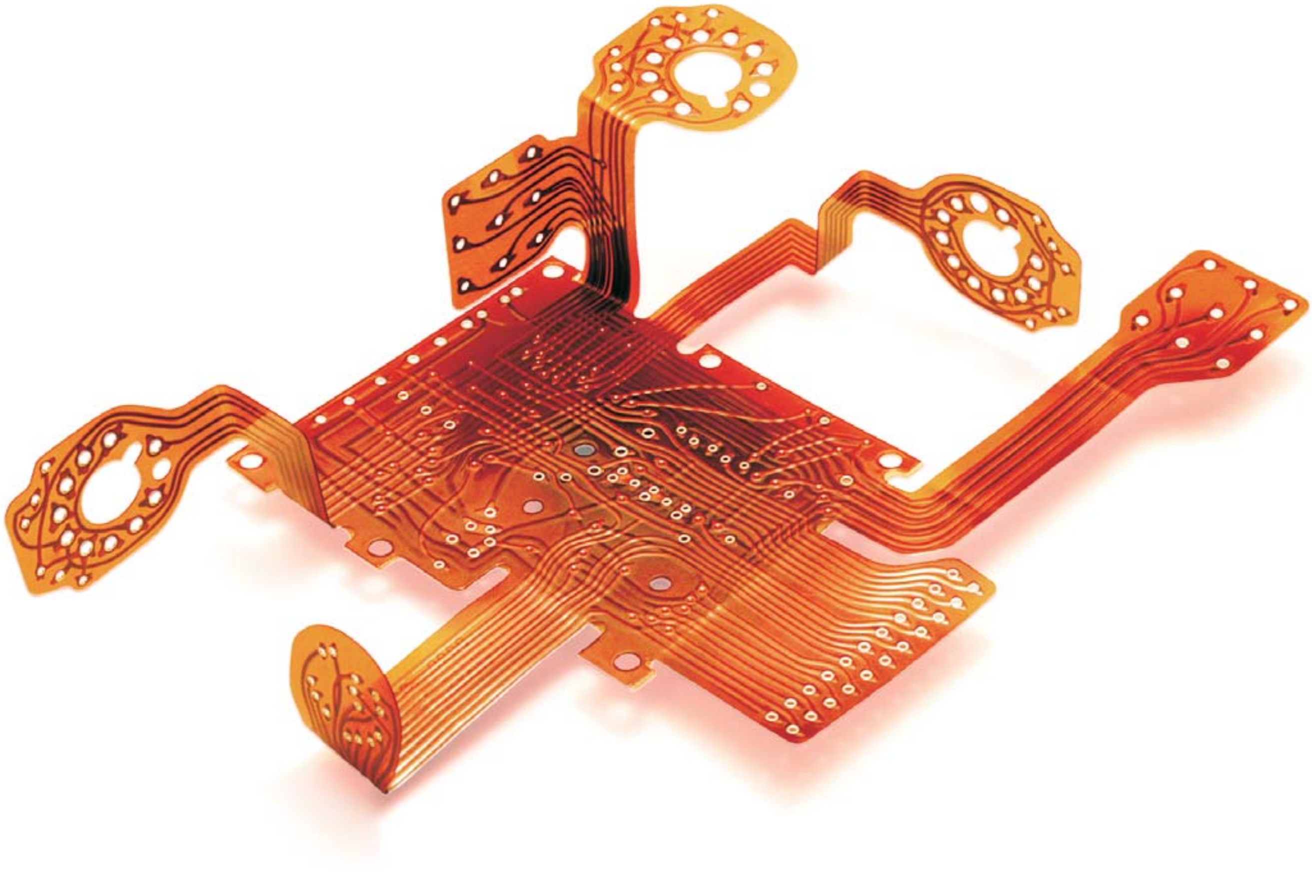
Flex Circuit Design Guide



Minco Flex Circuits

Table of Contents

Introduction

Flex Circuit Design Process

Ordering and Delivery

Glossary

Purpose of this Design Guide

The purpose of this design guide is to enable you to design a highly reliable, flexible printed circuit optimized for manufacturability.

While using this guide, keep in mind that the design information provided is only a suggestion. Minco takes pride in manufacturing flex circuits considered difficult to build. In most cases, we do build above and beyond the "standard" circuit specifications, provided that the circuit design and type allow for it.

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

Types of Flex circuits

- **Flex;** Flexible version of Printed Circuit Board (PCB), with unique capabilities. Flex circuits offer the same advantages of a printed circuit board: repeatability, reliability, and high density but with the added "twist" of flexibility and vibration resistance. The most important attribute compelling designers to adopt flex circuit technology is the capability of the flex circuit to assume three-dimensional configurations.
- **Rigid-flex;** A blend of rigid and flex emphasizing the best of both constructions, adding synergistic capabilities neither possess. In its most typical configuration, the rigid-flex is as a series of rigid PCBs joined by integrated flex circuits (with emphasis on the high percentage of rigid area content). There are many excellent possibilities for circuits designed primarily as a flex circuit with the addition of integrated rigid areas.

  Rigid areas provide excellent hard mount points for components, connectors and chassis while flex areas offer dynamic flexing, flex to fit, and component mounting poised to take advantage of these low mass and vibration resistant zones. This blending leads to creative solutions for your most demanding applications.
- **Flex-coils;** Custom wire-wound or etched coils may be integrated with any of our flex circuit or rigid-flex board types. Whether bonded to the surface or encapsulated in a high dielectric and abrasion resistant covering, these assemblies offer special capabilities to your coil designs.

Benefits of Flex Circuits

High reliability

Repeatable installation

Compared to discrete wiring, or ribbon cable, a flex circuit offers a customized repeatable routing path within your assembly. This gives you dependability where you need it. A flex circuit's longevity can reduce service calls.

Harsh environments

Standard practice for flex boards is to cover the conductors with polyimide. This dielectric layer protects your circuits far beyond the capability of simple soldermask. Other base and cover materials are available for a broad range of ambient conditions.

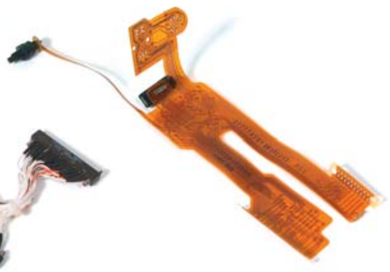
Long duty cycles

By design, a flex circuit can be made very thin, yet robust enough to withstand thousands to millions of flexing cycles while carrying signal and power without a break.

High vibration

Under vibration and/or high acceleration, a flex circuit's ductility and low mass will reduce the impact upon itself and solder joints. By contast, a PCB's higher vibrational mass will increase stresses upon itself, components and solder joints.

Before: A tangle of wires connects four circuit boards.



The flex circuit solution: A single circuit with 7 stiffeners and 2 connectors provides all the needed interconnects.

After: The package is neat, lightweight, and less susceptible to connection failure.

Superior packaging options

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

Flex vs. wiring harness

- **Space and weight reduction;** A single flex circuit can replace several hardboards, cables, and connectors.
- **Fast assembly;** Flex circuits eliminate the need to color code and wrap bundles of wire, reducing the chance of assembly rejects and in-service failures. Total installed costs are lower, especially with volume production.
- **Repeatable wire routing;** Eliminate wire routing errors; reducing test time, rework, and rejects
- **Robust connections;** Flat foil conductors dissipate heat better, carrying more current than round wires of the same cross-sectional area. Conductor patterns in a flex circuit maintain uniform electrical characteristics. Noise, crosstalk, and impedance can be predicted and controlled.

Flex vs. hard board (PCB)

- **Versatile shape;** The most important attribute compelling designers adopt flex circuit technology is the capability of the flex circuit to assume three-dimensional configurations.
- **Lower mass;** With a little experimentation and imagination, a flex circuit can save up to 75% of the space and/or weight of conventional wiring.
- **Vibration resistance;** Recurring costs are lower than many wire harnesses, and since a flex circuit is more resistant to shock and vibrations than a PCB, repair and replacement costs are less.

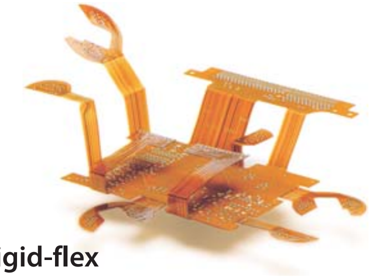
Rigid-flex

- **Double side component mounting;** Rigid-flex are the ideal solution for flex circuits where surface mount components will be mounted on both sides of the board.
- **Total cost of ownership;** The maximum benefit of rigid-flex is realized when the complete installation is reviewed for total cost of ownership. Using rigid-flex eliminates connections in the flex-to-rigid transitions while improving impedance control.
- **Most capable/Maximum vibration resistance;** Lets you integrate the best capabilities of resistant rigid areas and resilient flex areas.
- **High mass component mounting;** When mounting a high mass component, a rigid board is the right solution. A rigid-flex board gives you a smooth transition between rigid and flex areas while preserving the benefits of each.

Flex-coils

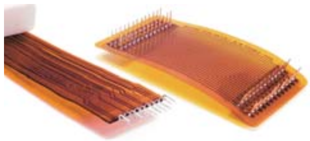
- **Custom coil winding;** State of the art equipment generates a highly repeatable component
- **Integrated assembly;** allows best packaging of your fragile coil in a flex circuit sub-assembly.

