

Flexible Printed Circuits:

A Design Primer

By Chris Keirstead

Flexible circuits consist of conductive strips in a sandwich of insulating or dielectric material. They resist moisture and contamination and are insulated from external shorts, with holes or contact surfaces for interconnection.

Understanding a package's electrical requirements, and not over designing, permits taking full advantage of flexible circuit's potential compared to conventional wiring. Among the benefits are weight and bulk reduction, shorter assembly time and fewer errors, reproducible electrical characteristics, custom shielding, improved reliability, ability to interconnect moving parts, and an engineered appearance.

The mechanical and electrical characteristics of flexible circuits determine what type should be used. Total installed cost, including inspection, interconnection, fixturing and testing, must be weighed against the advantages of each:

- Single-sided flexible circuits have one layer of copper on dielectric.
- Double-sided flexible circuits have copper on both sides.
- Sculptured circuits® are single or double-sided circuits created from thick copper that allows connections such as fingers and pads to be relatively thick and very rigid extensions of the flexible conductors.
- Multi-layer flexible circuits have several layers that are registered to each other and are separated by a cured layer of adhesive during lamination, with interlayer connections via plated-thru holes.
- Multi-layer rigid-flex is the combination of flex within and extending from rigid sections of the PC-Board.

There two general areas of application that may take the designer down different paths. The first is a circuit that will be bent or formed into a rigid package during assembly. The second is where the circuit will be flexed continuously or intermittently during operation.

Insulating Materials

The choice of material must take into consideration conditions of operation and manufacture, electrical characteristics, mechanical requirements, and cost. Some materials have qualities that may make them more suitable for certain applications, but they may not be universally useful.

Kapton®, perhaps the most widely used material, is dimensionally stable and is compatible with all pc board processes. It does not support combustion, is flexible and adhereable, has a high tensile strength and about the same coefficient of expansion as copper. It does, however, absorb significant

amounts of moisture.

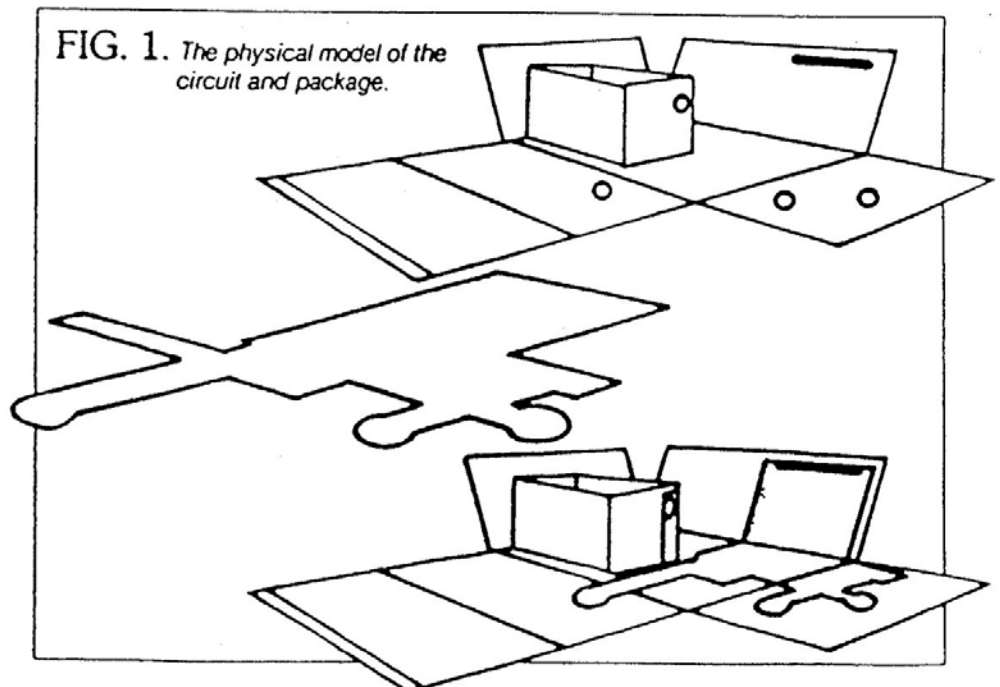
Mylar®, is inexpensive, flexible, and adhereable and dimensionally stable, but is sensitive to some chemicals used in pc board processing and to soldering heat.

