This maximizes the usage of panel space, thereby minimizing production cost.



Circuits nested on a panel

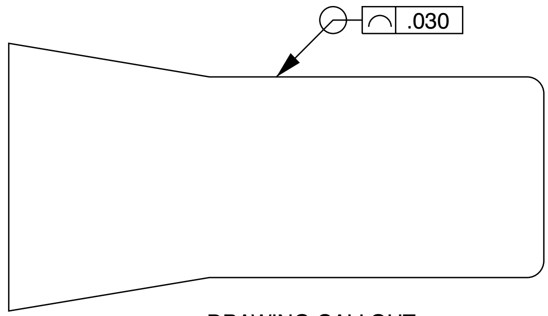
Pad

The portion of a conductor, usually surrounding a through-hole, that is used to connect a component for an electrical connection. Pads are sometimes referred to as 'terminals' or 'lands.'

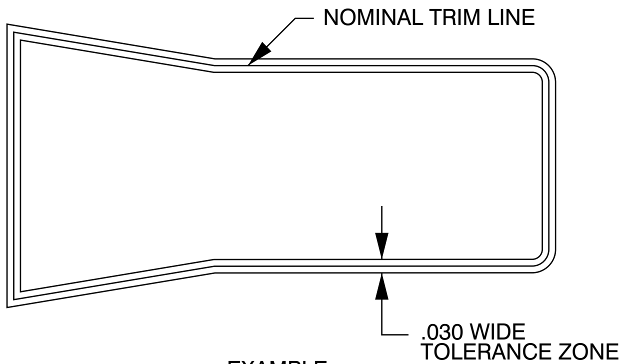


Profile Tolerance

Dimensional tolerancing where the part trim line is contained within a tolerance zone consisting of the area between two parallel lines, separated by the specified tolerance. For example, a circuit to be trimmed with a steel rule die might have a tolerance of ±0.015" (a 0.030" wide profile tolerance zone). The circuit trim line could vary anywhere inside the zone.



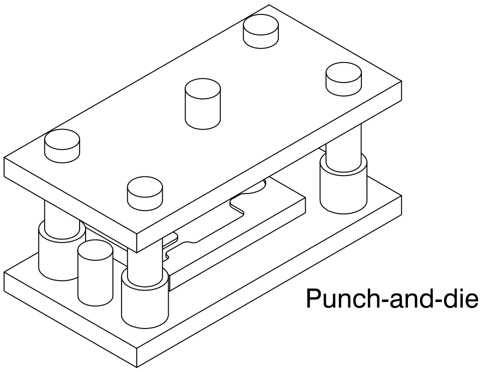
DRAWING CALLOUT



EXAMPLE

Punch-and-Die

Hard-tooling that is used in a punch press. A punch-and-die consists of two precisely matched metal plates held in special die shoes. When the punch press is activated, the plates come together in order to punch a specific pattern into material.



Rigid-Flex

A circuit containing both rigid and flexible areas. The rigid layers have conductors and plated through-holes connecting them to other layers.

Selective Plating

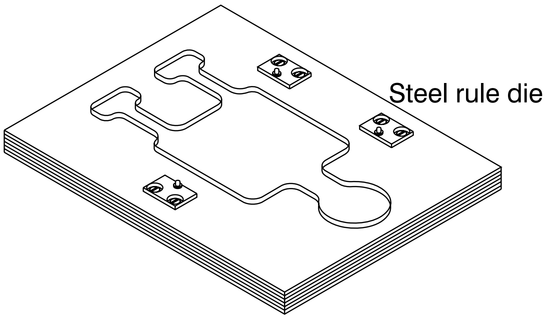
A method of plating flex-circuits so that only the circuit's through-holes and surrounding pads are plated. This greatly adds to a circuit's flexibility.

Static Application

The use of a flex-circuit in an environment that requires flexing during installation and maintenance, but not in operation.

Steel Rule Die

A tool used in a punch press, consisting of steel cutting blades in a pattern, embedded into a maple plywood base.