Design Options



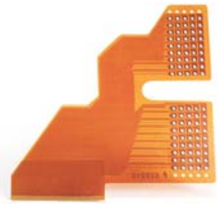
Rigid-flex

Hybrid hardboard/flex circuits can have up to sixteen layers. They replace bulky wire harnesses with compact, robust design!



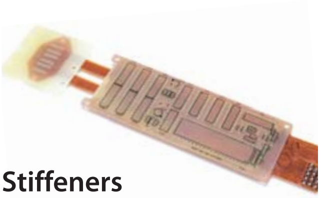
Pins

Minco can braze or solder pins to circuits, either through holes or as extensions to conductors.



Fine lines

0.002" conductors and spaces are possible.



Stiffeners

An inexpensive alternative to rigid-flex.

Connectors

Built-in connectors speed your assembly. Optional epoxy potting seals between the circuit and connector.

Shielding

Solid or patterned shield planes reduce noise and control impedance of signal lines. Use matched impedance flex circuits for high-speed signal integrity.

Wave Solder Carrier

Stiffener material frames the circuit to hold it flat during wave solder. After soldering, just clip out and install.

Factory forming

Factory formed circuits follow tight curves to save space.

Coils

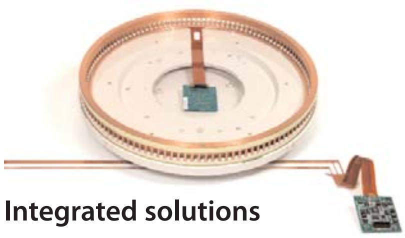
Minco's unique flat wound inductive coils can be laminated into flex circuits. Applications include pacemaker antennas and eddy current generators.

Surface mount

Combine the space and weight savings of surface mounting with those of flex circuits for the ultimate in high-density packaging.

Selective bonding

For better flexibility along circuit arms, individual layers are unbonded and allowed to flex freely. Each layer has it's own substrate and cover.



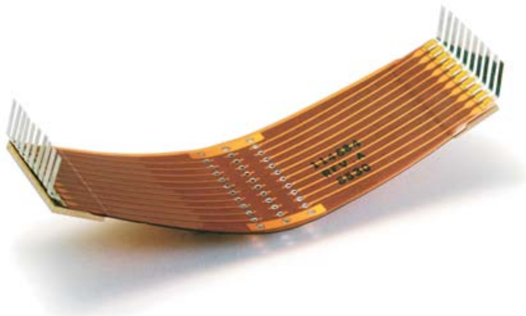
Integrated solutions

Minco integrates temperature sensors and etched-foil heaters with flex circuits for unified temperature control.

Flex Circuit Solutions

Cardiac devices

As cardiac devices have grown more sophisticated, many manufacturers have added remote telemetry for two-way communications. Mating coils inside and outside the body exchange the signals. This Minco circuit has an integral wire-wound antenna coil in addition to the etched conductors. The package is compact, rugged, and reliable.

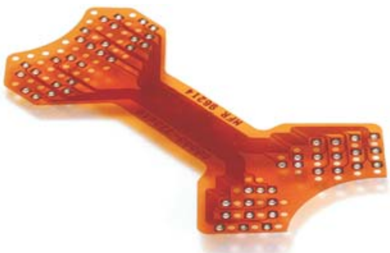


Integrated circuit testers

A high speed IC handler must make repeated electrical connections to chip leads as they pass through its test site. The answer is a Minco circuit with welded fingers for contact and shielding for controlled impedance and crosstalk. The circuit can withstand millions of flexures as it opens and closes.

Clinical analyzers

A diagnostic chemical analyzer must keep body fluid at a constant temperature during test operations. A Minco heater/sensor/flex circuit does the job singlehandedly. An etched-foil heater warms the sample, a wire-wound resistance thermometer senses temperature, and a flex circuit provides the link to control electronics.



Infrared detector modules

Forward-looking infrared lends night vision to aircraft, tanks, and foot soldiers. High density signal lines in the sensing module terminate through a fine-line circuit with 0.005" lines and spaces.

Military radio

The latest generation of all-purpose military radios have advanced features, compact size, and nearly indestructible construction. State-of-the-art packaging makes it all possible. An example is this Minco circuit, which has two 3-layer arms mated at a single connector.

Design Guidelines

Specification documents

Consult standard specifications and design documents pertaining to your application and circuit requirements.

IPC specifications* www.ipc.org

- IPC-2221A, Generic Standard on Printed Board Design
- IPC-2223, Sectional Design Standard 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-6013, Qualification and Performance Specification for Flexible Printed Wiring
- IPC-MF-150, Copper Foil for Printed Wiring Applications
- IPC Position Paper: Transitioning from MIL-P-50884C and MIL-PRF-31032 to IPC-6013 and Amendment 1

*For more information on IPC specifications, contact:
IPC
3000 Lakeside Drive, Suite 309 S
Bannockburn, IL 60015-1249 USA
Tel: 847.615.7100 / FAX: 847.615.7105
URL: http://www.ipc.org

Military www.dscc.dla.mil/Programs/MilSpec/DocSearch.asp

- MIL-P-50884 (inactive for new designs), Printed Wiring, Flexible and Rigid-Flex for Electronic Printed Wiring
- MIL-PRF-31032/3A, Printed Wiring Board, Flexible, Single and Double Layer, With or Without Plated Holes, With or Without Stiffeners, for Soldered Part Mounting
- MIL-PRF-31032/4A, Printed Wiring Board, Rigid-Flex or Flexible, Multilayer, with Plated Holes, with or Without Stiffeners, for Soldered Part Mounting

IPC recommends that companies using MIL-PRF-31032 specifications for printed circuits, alternatively specify that flexible circuits be supplied under IPC-6013 Class 3 performance requirements. Government agencies have generally accepted that this is a COTS (Commercial, Off-The-Shelf) equivalent to MIL-PRF-31032. Please see the IPC position paper which is available for download at IPC.org or Minco.com.

If your circuit must meet performance requirements of MIL-P-50884, MIL-PRF-31032 or IPC-6013, we urge you to read the IPC-2223 design specification for flexible circuits and follow its recommendations.

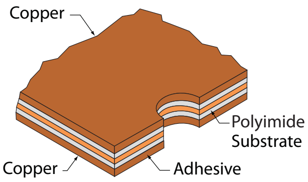
Minco documents www.minco.com

- Application Aid #31 - Designing for Flexibility and Reliability
- Flex Circuit Overview FCOV
- Flex-Coils™ Technical Specification FC01
- Minco/Omnetics Flex Circuit Interconnect Solutions Technical Specification FC04

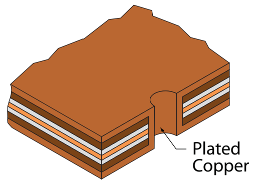
Design Guidelines

Manufacturing a flex circuit

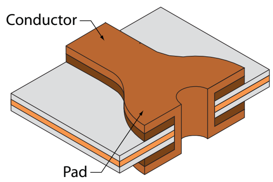
Building a flex circuit generally involves the same steps from circuit to circuit. However, certain circuit designs can add cost. For example, a single-layer circuit with access holes on both sides is more expensive than a single-layer circuit with access on one side, because the double-sided access hole circuit must have its substrate drilled separately. The adjacent flow chart and the illustrations below identify some cost driven 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.



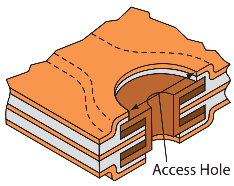
1. Double-sided material is drilled



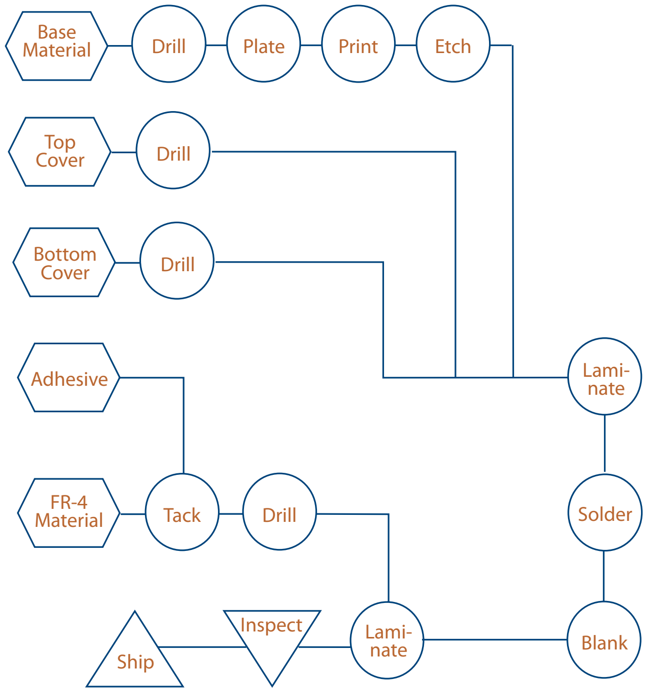
2. Through-holes are copper-plated



3. Copper is etched to create conductors and pads



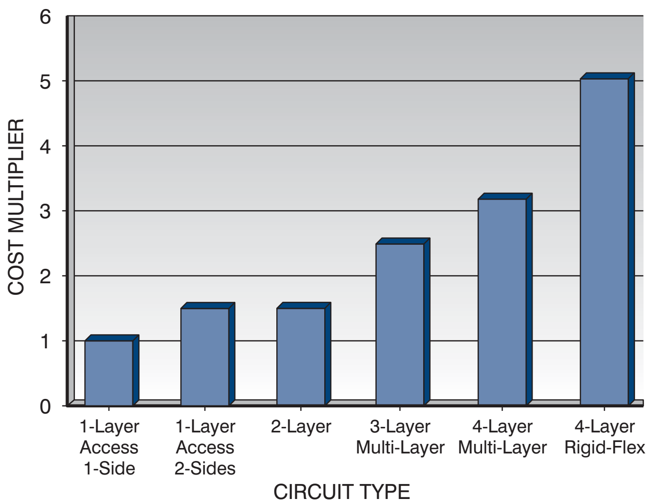
4. Polyimide covers are laminated over etched copper-



Cost impact of layer count

The information for the chart (right) was taken from a sample 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 consider all options to minimize cost. For example, use two circuits to do the job of one. Two double-layer circuits may be less expensive than one four-layer circuit. But the cost savings of the circuit may be offset by additional assembly requirements. Circuits can also be folded in order to save space and layers. Each situation is unique. A relatively small amount of time invested in project assessment can result in significant savings overall.