Teflon FEP®, is an excellent dielectric, does not absorb water, and is flexible and adhereable, but is not dimensionally stable and is expensive.

Nomex®, is dimensionally stable, flexible and adhereable, has high tensile strength, and resists soldering temperatures. However, Nomex® reacts to processing chemicals and absorbs water.

Design Procedures

If the actual unit to use the circuit is unavailable, construct a full-size model from cardboard, sheet metal or wood. Open the model to lay flat and place or sketch components in their required positions. Make a paper doll out of the cir-



cuit shape that will provide all connections and conductor runs in a size that fits the packaging constraints. (See Figure 1)

Prepare a single entry wire list showing each conductor in a continuous run to every termination point.

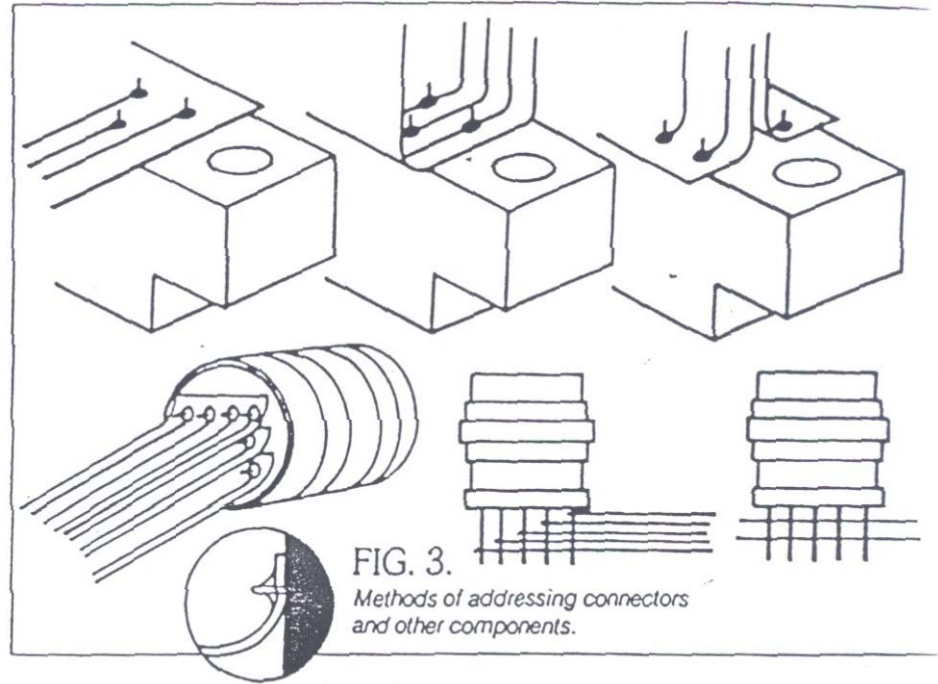
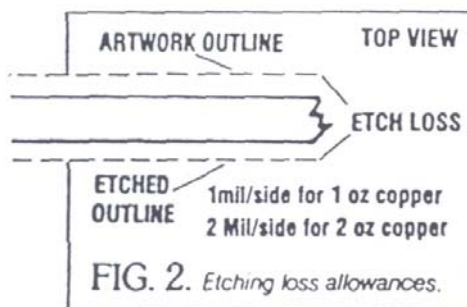
Establish current carrying capacity and resistivity requirements, and determine copper conductor width and thickness required for current capacity and voltage drops.

Next, condense the wiring list and determine the best layout of conductor runs to minimize the required circuit layers and the circuit's mechanical area. Separate analog and digital signals to minimize cross-talk by placing them on opposite sides of the circuit or on different layers. Shielding may be necessary.

With these preliminary steps completed, begin generation of specific artwork for circuit production.

Spacing between conductors is determined by the voltage between conductors (dc or ac peak volts) and ranges from a minimum of 0.005 in. for 15v or less to 0.020 in. for up to 500v. A general rule of thumb is 0.00012 in./volt. Larger than minimum spacing is preferred. Routing of conductors should be as direct as possible with no unessential overlaps that create the need for additional layers. Prepare a detailed drawing with all dimensions specified and all components positioned. Specify terminal areas without assigning pin addresses when possible. Keep edge distance as large as possible; 0.050 inches is recommended.