Stiffener

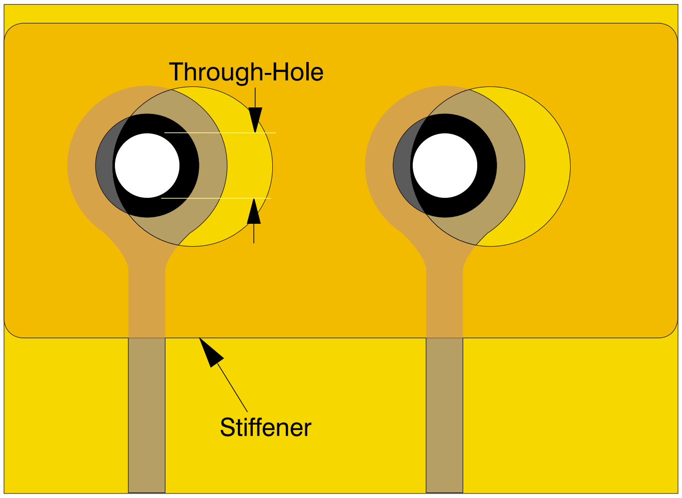
Flexible or rigid pieces of material (usually Kapton or FR-4) that are added to flex-circuits in order to reinforce them for component mounting. There are no conductors on the stiffeners, as compared to rigid-flex circuits.

Substrate

A layer of insulator material that is bonded on one or both sides with foil.

Tangency

A condition that occurs when the edge of a stiffener or cover access hole is flush with the edge of a through-hole.



Tear Stops

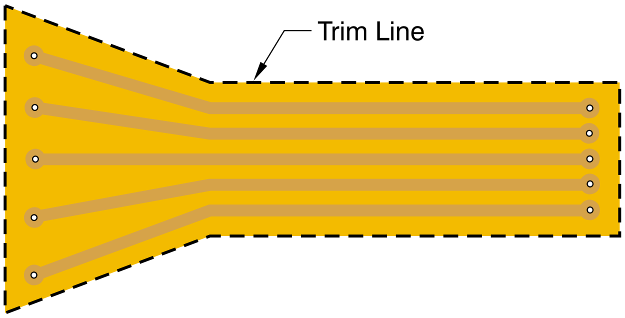
Copper, Kapton, or Teflon guards that are located in the inner corners or Kapton-insulated flex-circuits in order to prevent propagation of tears.

Through-Holes

Holes that are drilled through the layers of a flex-circuit in order to have component access to those layers. Connection from one layer to the next is provided by plating the through-hole walls with a thin layer of copper.

Trim Line

The area defined by a design engineer as the final cut-out area around a flex-circuit.



Unbonded Areas

A flex-circuit design technique that involves providing an insulating layer between every conductive layer of a flex-circuit, but with no adhesive bonding between the insulating layers in certain areas of the circuit. This technique improves circuit flexibility.

Via

A plated through-hole with no cover access holes that provides connection for internal layers.

Disclaimer: The recommendations in this Application Aid are guidelines only. You must ultimately determine whether a flex-circuit design is suitable for your application. Minco accepts no liability beyond its standard warranty.

Subject Index

Minco manufactures a wide range of circuits, from complex rigid-flex to assemblies incorporating pins, connectors, and components.

MINCO
ISO 9001

Minco Products, Inc. (Main Office)
7300 Commerce Lane
Minneapolis, MN 55432-3177
U.S.A.
Tel: (763) 571-3121
Fax: (763) 571-0927

Stock order desk:
Tel: (763) 571-3123
Fax: (763) 571-9142

Internet:
sales@minco.com
www.minco.com

Minco S.A.
Usine et Service
Commercial, Z.I.
09310 Aston, France
Tel: (33) 5 61 03 24 01
Fax: (33) 5 61 03 24 09

Minco EC
Hirzenstrasse 2
CH-9244 Niederuzwil
Switzerland
Tel: (41) 71 952 79 89
Fax: (41) 71 952 79 90