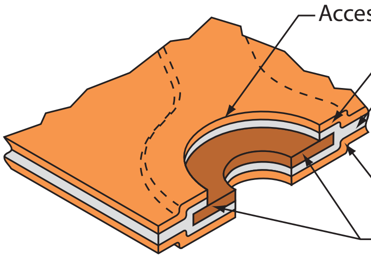


Design Guidelines

Circuit types

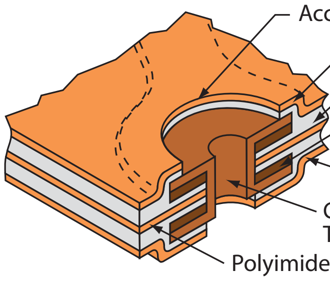
Single-layer

- IPC 6013, 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.



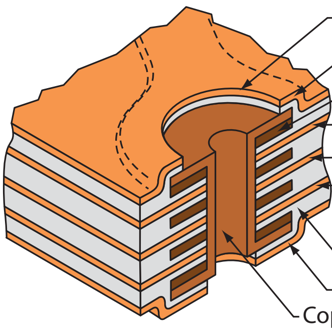
Double-layer

- IPC 6013, MIL-P-50884 - Type 2
- Two conductive layers with an insulating layer between; outer layers may have covers or exposed pads.
- Plated through-holes provide connection between layers.
- Access holes or exposed pads without covers may be on either or both sides; vias can be covered on both sides.
- Stiffeners, pins, connectors, components are optional.



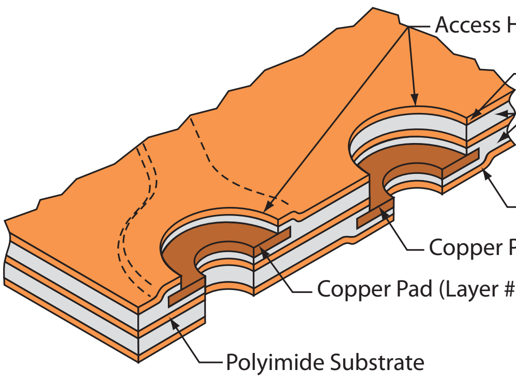
Multilayer

- IPC 6013, 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 provide connection between layers.
- Access holes or exposed pads without covers may be on either or both sides.
- Vias can be blind or buried.
- Stiffeners, pins, connectors, components are optional.



Multilayer, not plated through-holes

- IPC 6013, 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.

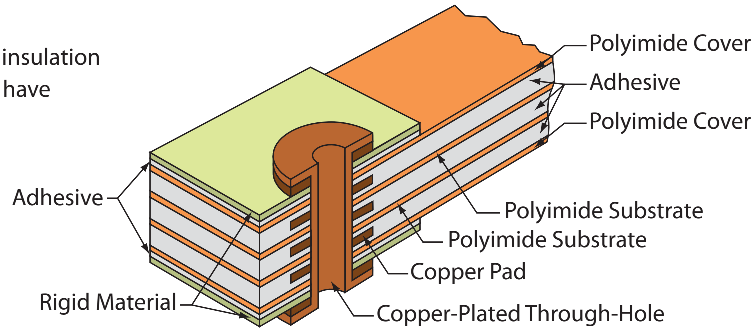


* Adhesiveless base material also available
**Cover may be replaced by photo-imagable coverlay (PIC)

Design Guidelines

Circuit types

Rigid-Flex

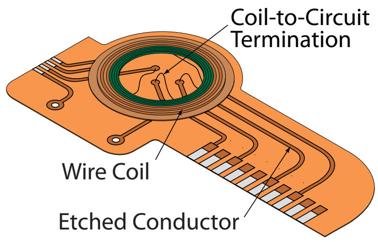


- IPC 6013, MIL-P-50884 - Type 4
- Two or more conductive layers with either flexible or rigid insulation material as insulators between each one; outer layers may have covers or exposed pads.
- Rigid-flex has conductors on the rigid layers, which differentiates it from multilayer circuits with stiffeners. Plated through-holes extend through both rigid and flexible layers (with the exception of blind and buried vias). Rigid-flex costs more than a standard circuit with stiffeners.
- 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 laminating, drilling, and plating circuits, which allows for more flexibility in designing the circuit.

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

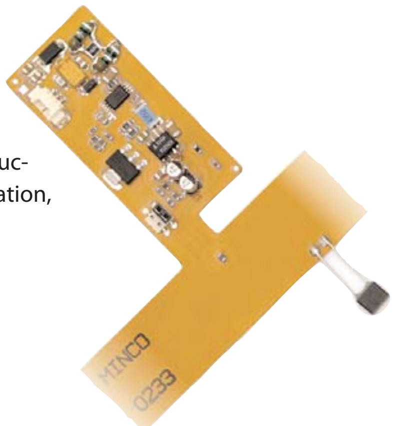


Flex-Coils have the same advantages that a flex circuit does. Wiring errors are reduced because the coil is oriented in the same spot every time, which provides repeatable signals. Flex-Coils are rugged and easy to assemble, and their design usually guarantees a reduced package size. A Flex-Coil can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

See Flex-Coils™ Technical Specification FC01, for more information on Flex-Coil capabilities, design considerations, and the information required for a quote or build.

Integrated solutions

Minco is a leading manufacturer of temperature sensors and Thermofoil™ flexible heaters. We have the unique ability to integrate these components and a flex circuit into a single package, drastically reducing assembly time and potential errors. Call Minco to discuss your application, or visit www.minco.com.



* Adhesiveless base material also available

Minco’s General Capabilities

Standard specifications

Physical properties

Circuit size/panel size:
10.5 × 22" max./12 × 24", 16.5 × 22" max./18 × 24"

Layers: 16 maximum.

Conductor width/space: 0.0025" minimum/0.0025" minimum

Hole diameter (plated): 0.003" minimum.

Aspect ratio (ratio of hole depth/hole diameter): 5:1 maximum.

Outline dimensions and hole-to-border tolerance:

SRD:	0.015" + 0.001" per inch distance
Hard tool:	0.007" + 0.001" per inch distance
CMD:	0.010" + 0.001" per inch distance

Cluster-to-cluster tolerance:
0.002" + 0.001" per inch distance

Bend radius (flexibility):

Single-layer:	6 × circuit thickness (minimum)
Double-layer:	12 × circuit thickness (minimum)
Multilayer:	24 × circuit thickness (minimum)

Circuit thickness is approximately 0.006" per layer. Sharper, permanent bends are common for bend-to-install applications — ask about factory forming.

Temperature: -65 to 150°C (-85 to 302°F).
Will withstand a 5-second solder immersion at 260°C (500°F) without blistering, delaminating, or discoloring.

Chemical resistance: No detrimental loss of physical properties when immersed for 15 minutes in acetone, methyl alcohol, toluene, or trichloroethylene.

Materials

Cover/substrate*: Polyimide film: 0.001", 0.002", 0.003", 0.005"; Photoimageable Coverlay (PIC); Epoxy glass (rigid-flex)

Conductor*:
Copper: 0.25 oz. (0.00035"; 9 micron), 0.33 oz. (0.00047"; 12 micron), 0.5 oz. (0.0007"; 18 micron), 1 oz. (0.0014"; 35 micron), 2 oz. (0.0028"; 71 micron), 3 oz. (0.0042"; 107 micron)
Cupro-nickel: 0.000625", 0.0009", 0.0013", 0.0019", 0.0023"
Nickel: 0.002", 0.005"

Adhesive*: Acrylic, flame retardant, epoxy, epoxy prepreg, polyimide prepreg, phenolic

Stiffener: Epoxy-glass (FR-4), polyimide-glass, polyimide, copper, aluminum.

* These are the standard materials Minco uses for manufacturing flex circuits. See page 19 or contact Minco for materials not listed or special considerations (e.g. implantable devices, extended temperature range, etc.)

Surface finish (plating)

Plating methods: Panel, selective, thruhole, blind via, buried via

Plating materials: Solder, hard gold, soft gold, tin, nickel, electroless nickel with immersion gold (ENIG), OSP

Electrical characteristics

Insulation resistance: 100 MΩ minimum @ 25°C (77°F), assuming 0.010" minimum conductor spacing.

Dielectric (typical): 1000 VRMS @ 60 Hz for 30 seconds, 1 mA maximum leakage current.

Shield layers: Solid or grid patterns; copper foil or screened conductive ink.

