

P.W. CIRCUITS LTD

Premier Works, Canal Street, South Wigston, Leicester, England, LE18 4PN
Tel +44(0)116 278 5241 Fax +44(0)116 277 1072

E-Mail: sales@pwcircuits.co.uk Web: WWW.pwcircuits.co.uk

MATERIAL AND DESIGN RULES FOR RIGID PCB'S

Material

Laminates FR4, Polyimide, Rogers, Arlon. **Thicknesses** 0.050mm to 6.000mm (Standard 1.600mm)

Copper Thicknesses 18 microns (½ oz) to 360 microns (10 oz) (standard 36 microns 1 oz)

Finishes Hot Air Solder Level (HASL) Spb (Not RoHS Compliant), Electroless Silver, Electroless Nickel Immersion Gold, Electrolytic Gold (hard Gold).

Solder Resist Colours Green, Blue, Black, Red, Orange, White, Clear (Standard Green).

Legend Colours White, Yellow, Black, Red, Orange (Standard White)

Manufacturing capabilities

Standard track and gap against Copper thickness.

0.150mm (0.006") = 18 microns 0.5oz

0.200mm (0.008") = 35 microns 1.0oz

0.250mm (0.010") = 70 microns 2.0oz

Drill and route sizes.

Minimum drilled holes size 0.200mm

Maximum drilled holes size 6.400mm

(Larger holes will be routed)

Minimum router size 0.800mm

(Standard route size 2.400mm)

Annular ring.

Minimum annular ring 0.150mm (0.006")

Pad size to be minimum of 0.400mm

(0.016") larger than hole size.

Advanced track and gap against Copper thickness.

0.100mm (0.004") = 18 microns 0.5oz

0.150mm (0.006") = 35 microns 1.0oz

0.200mm (0.008") = 70 microns 2.0oz

Drill and route sizes.

Minimum drilled holes size 0.100mm

Minimum router size 0.600mm

Annular ring.

Minimum annular ring 0.100mm (0.004")

Pad size to be minimum of 0.300mm

(0.012") larger than hole size.

Design notes for rigid pcb's

- Try not to design your circuits to the minimum capabilities a circuit with every aspect at the limit is a manufacturing nightmare and will cost you more.
- Traces should be drawn at forty five and ninety degree bends.
- Solder resist pads should be a minimum of 0.150mm (0.006") larger than copper pad.
- Legend minimum line width 0.200mm (0.008").
- Legend minimum clearance to copper pad 0.200mm (0.008").
- Allow 0.200mm clearance from profile edge the nearest trace or pad.
- Data supplied Gerbers should be RS274X format, Drilling should be Excellon format.

The information on this leaflet is only a small selection of our capabilities and products please contact us today so we can discuss your requirements.