Prepare a dimensionally accurate layout. Work oversize (i.e. 4:1) for clarity. With drafting accuracy of 0.010 in., the 1:1 should be 0.003 in. or less. Leave radii for bends or folds equal to 10 to 12 times the total circuit thickness. Use 10:1 layouts of close tolerance areas (i.e. conductor pin clusters) and re-



duce to layout scale.

Make multiple copies of the master layout (1:1 scale) for subsequent fine-tuning of the layout.

The metal most commonly used in flexible circuits is rolled and annealed copper with greater than 99% purity. Other materials may be used for special applications. Copper thickness is usually specified by weight per square foot, but may also be specified in linear measurements. For example: 1 oz./ft² = 0.0014 inches and 2 oz./ft² = 0.0028 inches.

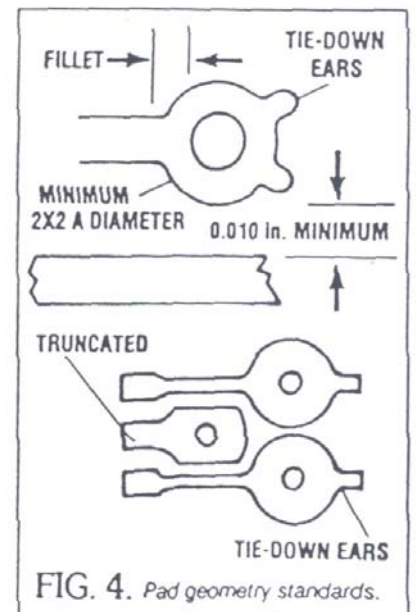
The resistance and current carrying capacity of conductors are interrelated. Resistance of flat flexible conductors depends, as is the case of round wires, on cross-sectional area. Specify the final width and thickness by the desired current carrying capacity and voltage drop.

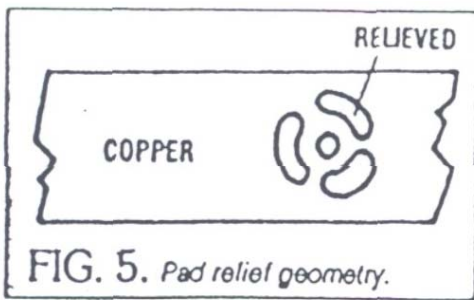
Allow for adequate head dissipation. Flat copper conductors will dissipate heat more rapidly than round wire. The thin dielectric film of a flexible circuit results in less heat build-up than conventional wire coatings.

A range of 1, 2 or 3 oz. copper is typical. For 2 oz. copper use 0.015 inches widths before etching for unspecified (or signal) conductors. Allow for etch loss contraction of flexible circuit layer of 1 mil/side for 1 oz. copper

and 2 mil/side for 2 oz. copper (See Figure 2).

Use 0.015 to 0.020 in. for conductor spacing unless the circuit requires tighter conductor packaging or uses components with high density pin clusters. Spacing of 0.005 in. can be produced with good artwork. To avoid conductor stresses, don't lay them over each other at bend areas and don't put them on the bias in bend areas.





Terminal pads associated with large conductor areas (ground planes, voltage planes, heat sinks, etc.) should be relieved locally to facilitate soldering (See Figure 5). Specify hole sizes with standard drill tolerances. Do not specify hole-to-pad concentricity, specify annular pad. Minimum annular on single-sided circuits (unsupported pad) should be 0.015 in.