Inductor/Antenna coils: Specify inductance (10 mH to 30 mH, typical). Wire-wound coils may be integrated into the circuit. The cover encapsulates the coil, conductors, and coil connections. For details, see Flex-Coils™ Technical Specification FC01 (call 763-571-3121 or visit www.minco.com/support).

Heaters/Temperature sensors: Minco is a leading manufacturer of temperature sensors and Thermofoil™ flexible heaters. We have the unique ability to integrate these components and a flex circuit into a single package, drastically reducing assembly time and potential errors. Call Minco to discuss your application, or visit www.minco.com.

Value added assemblies**

Connectors:

Clincher:	0.100" minimum, center-to-center;
Micro series:	0.050" minimum, center-to-center;
Nano series:	0.025" minimum, center-to-center.

Optional epoxy potting is available.

Fingers:

Supported:	0.006" minimum, center-to-center;
Unsupported:	0.020" minimum, center-to-center.

In-line or right angle.

Pins:

Swaged/soldered:	0.085" min., center-to-center; 0.100" typical;
Brazed:	0.035" min., center-to-center.

Active Components:
Pick-and-place, Hand solder or braze
Surface mount, through hole, embedded

** See pages 20 – 22 for more information on incorporating these assemblies into your design.

Quality management

Minco is certified to ISO 9001: 2000 and AS9100 (aircraft) quality system requirements

Minco's General Capabilities

Testing

When specifying testing, consider your needs carefully. Over-specification can greatly increase circuit cost. 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 below 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
Dielectric withstanding	Up to 6000 V
Electrical continuity	1 Ω to 10 k Ω; suggest 5 Ω Stimulus: 0.01 V to 5.0 V
Ionic cleanliness	.5 microgram/square cm NACL equivalent
Insulation resistance	10 k Ω to 100 M Ω at 10 V to 250 VDC Suggest 100 M Ω at 100 VDC
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 1000×

Marking

Minco can meet your marking requirements.

Our legend marking system offers silkscreen-like printing using a durable white ink that meets IPC-TM-650 industry standards. This system allows us to incorporate date code and serial numbering, along with panel based marking, at the same time.

We can also offer traditional epoxy ink hand stamp or silkscreen printing if an alternate color or legacy specification is required.

Etched marking within the part is also an option. Stiffeners and covers may be marked with component mounting locations.

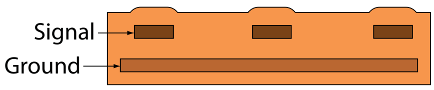
Controlling impedance and electrical noise

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 control impedance and reduce crosstalk. And with flex circuits, you can eliminate connectors and other transitions that contribute to signal attenuation.

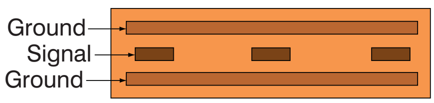
Minco can provide tight tolerances on line width, spacing, and distance to ground layers in order to meet your impedance requirements. Actual impedance will also depend on the circuit's shape after installation.

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

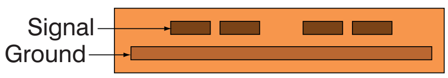
- Microstrip - a single ground plane beneath the signal lines.



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



- Edge coupled differential pairs – traces are adjacent to each other in the same plane with tightly controlled width and spacing, ground plane optional.



- 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 is more flexible than copper.

Minco’s General Capabilities

Conductor width nomograph

The nomograph on the facing page will help you determine the maximum allowable current capacity (in amperes) of a conductor. Reprinted from IPC-2221 (and MIL-STD-2118), the nomograph shows current for various conductor thicknesses, width, and temperature rises.

Using the nomograph

1. Locate the width of the conductor on the left side of the bottom chart.

2. Move right horizontally, until you intersect the line of the appropriate conductor thickness. Move down vertically to the bottom of the chart to determine the cross-sectional area of the conductor.

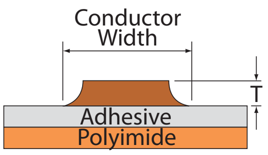
3. Move up vertically, until you intersect the line of the appropriate allowable temperature rise. This is the increase in temperature of the current-carrying conductor. Conductor temperature should not exceed 105°C. For example, if the ambient temperature might reach 80°C, the temperature rise above ambient of the conductor should be less than 25°C (105°C - 80°C). In this case use the 20°C curve.

4. Move left horizontally, to the left side if the chart to determine the maximum allowable 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") design the conductors to be 0.0140" or wider. In tight situations Minco is successful in achieving 2.5:1 ratio conductor widths.



Ideal: Conductor width greater than 5x conductor thickness.

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 lateral 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 determine the temperature rise.

6. Current ratings are for still air environments. Forced air cooling will increase the maximum allowable current. Operating circuits in a vacuum will greatly decrease the maximum allowable current.

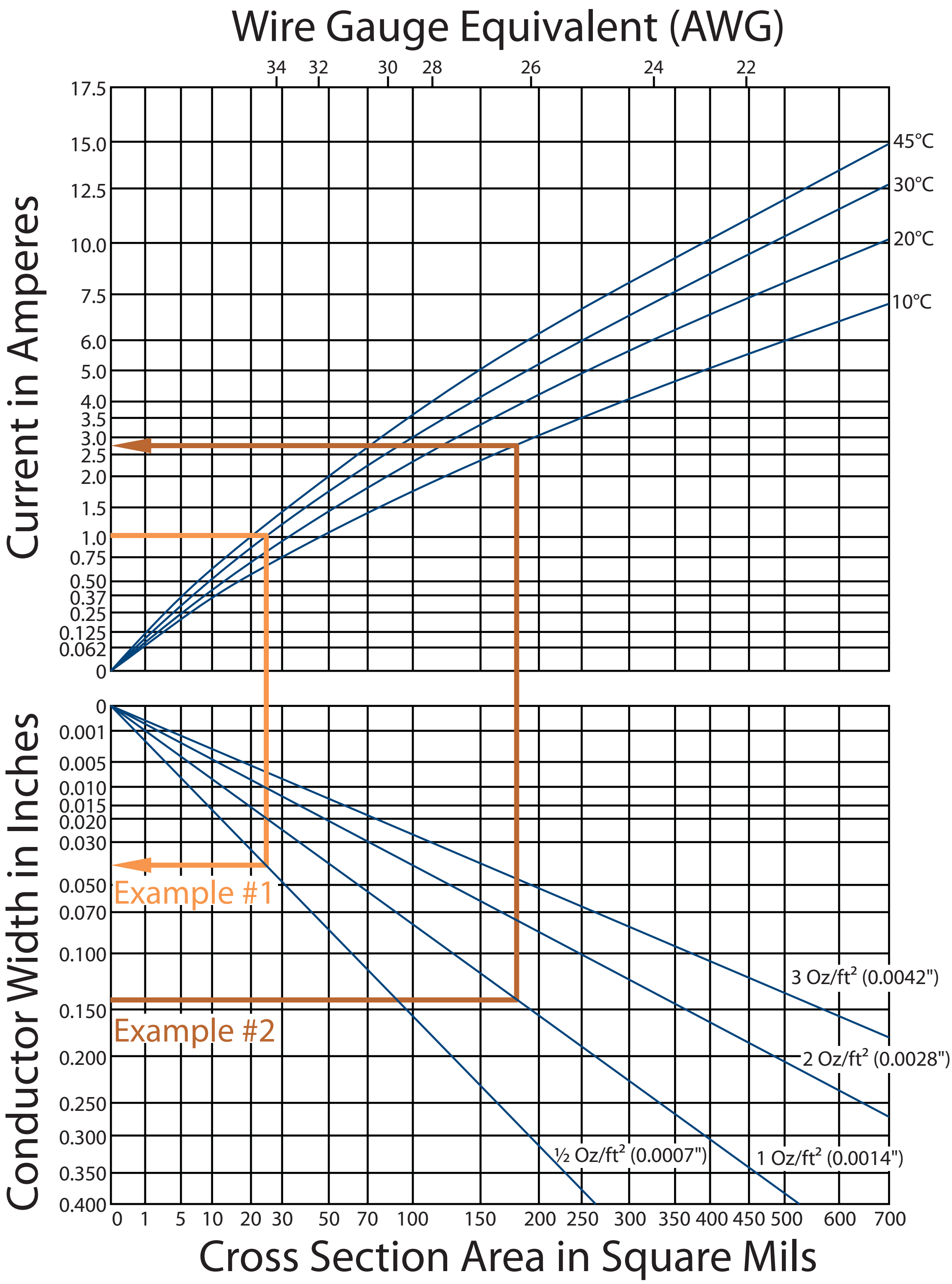
Contact Minco for assistance in cases where the nomograph does not apply. Also contact us if you have difficulty designing sufficient current capacity into the space available. We can suggest ideas to increase current capacity.

Minco's General Capabilities

Conductor width nomograph

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 0.140" wide conductor etched from 1 oz. copper (0.0014") will produce a temperature rise of 10°C at 2.7 amps.



Standard Design Recommendations

Design differences and special considerations

Define circuit parameters by application

It may be helpful to use a paper template to represent the circuit. Experiment with bending and forming the template to optimize shape and fit. When designing the final shape, consider how the circuits will lay out on a processing panel ("nesting"). The greater the number of circuits per panel, the lower the cost.

Another consideration concerns rigid-flex. While Minco is capable of building a traditional rigid-flex board for you it may not be your best choice. Multilayer or stiffened flex boards may be able to meet your requirements for component and board mounting at reduced cost.

