Flex Circuit Design and Manufacture.

Hawarden Industrial Park, Manor Lane, Deeside, Flintshire, CH5 3QZ
Tel 01244 520510 Fax 01244 520721
Sales@merlincircuit.co.uk www.merlincircuit.co.uk

Flex Circuit Design and Manufacture.

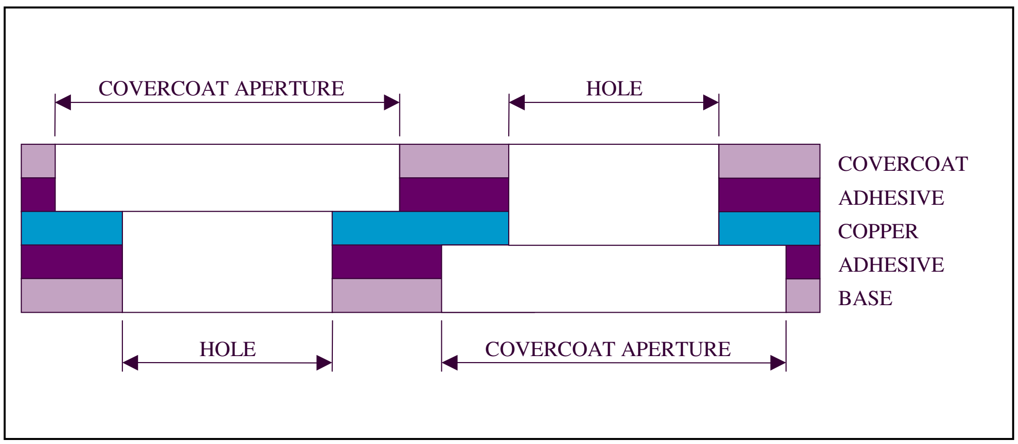
Why Flex circuits

Cost
Weight saving
Space saving
Remove the need for hard wire links from one PCB to another.
Ease of assembly.

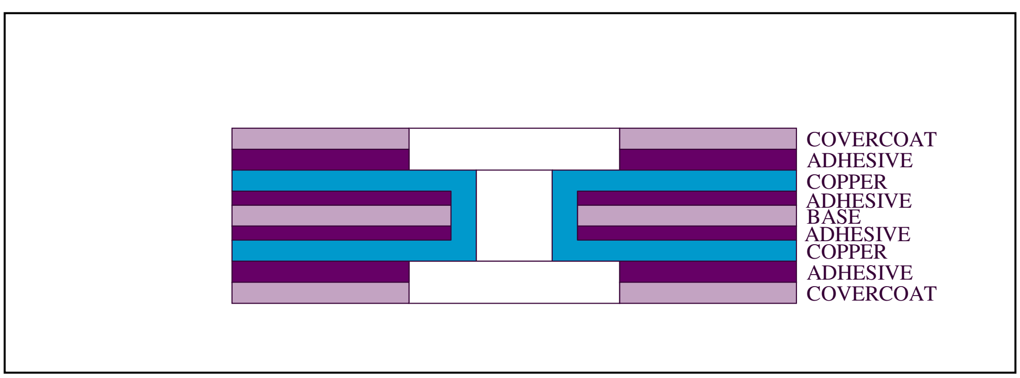
Flex circuit types S/S, Reverse Bared, D/S, M/L.

Single sided PCB's are constructed out of a single sided laminate with a bonded cover coat. The cover coat will be pre drilled to allow access to the copper pads.