Figure 3 shows some of the recommended methods of addressing components:

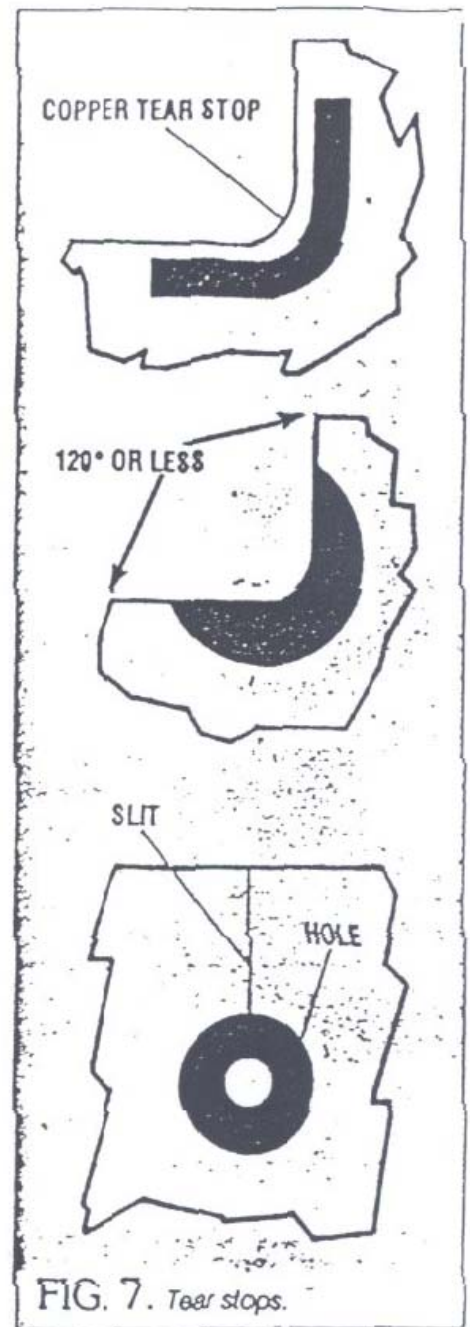
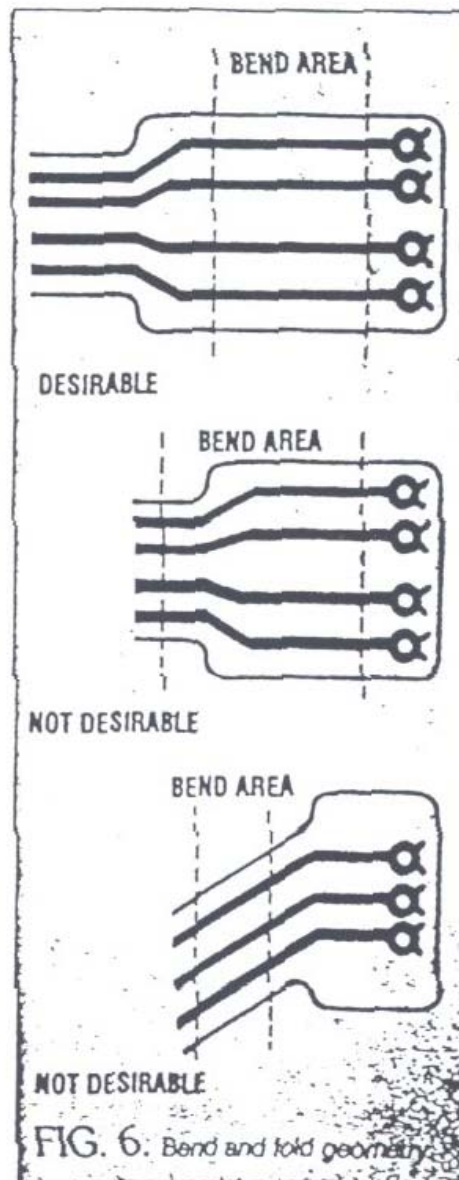
- Right-angle address to multi-row rectangular connectors with single layer requiring conductors to pass between pins.
- Inverted, bilateral address to multi-row rectangular connectors.
- Inverted address to circular connectors with straight rows.
- Right-angle address to circular connectors with straight rows.
- Right-angle multi-layer approach to a circular connector with radial rows.

Make pads as large as possible, at least twice the size of the hole for the component lead to avoid manufacturing yield problems. If circuit densities force smaller pads, consider internal vias for sequential layers. Allow 0.008 to 0.010 in. clearance between the maximum pin diameter and the minimum finished hole size. Design all pads with fillets and tie-down ears to protect against pads lifting during component soldering (avoid potential shorts). The increases in pad area will reduce the chance of hole breakout, prevent stress build-up from drilling, and possible conductor-to-pad interface cracks. Apply minimum spacing requirements to adjacent conductors and pads. Maintain a minimum of 0.050 in. from the edge of a trace to a conductor or pad and allow 0.020 in. for required outline tolerances, material mis-registration and process tolerance build-up.