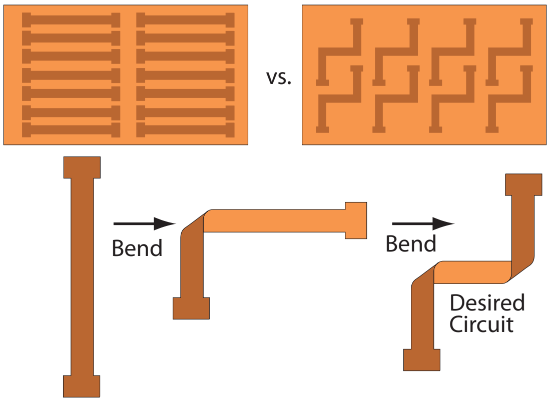
Flex circuit vs. hardboards

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

- Consider how circuits will be "nested" on a panel.



- 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 absolutely necessary.
- If circuits have only a few layers, using stiffeners can be far less expensive than a rigid-flex circuit.

Special considerations for rigid-flex

- Rigid-flex is the ideal solution for applications with multiple rigid PCBs having SMT components on both sides and requiring interconnects between the rigid PCBs.
- 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 less expensive alternative to 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.
- Observe aspect ratio (hole depth/hole diameter) limits (see Capabilities on page 10).
- Minco builds circuits up to 16 layers, but costs increase significantly above 10 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 within a single rigid area.
- Minimum inside corner radius of 0.031" is standard, but smaller radii are available.
- Unbonded layers can increase flexibility in multilayer flex circuits, but this option is more expensive. Specify unbonded layers only in areas of the circuit that will bend.
- Minco can provide an epoxy 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.

Standard Design Recommendations

How to improve flexibility and bend radius

Single-layer circuits are the best choice for dynamic (flex-in-use) applications. Two or more layer circuits are best suited to static applications, with flexes only during installation.

Several problems can arise when a circuit is bent sharply. Compression can cause wrinkles in the cover coat on the inside of the bend. Stretching can result in tears in the cover material and/or broken conductors on the outside of the bend.

Start the mechanical design by establishing the bend radius. If the radius is at least ten times the thickness of the material, there is a good chance that the circuit will function reliably.

The minimum allowable bend radius depends on a number of factors, and is best defined by IPC-2223. Overall circuit thickness is slightly less than the sum of the individual insulator, adhesive and foil layer thicknesses.

Bend radius (flexibility):

Single-layer:	6 × circuit thickness (minimum)
Double-layer:	12 × circuit thickness (minimum)
Multilayer:	24 × circuit thickness (minimum)

Circuit thickness is approximately 0.006" per layer.

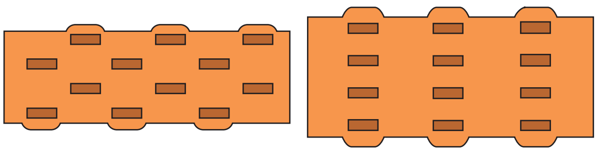
Incorporate these features into multilayer and reduced bend ratio designs to increase reliability

1. Reduce overall thickness in the flex area

- Reduce the base copper weight (and the corresponding adhesive thicknesses) or reduce the dielectric thickness.
- Use adhesiveless base materials. Adhesiveless materials will usually reduce the starting thickness of each substrate by .001"-.002" when compared to adhesive based substrates.
- Eliminate copper plating on the conductors in the flexing area by utilizing selective (pads-only) plating or adding outer pads-only layers to the circuit.

2. Make the circuit robust to withstand flexing

- Balance the conductor weights and material thicknesses on each side of the neutral bend axis.
- Conductors should be staggered from layer to layer and not stacked on top of each other to increase flexibility.



GOOD VS. BAD

- Conductors should always be routed through bend areas as close to perpendicular as possible.
- Conductor thickness and width should remain constant in bend areas.
- Plated through holes should be kept out of the bend areas.
- If the circuit will be bent within 1" of termination pads, fillets should be placed at each conductor/pad interface. Stresses from a bend are not isolated to the immediate bend area and residual stresses can radiate out from the bend. See page 17 for more information on fillets.
- If shields and/or ground planes are required on the circuit, use a crosshatched pattern rather than solid copper. Another shielding option is a screened-on conductive coating such as silver epoxy, which is much more flexible than copper.
- Incorporate tear stops or reliefs for slits in the circuit. The end of the slit represents a vulnerable point for a tear to start and to propagate.

- Avoid any discontinuities in the cover coat or substrate near a bend.
- The circuit outline should be designed so there are no twists in the finished assembly. Any burr or irregularity from the blanking operation could potentially propagate into a tear.
- Consider factory forming. Reliable bend radii tighter than 10:1 are possible if the circuit is formed using specialized tooling and will only be flexed one time.
- If bend reliability is still a concern, consider "unbonding" the flexible substrates from each other. Since each of the substrates in the unbonded area has a much lower thickness than the total circuit, they are able to bend tighter than if they were fully bonded.

For a more in-depth look at this subject please see Application Aid #31 - Designing for Flexibility and Reliability at www.minco.com

Standard Design Recommendations

Holes

General requirements for pads, access holes and annular rings

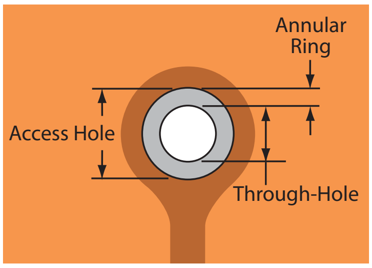
Feature	Single-layer	Double-layer	Multilayer
Pad*	0.060" + t	0.030" + t	Outer pad: 0.030" + t Inner pad: 0.025" + t
Major access hole*	0.050" + t	0.030" + t	0.030" + t
Minor access hole‡	0.015" + t	0.015" + t	0.015" + t

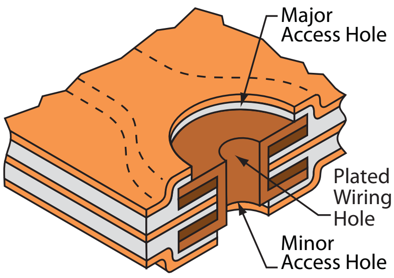
t = nominal through-hole diameter.

* 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 all external layers, and 0.002" minimum for multilayer innerlayers.

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

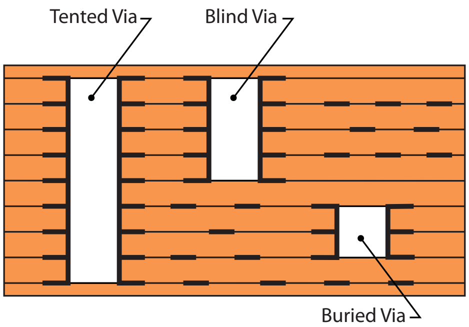
For more information on access holes (major and minor) annular ring, pads, and thruholes, see the Glossary on pages 27 – 29.





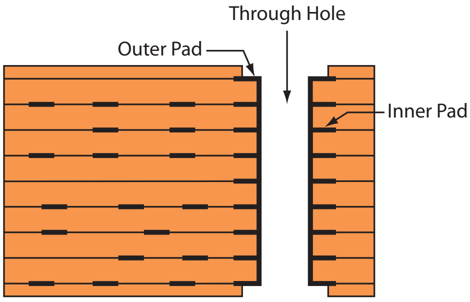
Vias

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.



Thruhole or Through-hole

Plated through holes (PTH) connect together the top, bottom, and any required internal conductor layers. PTHs are drilled oversize to accommodate the thickness of the copper plating that will cover the entire barrel of the hole as well as the surface of the outer pads. Exterior pads may be plated along with the entire foil surface or selectively plated just at the PTH site.



Stiffener holes

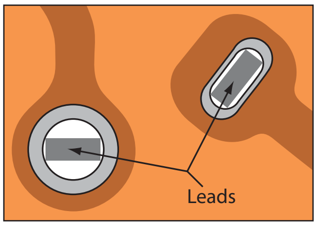
Stiffener holes should be a minimum of 0.015" in diameter larger than the access hole. It is better if the access hole underneath the stiffener hole is a minor access hole in order to increase the stiffener web between holes and to prevent potential solder wicking between the stiffener and the circuit. 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.

Standard Design Recommendations

Holes

Wiring holes

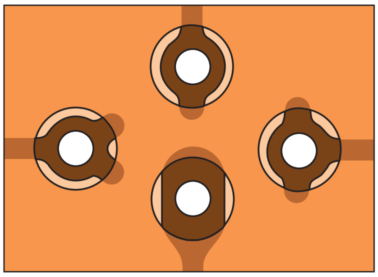
Minco can drill through-holes as small as 0.0039". A 0.020" through-hole size is typical. Standard finished hole tolerance is ±0.003". For all circuits, the finished through-hole size should be 0.003" to 0.010" larger in diameter than the component lead. This depends on the number of leads per component,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

Pads

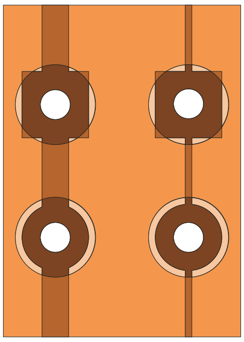
Whenever possible, design pads larger than the access holes. If space is critial, use hold-down tabs. Hold-down tabs are especially important for single-layer circuits, because a single-layer circuit does not have the added strength of plated through-holes. A variety of hold-down tab designs are available.