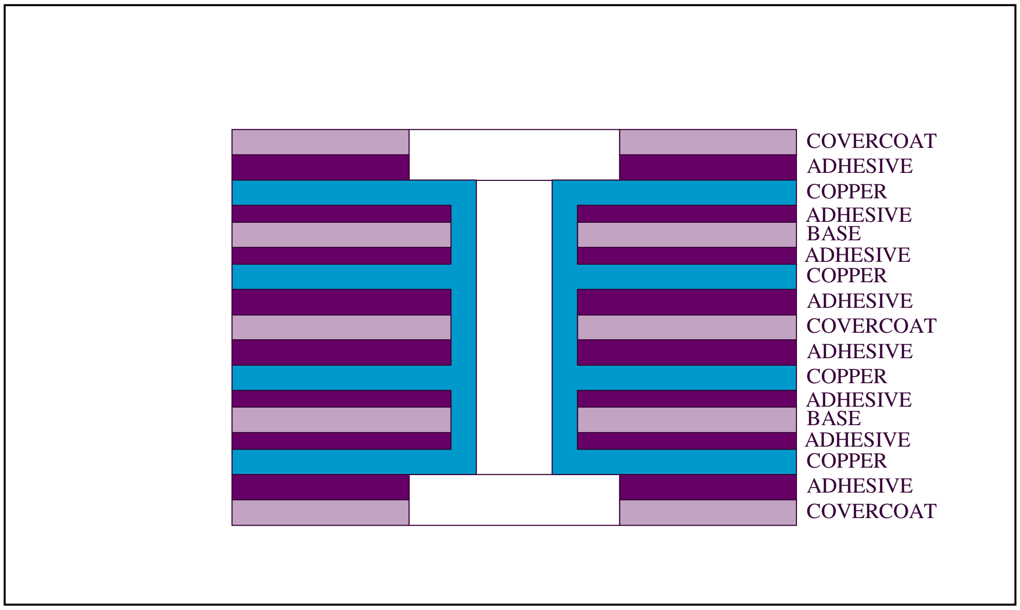
Reverse bared PCB's are similar to single sided circuits but the single copper layer is accessible from both sides of the circuit.



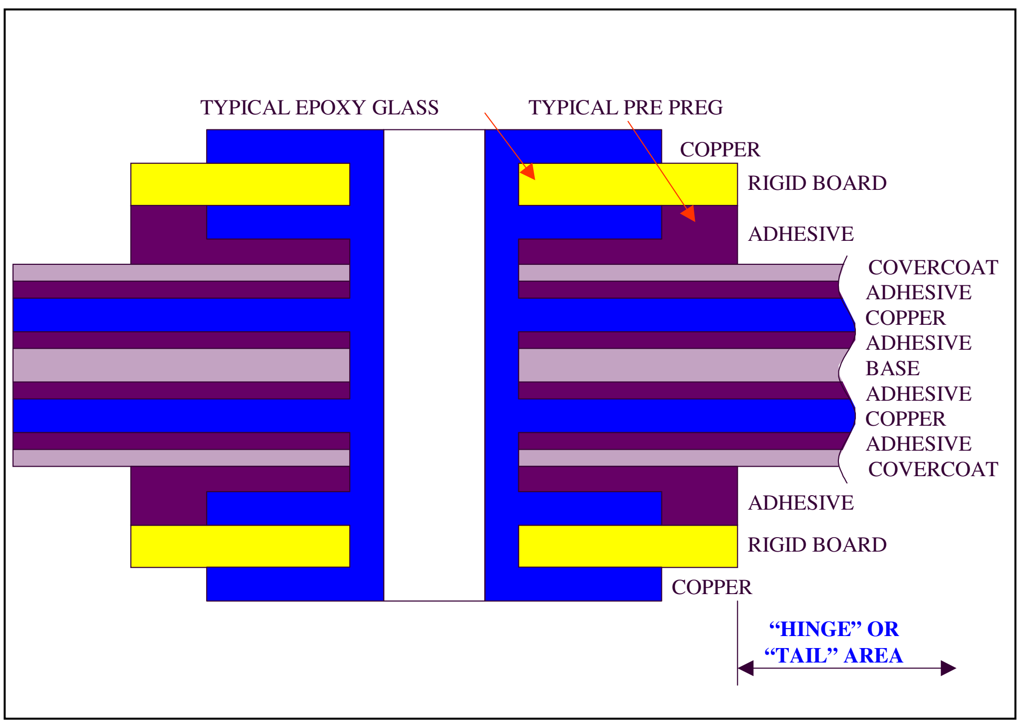
Double sided circuits are constructed out of double sided laminates with pre drilled cover coats bonded top and bottom. This style of circuit can be plated through or non-plated through.