In connector areas where small center-to-center spacing between pads is used, it may be necessary to stab or truncate pads to run conductors through the pattern. Elongated pads may be used to increase solderable pad area in at least two directions and give the ability to capture the pads with the cover-layer (See Figure 4)

The radius of a bend in a one-layer flexible circuit should not be less than 10 times the circuit thickness. Conductors should run perpendicular to fold lines. Keep them neat and uniform and add extra copper (space permitting) to help hold their folded shape (See Figure 6).

Kapton® tensile strength is typically 25,000 pounds for 1 mil material, but tear strength is low. Copper reinforced areas (tear stops) should be used for corners of 120° or less to restrict possi-



ble damage during assembly. Glass cloth tape and FEP Teflon can be used (See Figure 7).

When pads cannot have tabs or fillets, the cover-coat should lap onto the pad at least 0.005 in. for 270°. Use slots or irregular shapes as a last resort only. Adhesives will extrude 0.003 to 0.010 in. around the cover-coat opening. A very tight hole-to-pad ratio may cause problems meeting specifications. Terminal pad baring can be done by combination, stop on individual pads. When individual pads access holes are used, their diameter should be larger than the pad diameter. Avoid reverse pad baring if possible to reduce labor and processing

costs (See Figure 8). For circuits to be plated with a special finish only on the pads, all plated areas must be run to a common bus outside the circuit. This connection is severed in finishing operations.

Final refinement of conductor runs should result in tolerances and dimensions within the following parameters to result in the most economical and functional flexible circuit.

- Minimum edge distance 0.030 in. Design for 0.050 in. if space is available.
- Pad diameter should be at least twice through-hole diameter.
- Through-hole should be 0.008 to 0.010 in. larger than component lead diameter.
- Minimum trace to trace and trace to hole spacing of 0.010 in.
- Minimum conductor width of 0.010 in.
- Offset conductors from one side to the other to avoid an I-Beam effect in fold areas.

Reinforcement in key areas often is desirable to facilitate component insertion, assure mechanical strength, or obtain the selective stiffness required in some applications. Any standard board material may be used (FR4, G10, Polyamide Glass). Added thickness of dielectric film will provide some stiffening and wear resistance; glass cloth tape also can be used as a stiffener. Hole diameters in reinforcement materials should be at least 0.020 in. larger than through holes. Adhesives for bonding reinforcements can be the same as used in the main circuit or transfer tape can be used. A strain-relief material

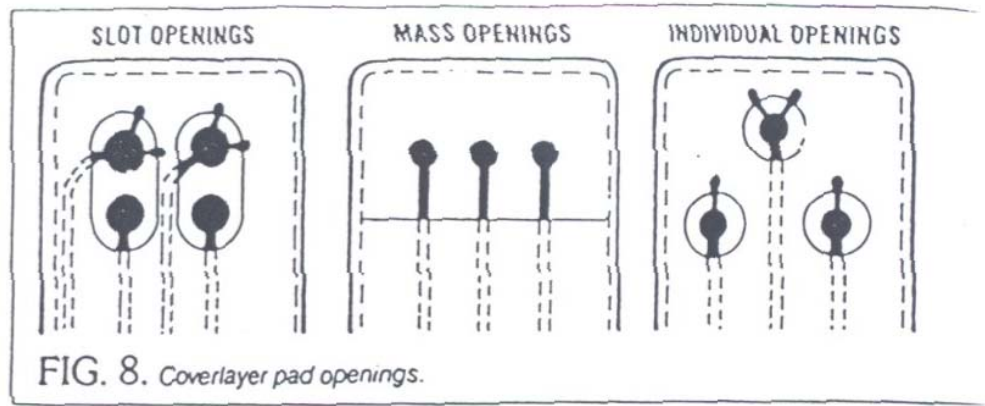


FIG. 8. Coverlayer pad openings.

should be used to form a radius between the flexible circuit and stiffener materials in any sections subject to flexing (See Figure 9).