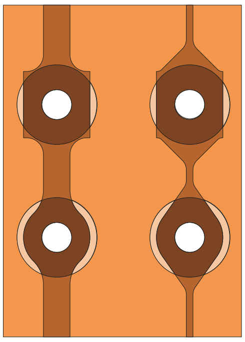
Pad fillets

Pad fIllets improve etched yield and material strength. Fillets are appropriate when the pad diameter is greater than the connecting strand width. Acute angles at the interface between conductors and pads are to be avoided by using fillets to minimize the concentration of stress at the interface.

Before fillets

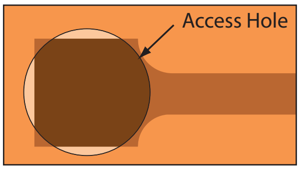


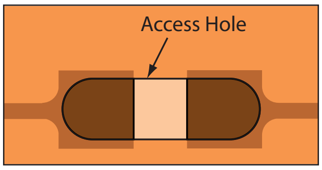
After fillets



Surface mount access

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





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

Soldering tips

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

Thermal relief

- When hand soldering pins in dense clusters, try not to solder adjacent pins one after another. Move around to avoid local overheating.
- Minco can solder connectors or components (SMT or Thru-hole) as an added service.
- Minco can supply circuits in panel form for easier component assembly.

Standard Design Recommendations

Tolerances

You are not limited to the tolerances listed in this section. Tighter tolerances are achievable, but often at a higher cost. Accordingly, more relaxed tolerances will typically cost less. Even with relaxed tolerances, a flex circuit will have a uniformity that is impossible to attain with conventional wiring. The flexibility of materials constructing a flex circuit allow it to be more compliant than rigid circuits, so it is not always necessary to specify tight tolerances across all dimensions.

Trimming

Each trimming method has advantages and disadvantages. Routing and laser trimming provide hard tooling (punch and die) tolerances for small quantities of circuits. Laser trimming is also capable of complex cutouts not feasible with other methods. Steel rule dies (SRD) are best for intermediate quantities and tolerances. Chemical milled dies (CMD) offer tighter tolerances than SRDs for an incremental increase in cost. Hard tooling (punch and die) is recommended for tight tolerances, complex circuits, and/or high quantity. For more specific information on SRD, punch-and-dies and CMD, see the Glossary at the end of this guide.

Circuit dimension in inches†	Outline dimensions (profile tolerance)			Hole-to-border dimensions			Cluster to cluster§
	SRD	CMD	Punch and die /laser	SRD	CMD	Punch and die /laser	
1	±0.015	±0.010	±0.003	±0.015	±0.010	±0.007	±0.003
5	±0.020	±0.015	±0.007	±0.020	±0.015	±0.012	±0.007
10	±0.025	±0.020	±0.012	±0.025	±0.020	±0.017	±0.012
15	±0.030	±0.025	±0.017	±0.030	±0.025	±0.022	±0.017
20	±0.035	±0.030	±0.022	±0.035	±0.030	±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 definition of profile tolerance.

Solder thickness

Minco follows IPC-6013 requirements of coverage and solderability for solder coatings.

Conductor width, thickness, and spacing

See the nomograph on pages 12 – 13 for calculating the necessary conductor width and spacing. Minco can provide a 0.004" minimum conductor width/spacing (0.0025" 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.	±0.001"	±0.001"
1 oz.	±0.002"	±0.001"
2 oz.	±0.003"	±0.002"
3 oz.	±0.004"	±0.003"

Standard Design Recommendations

Materials

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

Material function	Material type	Sizes/thickness available
Flexible insulator	**Kapton*** and other polyimide films †	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	Variety of thicknesses between 0.003" and 0.125"
Conductor	**Copper**	1/4 oz., 1/3 oz., 1 oz., 2 oz., 3 oz., 5 oz., 7oz., 10 oz.
	Different forms of copper	Half-hard, rolled-annealed, electro-deposited
	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"
	Pressure-sensitive adhesive (PSA)	0.001", 0.002", 0.005"
	Preimpregnated material: FR-4, polyimide	0.002", 0.008"
Stiffener	Copper, Aluminum, and other metals	Variety of thicknesses available
	Polyimide glass	See "Rigid substrate/Polyimide glass" above
	FR-4 **Polyimide**	Variety of thicknesses between 0.005" and 0.125" 0.0005", 0.001", 0.002", 0.003", 0.005"

* Kapton is a registered trademark of DuPont for polyimide. Dielectric strength of plain Kapton film is 3500-7000 volts/mil depending upon material thickness. Kapton/modified acrylic has a dielectric strength of 3500 volts/mil and 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.

† Other polyimide films are available for special applications.

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

§ In general, Minco recommends 0.001" of adhesive on the cover material per 1 oz. of copper (including plated copper). There may be special circumstances where more is required. Consult Minco for details.

Value Added Design Options

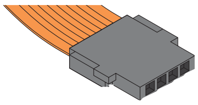
Terminations

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

Connectors

Connectors are usually customer selected, but Minco can recommend certain types of connectors to meet specific application requirements. Connectors can be attached to flex circuits by hand soldering, wave soldering, crimping, or simple insertion with zero insertion force (ZIF) models. Connectors can be potted after attachment or conformally coated for protection and insulation with epoxy, polyurethane, or RTV.

One good option for many low cost applications is the Clincher™ insulation displacement connector.



High density connectors, with 0.050" or 0.025" center-to-center terminals, are available from Omnetics Connector Corporation in several forms, including high temperature and MIL-spec options.

Vertical or horizontal mounting to fit your designs.

Up to 44 Beryllium Copper pins in less than half the space of an 0.050" connector!

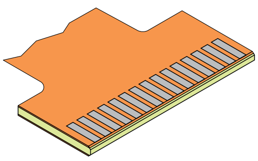
Mil-spec connectors with temperature ratings to 200°C.

Round form connectors for surface mount installation.

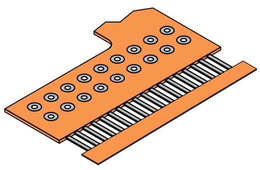
Connector type	Centerline distance
Clincher connector	0.100" min.
Micro series pin center-to-center	0.050" min.
Nano series pin center-to-center	0.025" min.

Fingers

Fingers can be supported or unsupported. Supported fingers are ideal for ZIF connectors mounted on rigid boards. Unsupported fingers can be hot bar soldered to hard circuit boards.



Supported fingers



Unsupported fingers

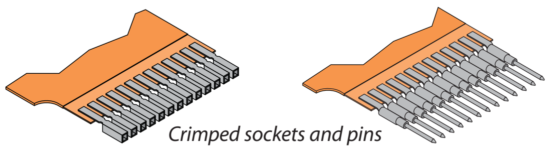
Finger type	Centerline distance
Supported	0.006" min.
Unsupported	0.020" min.

Pins

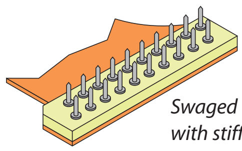
- Socket pins are pressed in place and then soldered. Pins can be swaged to the circuit and soldered after the swaging procedure, or pins can be swaged to an FR-4 stiffener and then soldered. Swaged/soldered pins are moderately priced and have good mechanical strength.
- End pins that are in line with 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).

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

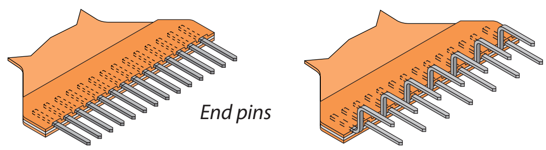
Pin type	Centerline distance
Swaged	0.100" typical, 0.085" min.
Brazed	0.100" typical, 0.035" min.