Multi-layer flex comprises of multiple layers of copper connected by plated through holes. Above 4 layers this style of circuit has limited flexibility, it is possible to construct this style of PCB with unbonded areas where the circuit is expected to flex allowing higher layer counts.



Flex rigid multi-layers combine rigid and flex circuit technologies, this style of circuit normally consists of flex inner layer cores and rigid outer layer stacks, the number of flex and rigid layers can vary depending on the complexity of the circuit.



Design criteria.

Material selection.

Base laminate, polyimide high temp material suitable for most flex applications. Polyester is a low temp material not suitable for soldering applications. The most commonly used thickness for polyimide is 0.025mm or 0.050mm and for polyester 0.025mm, 0.050mm and 0.125mm, the thickness will be determined by cost, flexibility and toughness required for the particular application. Common copper thicknesses 18, 35, 70 microns as in the rigid industries are available, again depending on application and current carrying capacity.

Cover coats are normally selected with the same type and thickness of laminate as the base; this gives symmetry to the circuit construction and consistency of performance. If a single sided circuit had a thicker base than cover coat then the copper within the circuit would not be in the centre of the build and give differing flexural characteristics. Adhesive systems used are epoxy or acrylic of a thickness 25, 35, 50 microns, suitable to encapsulate the various thickness of copper.
Cover coat access, cover coats are easily drilled and where large apertures are required then they can be routed or punched. For surface mount technology boards cover coats are not suitable as small rectangular apertures are expensive to produce requiring expensive pierce tooling, in addition the cover coat thickness often prevents small SMT components from sitting flat on the circuit board causing assembly problems.

Solder masks for SMT circuits. The solder masks used for rigid PCB industry are usually epoxy based and when cured are brittle, if applied to a flex circuit then they are liable to crack and flake. Solder masks are available that are specifically design for flex circuit technology, this style of solder mask does have good flexural characteristics but it is still recommended that they be restricted to none dynamic areas as they do not give the support or are as robust as the application of a cover coat layer.

Stiffeners are applied locally to flex circuits to give a greater thickness or stiffness. In some areas on a flex circuit greater stiffness is required for component mounting, a greater thickness may be required i.e. for zero insertion force connectors (ZIF) that must have a thickness of 0.3mm. Stiffeners normally comprise of additional layers of cover coat.

Rigidisers. It is also possible to rigidise areas of the flex circuit for component mounting. Rigidisers, normally FR4 are available in a range of thickness form 0.1mm up to 3.0mm. bonding of rigids can be achieved using D/S tape or thermally bonding.

Copper weights and styles

Rolled Annealed (RA) Copper foils as the name suggests are produced by passing the copper through a series of rollers with decreasing gaps until the desired thickness is achieved. The action of rolling the copper creates a long flat grain structure that is very ductile. This style of copper is used where repeated flexing is required in what is termed as a dynamic application such as a printer head flex connection.

Electro Deposited (ED) Copper. This style of copper is produced by plating the required thickness and is suitable for most flex applications. Although not recommended for fully dynamic situations, it has excellent flexural characteristics.

Sputtered Copper. This style of copper produces very thin deposits as low as 2um, though more commonly 5 or 7um. This enables much finer features to be produced in the etched pattern.

General lay out.

Positioning of PTH holes away from flexing areas will prevent cracking of the plating within the holes

Stiffeners can be used to encourage the flex to bend in a specific area.

Via pads do not need to be exposed and can be tented with cover coat.

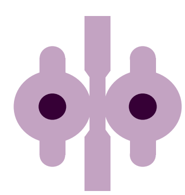
Shielding can be achieved by using additional copper layers (normally hatched) or silver inks.

Multi-layer circuit tracking details should be kept away from the flex/rigid interface, clearance recommended 1.0mm.

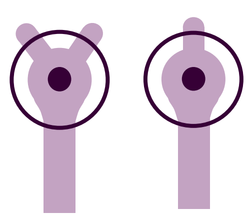
Pad to hole ratios and ground plane clearances should be enlarged where ever practical on flex layers, as the movement of flex circuit material is greater than that of rigid materials. Hole sizes / pads to hole ratios. Smallest finished hole size 0.2mm (drill at 0.35mm) pad 0.75mmm, ie 0.2mm radially larger. GND plane clearances 1.15mm.