Flexible circuits can be adapted to any connector design. Provide exact dimensional data for all connector types. Components can be soldered directly to a bared pad in the flexible circuit. Pins can be soldered into flex for direct board plug-in or for wire wrap-around. Exact dimensional data is required including pin diameter, length, and spacing relative to the conductor's center. Insulation displacement terminals penetrate the insulation and are rolled over to mechanically grip the conductor. External shells plug into the crimp-on pins to serve as connectors. Wires can be wired through circuit pads or lap-soldered to circuit fingers. Gold pressure contacts or sculptured posts have limited application.

Terminal areas do not normally require reinforcement since the laminated construction divides stress among all terminations. If additional strain relief is required, additional rigid areas or mounting holes for mechanical fasteners can be

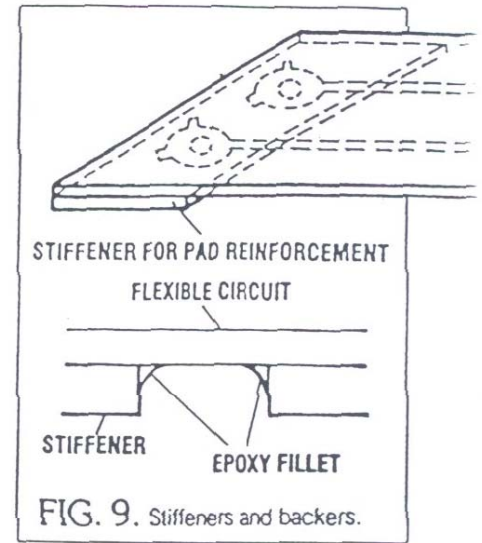


FIG. 9. Stiffeners and backers.

provided. FR4 and G10 stiffeners can be bonded to the circuit. Terminal areas also can be potted for strain relief and environmental protection. Pinheads can be potted or covered with a laminated Kapton® strip, and pin-headers can be furnished to help position fingers for easy assembly.