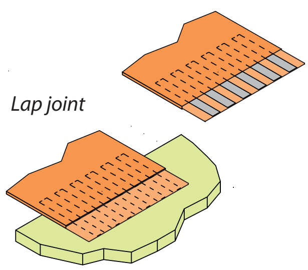
Crimped sockets and pins



Swaged or nailhead pins with stiffener



End pins



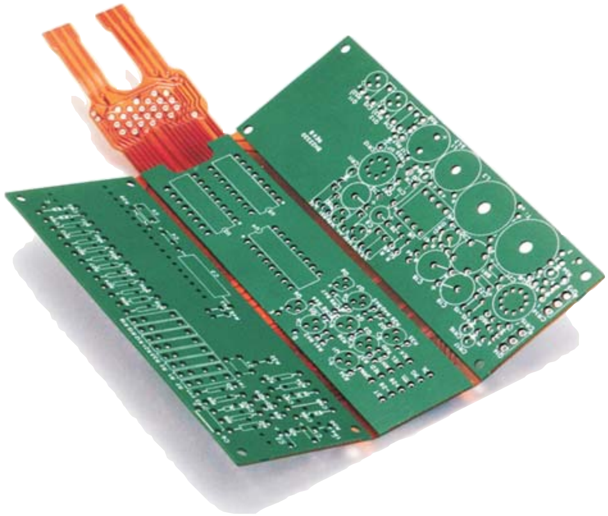
Lap joint

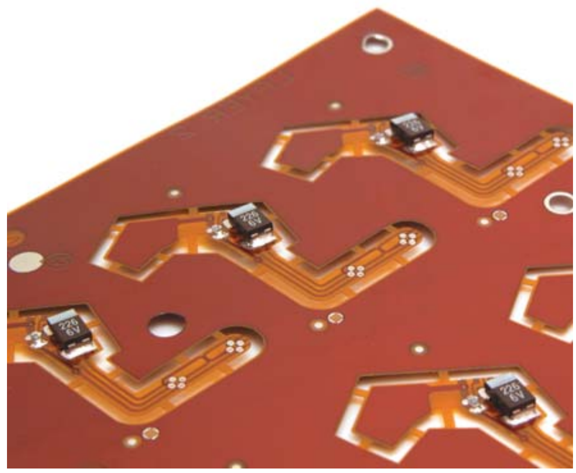
Value Added Design Options

Stiffeners

Benefits of using a stiffener

- Stiffeners are an inexpensive option for rigidizing pin areas, surface mount areas, or hole patterns for component mounting (provided SMT components are on one side only). 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 be utilized to force a bend line in selected areas. Minco can provide epoxy fillets for the edges of the FR-4 stiffener, where flexing occurs.
- Stiffeners reinforce solder joints and increase abrasion resistance.
- Circuits may be attached to a stiffener pallet (multiple parts) to provide easier handling for automated pick-and-place and component soldering. Circuits can be held together for processing on the pallet, then singulated (clipped free) after wave soldering and circuit testing.
- Stiffeners can be silk-screened with component mounting locations for rapid assembly.
- Stiffeners are commonly FR-4 or polyimide material. They are usually applied with modified acrylic adhesive.
- Standard FR-4 material thicknesses range from 0.003" to 0.125". 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. The polyimide stiffener lay-up procedure is performed with alignment pins, therefore, registration is better. The polyimide stiffeners are trimmed with the cover on the final blanking procedure, which guarantees perfect outside alignment.
- When using multiple stiffeners, maintaining the same stiffener thickness consistent throughout the entire construction will help keep costs under control.





Value Added Design Options

Forming

Flexible materials don't guarantee that the circuit will function reliably when bent or flexed. There are many factors that contribute to the reliability of a printed flex circuit and all of these factors must be taken into account during the design process to ensure that the finished circuit will function reliably.

When designing a flex circuit, the designer must factor in all of the parameters that will have an impact on the circuit's ability to bend or flex in the specific application. These include, but are not limited to: whether the application is static or dynamic, bend radii, dielectric thicknesses and type, foil weight, copper plating, overall circuit thickness, number of layers, and number of flexures.

The tighter a bend radius becomes, the higher the probability of failure during flexing. Keeping the overall thickness of the flex circuit in a bend area to its minimum will increase reliability. The ratio of bend radius to thickness is one indicator of whether the design is going to be reliable or have a high probability of failure. If the bend radius is at least ten times the thickness of the material, there is a good chance that the circuit will function reliably. If the calculated bend radius falls below ten to one, the design may be questionable. Formulas for calculating the minimum allowable bend radius for several circuit types can be found in IPC-2223.

It is possible to design for much tighter bend radii, in a bend-to-install application which retains the formed shape of the circuit. The circuit must be designed to withstand stretching along the outer bend and compression of materials on the inner bend. Stretching can tear covers or crack conductors while compression causes foil and cover wrinkling that can also lead to tears. These problems become more of a concern in applications that require the circuit to be bent beyond a 90-degree angle. As the bend angle increases beyond 90 degrees, the damaging effects of stretching and compressing increase dramatically. Any time that a reduced radii bend beyond 90 degrees is incorporated into a circuit design, the circuit should be bent one time only. On bends over 90 degrees, it is also advisable to constrain the circuit in the formed condition to keep it from relaxing or being inadvertently reopened.

The ideal circuit design would have no copper plating on the conductors in the forming or flexing area. Electrolytic copper has much lower ductility than that of rolled annealed copper, making it much more susceptible to fracturing when it is bent or flexed. Other types of plating, such as gold and/or nickel, should be avoided in the flexing area for the same reasons.

Copper plating on the flexing conductors may be eliminated by using pads-only plating or designing with pads-only layers on the external surfaces.

Minco can factory form some flex circuits with radii all the way down to a "crease", dependent upon board design, to improve installation precision and repeatability within our customer's assembly process.

When designing a part for forming, it is important to avoid mechanical stressors in the bend zone. Stressors include pads, holes, components, and sharply angled conductors. These reflect the most common features problematic to forming.

Forming imparts stress into circuits. Some designs will be better suited to this forming process than others. Minco encourages customers to discuss the intended usage of parts they want formed with our engineers to help determine suitability.

Forms are likely to relax slightly over time. Tightly toleranced forms are not normally held by flex circuits. Minco recommends specifications to read "reference only" or to describe the parts as "capable of attaining 'x' dimensions" for both angular and linear aspects of forming.

Formed parts are less capable of withstanding temperature variations than their un-formed counterparts. This is particularly true where the bend zones are exposed to solder reflow or high temperature sterilization procedures. To work around these limitations Minco can suggest optional designs or assembly processes. Please contact us to discuss your needs.

Population

Minco will also assemble hardware and electronic components onto your flex and rigid-flex circuits.

Numerous connectors are available for flex termination, ranging from crimp connectors to nano-size SMT connectors or discrete pins.

Heat-sinks, metal stiffeners, and plastic mounting frames may also be laminated, heat staked or glued to flex circuits.

Flex-coil designs are an example of an embedded component supplied within flex, multilayer, and rigid-flex circuits.

The most common electronic components are typically surface mounted to the flex circuit using automated pick-and-place equipment. Design considerations for populating flex circuits differ slightly from rigid boards. Simple flex circuits usually need to be stiffened for surface mount components. Our design engineers will make recommendations for necessary design factors to meet your needs.

Between our in-house capabilities and our vendor network we're able to offer our customers bare parts, palletized and unpopulated, partially populated or fully populated.

Request a Quote

Providing information for a quote

Information required for a ballpark quote

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

Information required for a firm quote

- Drawing (Gerber data preferred)
- Quantity required
- Complete physical shape
- Materials: conductors, insulators, stiffeners, 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 and/or packaging requirements
- Testing requirements: type, percent to be tested, and frequency. Is IPC-6013 testing required?
- Additional components that Minco is expected to supply/assemble (inform Minco of preferred suppliers if the components are unique)

Information required for manufacturing

All the information that is required for a firm quote, plus:

- Dimensional drawing including notes and requirements
- Artwork for conductors in digital format, coupons, and screened marking, if applicable (unless Minco is to generate)

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

Specify testing requirements

Testing is labor intensive and will directly impact costs and delivery schedules. Some tests are automated, others are manual. Test frequency must be considered as well as the destructive nature of some tests. All tests are documented and certificates of conformance are routine for Minco.

Test requirements must be documented to avoid any confusion. Your sales and design engineer contacts can discuss these needs with you.

Physical testing options include, but are not limited to

- Dimensional measurements
- Ionic contamination
- Thermal shock
- Solderability

Electrical testing options include, but are not limited to

- Resistance
- Insulation resistance (IR)
- Continuity
- Inductance
- Capacitance
- Sencore Ringer
- Dielectric, net-to-net
- Dielectric, high potential test of exterior insulation
- Impedance

Note: Standard electrical testing includes continuity (5 Ω) and insulation resistance (40 mΩ @ 150 VDC).

Request a Quote

Artwork checklist

Minco strongly encourages customers to use the artwork checklist 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)

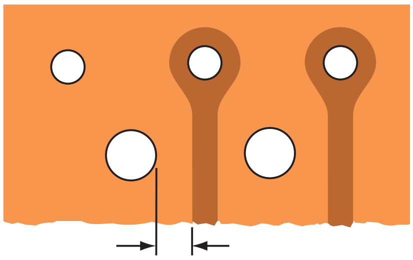
- Are all features of letters and symbols at least 0.010" wide?
- Are letters and symbols clear of conductors and borders?

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?



Conductor width

Is artwork conductor width:

- 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 15 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 .east 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?

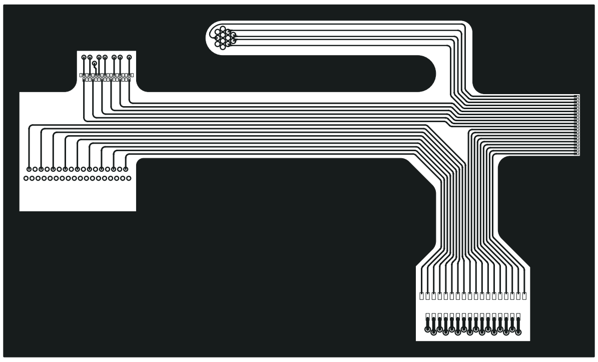
Request a Quote

Designing CAD artworks

This section provides the information necessary for designing CAD artworks 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 18.
- Location and size of conductor pads. See the table on page 16.
- Minimum conductor widths, minimum spaces between conductors and conductor thickness. These will depend on current carrying capacity, impedance, dielectric, and the flexibility requirements of the circuit. See the nomograph on pages 12 – 13.
- Net list
- Conductor paths can be captured from a net list or schematic when required. A design charge may apply.
- 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. If you cannot furnish CAD data, we can digitize a physical artwork at additional cost.



CAD-generated artwork

CAD data guidelines

Minco can accept CAD data in the following forms

- Gerber RS-274X format (embedded aperture) photoplotter code is preferred!
- RS-274D Gerber data with separate, detailed aperture list can be used, but is not preferred.
- AutoCAD DXF (2D)
- AutoCAD DWG (2D)

Other formats may be acceptable — contact Minco for details.

Transferring data to Minco

You can transfer your media to Minco 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 (3.5"), Zip disk, or CD: IBM-PC format

Guidelines for all formats

- Single entity draws for conductors are required.
- Single pad flashes are required.
- Minimize the entities used to create conductor to pad transitions ("fillets").