Tracking

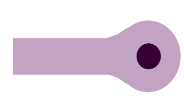
Fine line tracks between pads may be required
Normal production track and gap min 0.125mm
However this should be kept to a minimum and
Tracks enlarged wherever possible



Ears on pads



Fillets



Track widths in flex hinge areas should be increased wherever possible for greater strength. Solid copper shield can be hatched to give greater flexibility.

Symmetrical builds with the flex in the centre of the construction are preferable for flex rigid applications.

Note: Flexibility is dramatically reduced for PTH circuits as the ED copper deposited during the processing is more brittle than the RA, repeated flexing of this style of circuits will result in open circuits caused by track breaks.

Bend radii. Normally 10 x material thickness

Methods of connection. Stake through pins, Berg clincher connectors, zero insertion force connectors.

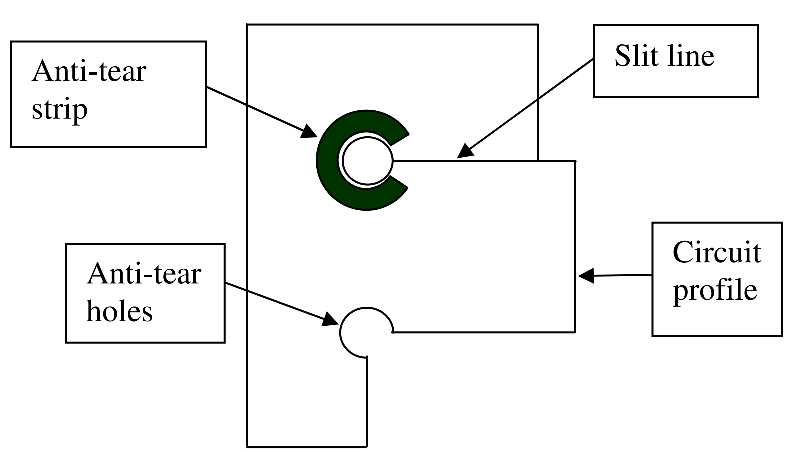
Metal finishes.

Most standard finishes used in the manufacture of rigid circuits can be applied to flex circuits, the finishes Merlin Circuits can offer are Hot Air Solder Level (HASL), Electroless Nickel and Gold, Immersion Tin and Immersion Silver. Note: Copper is very ductile, many of the solderable finishes are extremely brittle and if applied to an area, which is exposed to flexural stresses, then open circuits can be expected.

Profiling.

When deciding on the profile of a flex circuit there are a number of considerations:

Track to profile, a minimum of 0.25mm is recommended 0.5mm or larger is preferred, however the tolerance of the profiling operation must be added to this to give the actual clearance by design.
Internal corners should have radii avoiding sharp angles in the profile; sharp corners are potential stress points where tears are likely to start.
Where sharp corners cannot be avoided or slits are required then anti-tear holes should be used.
Anti-tear strips can be added to the copper detail to strengthen sharp corners and anti-tear holes.



Steel Rule Die tools (SRD), these are precision laser cut base plates with metal blades formed to the required profile dimensions. This style of tool is commonly used as the tools are relatively inexpensive however they have a limited life, 2000 hits, and are only accurate to +/-0.25mm.

Pierce profile tools are precision spark eroded die sets used in power presses or fly presses. These are substantial tools used when a greater accuracy, (+/- 0.1mm), is required or high volumes are expected.

Routing may be used for smaqll quantities where a greater accuracy is required than that achievable using SRD tools. The draw back to routing is that flex material being soft does not rout smoothly and leaves a rough edge. The rough edges can be cleaned but this is a time consuming manual process, which only lends itself to small batch production. The accuracy expected from a routed profile is +/- 0.1mm.

Laser Profiling. Modern laser techniques give the accuracy of the router, the finish of the SRD tool but without the high expenditure of hard pierce profile tooling.