If necessary, secure the circuit within its package to prevent flopping

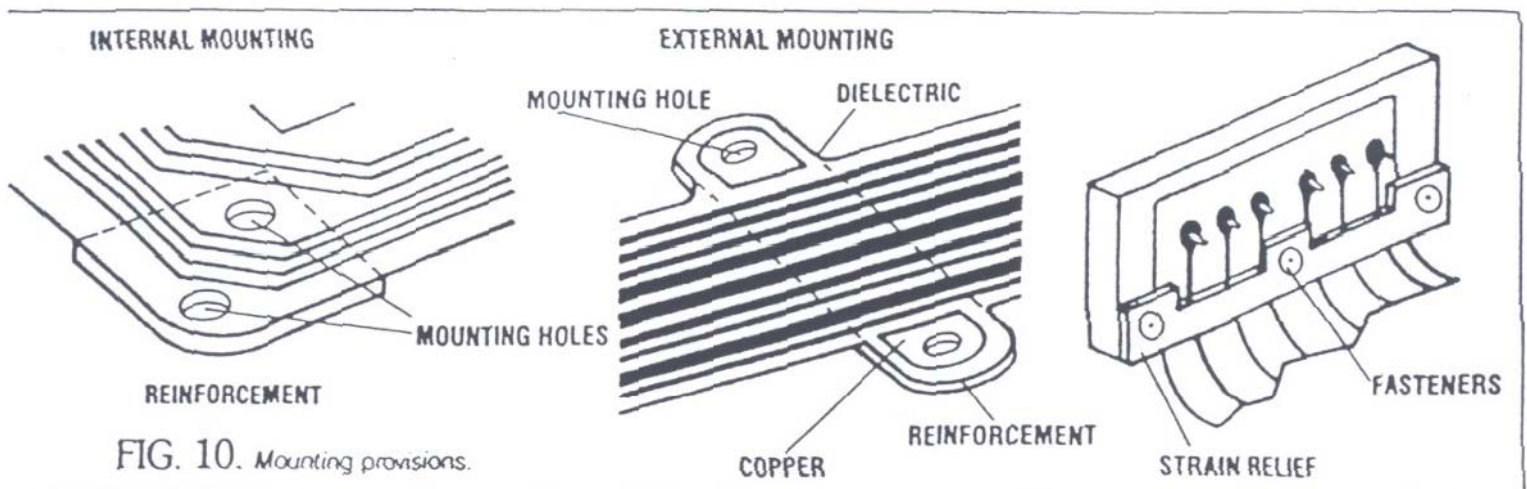
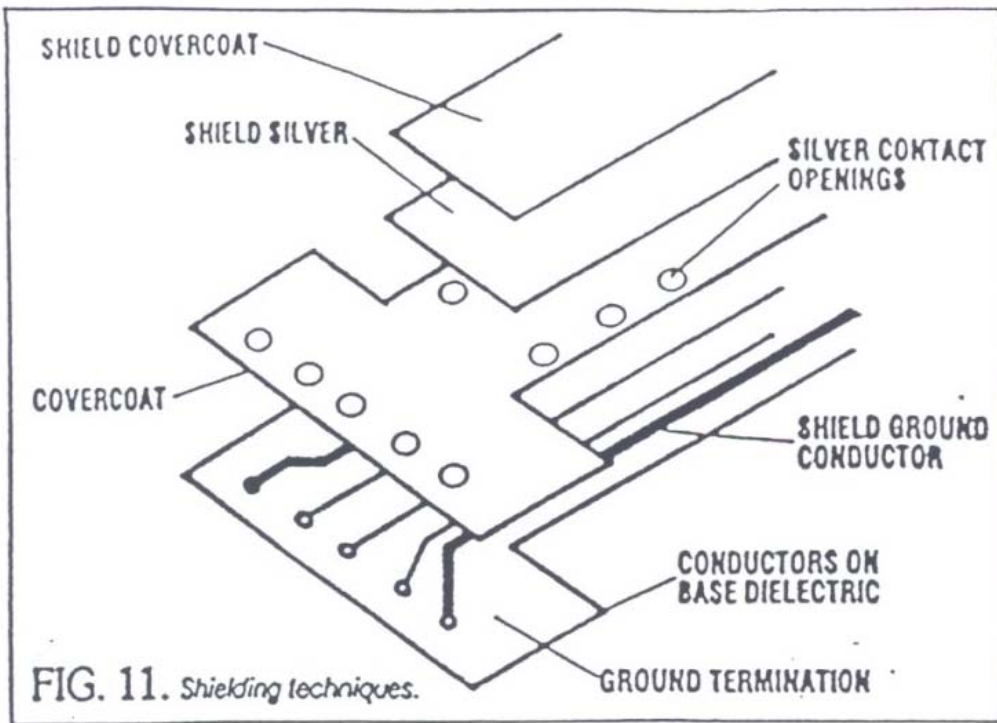


FIG. 10. Mounting provisions.



and provide vibration resistance. Mounting holes can be built into the circuit for internal or external mounting (See Figure 10).

For dynamic flexing 1 oz. copper is the best metal thickness. Use uniform dielectric layers, top and bottom. Provide largest possible flex radius, at least 0.25 in. for extended life. Flex life in ex-

cess of 1 million cycles is attainable.

Shielding & EMI

Since flexible circuit conductors are location-fixed, designing sensitive conductors away from radiating lines can avoid the need for shielding. Grounded guard conductors can also isolate sensitive circuits and eliminate cross talk. Thin

copper foil, copper mesh foil, or silver epoxy compound are the most commonly used materials for shielding. Limit shield use to minimize cost and maintain flexibility. Copper shields are recommended for reliability. A cross-hatch pattern in art work will increase flexibility.

Silver epoxy, applied by spraying or silk screening is used for cost reduction and flexibility. It can terminate many points simultaneously and has excellent isolation capability. Its poor bond strength, however, limits it to outer surfaces only. Another method of shielding is copper foil with a pressure sensitive electrically conductive adhesive. Designate any areas requiring flexing.

Adding inner layer shields to multi-layer circuits will increase the bend radius and reduce flexibility. In cases of high RF energy, video signals and digital pulses, a full 360° shield may be required. This can be achieved by incorporating guard conductors on the edge of the circuit. The guard conductors are exposed in the cover-layer and base dielectric at specified increments. Silver epoxy then is applied to the circuit's top and bottom making contact at the exposed areas. The same result can be achieved with copper shields using plated through holes to make connections to guard conductors.