Guidelines for DXF

- Place artwork data, part outline, hole centers, soldermask, coverlay, screen 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 supporting CAD system layer with proper line width conductors and pads.

Guidelines for Gerber

When sending your photoplotter code, please include:

- The format of the data
- An aperture wheel listing (when required)
- A list of layers with descriptions
- The number of files supplied

E2E — Engineer to Engineer

Early engineering involvement

Great flex designs are achieved when the designer understands that flexible circuits are as much a mechanical component as they are electrical. Engineering consultation can be invaluable early in the design process. Minco wants to make your access to engineering tools and expertise as convenient as possible. That is why we have developed an E2E (engineer to engineer) community online at www.minco.com/e2e.

Circuit design assistance is available with online resources, including an "Ask the Experts" link to request help from Minco experts and an "E2E Community Discussion Forum" where questions may be posted to the engineering community beyond Minco.

Minco engineer review

Our engineers will review your quote or order documentation and data to determine if changes are needed for manufacturability. If needed, we will discuss these issues with you to our mutual agreement before construction begins.

Design services

Start to finish or problem specific, design engineers are available to assist our customers. Your sales engineer will put you in contact with the design engineer most able to help you with your specific design.

Ordering and Delivery

Place your order with Minco

When you receive your Minco quote you will see any findings listed. Please review your Minco quote. Placement of an order includes acceptance of all terms, conditions, exceptions and substitutions. We are more than happy to discuss any questions you have regarding your Minco quote.

When placing your order, include final design data, drawings, and procurement documentation reflecting your expectations and Minco's DFM changes. Clarity is key to Minco's relationships with customers.

All quality clauses and expectations should be detailed during the RFQ/Quote stage.

Send your purchase order and all other required information to your sales engineer to begin our construction process.

Delivery information

Our sales engineer will provide estimated ship dates that will vary due to your design, our work load, and which group builds your parts.

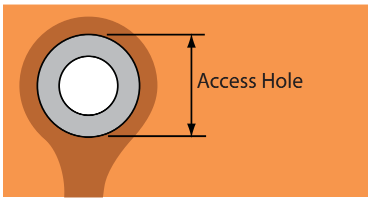
Production models will follow our best practices and applicable lead times. Delivery times and quality are predictable for all standard designs.

Prototype/quick turn models are generally offered on a "best effort" basis with regard to meeting all print requirements. We can also provide many designs completely "per print." Quickturn offers fast proof of concept constructions that may uncover improvements as well as road-blocks. Either way, you will see your product more quickly with our quick-turn group than through our production group.

Glossary

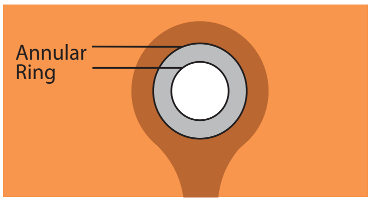
Access hole

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 circuits through-holes.



Artwork

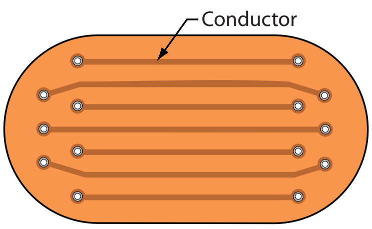
The original pattern of conductor strands for a flexible circuit.

Chemically milled die (CMD)

A tool used in a punch press with blades formed by a chemical milling process, and mounted on a 1" think aluminum base..

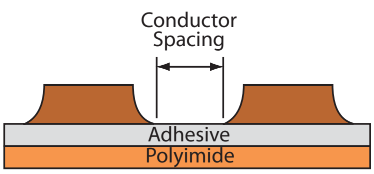
Conductor

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



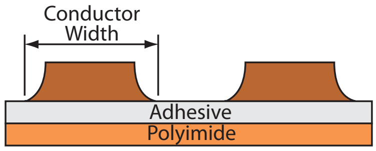
Conductor spacing

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 laminated to an etched element. Covers can be located on the inner or outer layers. 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

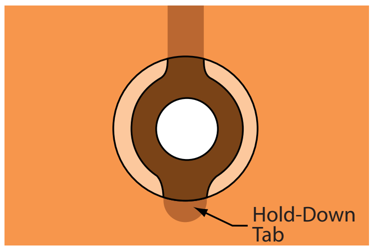
Flexible printed circuits made from etched foil conductor strands. The conductor strands are laminated between insulating layers. Flex circuits can vary in complexity from the simplest single-layer circuit to a complex multilayer.

Flex-Coil™

Flex circuits with internal or attached wire-wound coils.

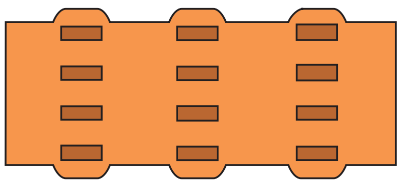
Hold-down tabs

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 fracture conductor strands if the conductor strands are layered directly over each other, instead of being staggered from layer to layer.



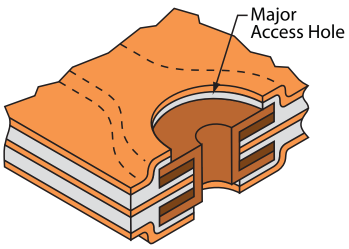
Glossary

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.

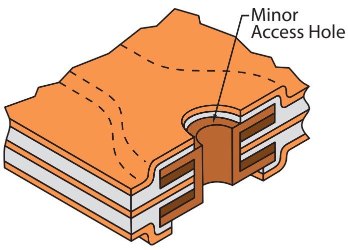
Major access hole

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.



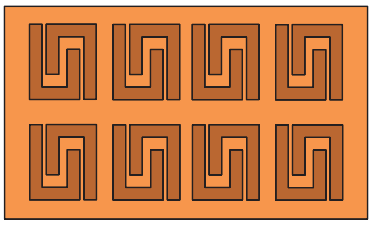
Minor access hole

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



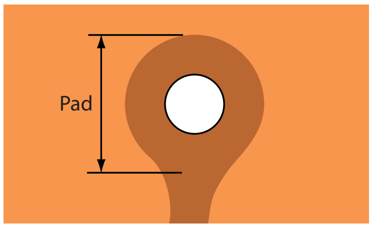
Nesting

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



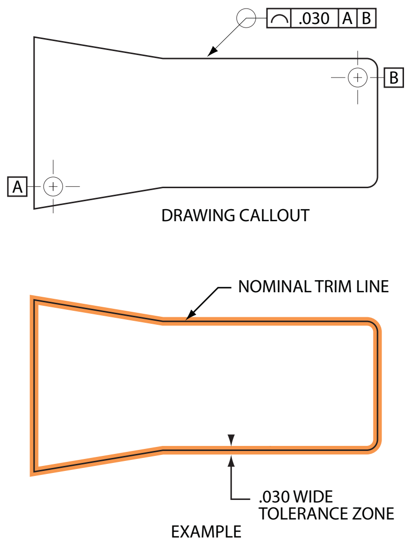
Pad

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."



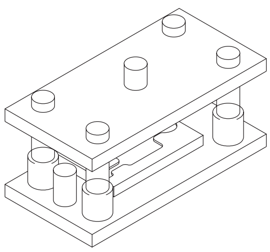
Profile tolerance

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.



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.



Glossary

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

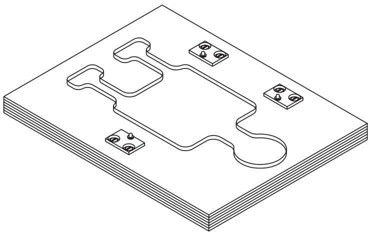
A method of plating flex circuits so that only the circuit's through-holes and surrounding pads are plated. This greatly adds to a circuit's flexibility.

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 (SRD)

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



Stiffener

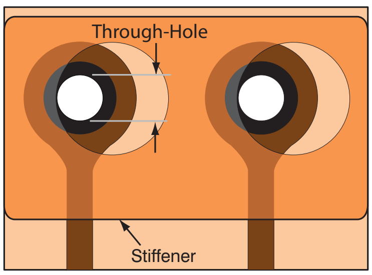
Flexible or rigid pieces of material (usually Kapton or FR-4) added to flex circuits to reinforce them for component mounting. There are no conductors on stiffeners, as compared to rigid-flex circuits.

Substrate

A layer of insulator 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

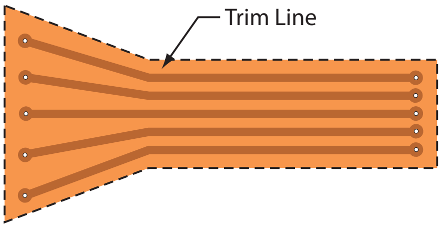
Copper, Kapton, or Teflon guards that are located in the inner corners of Kapton-insulated flex circuits in order to prevent propagation of tears.

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.

Worldwide Headquarters
7300 Commerce Lane
Minneapolis, MN 55432 USA
Tel: 1.763.571.3121
Fax: 1.763.571.0927
sales@minco.com
www.minco.com

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

Asia Pacific Headquarters
20 Science Park Road
#02-31 Teletech Park
Singapore Science Park II
Singapore, 117674
Tel: (65) 6511 3388
Fax: (65) 6511 3399

ISO 9001:2000 / AS9100B
FC302021406R0
© Minco 2006

MINCO
A critical component of your success™

Flex Circuits
Thermofoil™ Heaters
Sensors
Instruments