

Designing PCBs for Economy

- 1) Sizes & Clearances. For economical production, the following should be used as absolute minimums. Always allow extra where possible. Smaller clearances are achievable but costs increase. For single sided, drill size = hole size. For DSPTH, drill size = hole size + 0.1mm. DSPTH = Double Sided Plated Through Hole. SP = Screen Printed, PI = Photo Imaged.

HOLE SIZE	Single sided, SP copper	0.8mm
	Single sided, PI copper	0.7mm
	DSPTH	0.6mm
COPPER PAD SIZES	Single sided, SP copper	Hole + 0.8mm
	Single sided, PI copper	Hole + 0.6mm
	DSPTH	Hole + 0.5mm
SOLDER MASK PAD SIZES	SP solder mask	Pad + 0.6mm
	PI solder mask	Pad + 0.3mm
TRACK WIDTH	Single sided, SP copper	0.30mm
	Single sided, PI copper	0.25mm
	DSPTH	0.20mm
COPPER TO COPPER	Single sided, SP copper	0.30mm
	Single sided, PI copper	0.25mm
	DSPTH	0.20mm
COPPER TO OUTLINE/SLOT	Routed outline	0.25mm
	Punched outline	0.30mm
	Scored	0.50mm

Examples (using 1mm hole):

Screen printed copper and mask, copper pad = 1.8mm, mask pad = 2.4mm

DSPTH, PI copper & mask, copper pad = 1.5mm, mask pad = 1.8mm

2) Single sided production flow:-

PRINT TRACKS, ETCH, SOLDER MASK, FINISH (EG HASL), DRILL, LEGEND, PROFILE

3) DSPTH production flow:-

DRILL, IMAGE TRACKS, PLATE, SOLDER MASK, FINISH (EG HASL), LEGEND, PROFILE

4) Screen printing / Photo imaging. Printing is a lower-cost method of applying solder mask (SS & DSPTH) and track patterns (SS only). OK for smaller panels but not for larger panels because the registration becomes less accurate towards the outside of the panel. Photo imaging of either track or solder mask produces more accurate results.

5) Include some text on each layer to assist identification and prevent the board being made inside-out.

6) Design as seen from the component sided ie for DSPTH, layer 1 = component side (top), layer 2 = solder side (bottom).

7) Profile info. Industry protocol is that if the outline is complete (ie not crop marks) we measure to the centre of the line and ignore the line width. If the outline is incomplete (crop marks), we measure to the inside edge. Good practice: Don't put profile lines or crop marks on copper, mask or legend layers (we remove them anyway). The profile should be on a separate layer and consist as follows:

- a) pad plot (preferred) or targets to allow registration
- b) complete external outline in smallest possible aperture (0.01mm)
- c) any internal slots either in actual aperture (eg 1.6mm line for 1.6mm slot) or complete outline in 0.01mm aperture.
- d) Info on any internal & external radii
- e) integrate the drill plot if possible.

Remember that profiling is the last operation in the production process and is separate from the drilling process. Hole to hole registration is usually very good but hole to profile is less precise. Try to avoid situations where a component relies on a close profile tolerance for fitting - consider using multiple over-lapping holes instead (chain-drilling).

8) Route tools come in the most sizes but unlike drill tools, factories don't stock them all. Routing is an expensive process so keep to common sizes where possible ie don't specify 1.5mm slots if 1.6mm is OK. Most commonly used sizes (in order of preference) are: 2.4mm, 2.0mm, 1.6mm. Sizes below 1.6mm are best avoided if possible because the tools are much weaker so the route speed is much slower (and therefore more expensive). Routing is performed on stacked pcbs ie 1, 2 or 3 can be routed simultaneously depending on the tool size and thickness of the PCB.

TOOL SIZE	SECTION AREA	MAX THICKNESS
1.0mm	0.78mm ²	1.6mm
1.2mm	1.13mm ²	1.6mm
1.6mm	2.01mm ²	3.2mm
2.0mm	3.14mm ²	3.2mm
2.4mm	4.52mm ²	4.8mm

9) Specify the size of any radius that is important but try to keep to 1 tool size only.

10) Build information. Should be included in the gerber files. Typically includes: material type and thickness, copper weight (bare and/or finished), colours of mask & legend, profile info (panelisation, slots, margins), finish (flux, HASL, Ni/Au etc).

11) Non PTH fixing holes. Info should be included in the drill file or gerber files. If possible reduce any associated copper pad to smaller than the drill size.

12) Non PTH tooling holes. If registration is important (perhaps for auto-insertion or testing), these have to be drilled at the same time as all other holes. Please inform us.

13) Profiling techniques (route, score, punch).

- a) Scoring. The cheapest but least accurate method. Obviously only applies to circuits with straight edges but can be combined with routing or punching. Ensure adequate copper free area along the scored edges. Score width on 1.6mm material is approx 0.020" (ie you lose 0.010" of each circuit) but add an extra 0.10"

on each side for working tolerance. In total, you need 0.020" (0.5mm) of copper free material.

- b) Routing. See notes above for tool sizes. Routing can be accurate but generally is not as repeatable as punching. The route tool is similar to a drill bit but has cutting edges on its sides. Remember that profiling is the final stage in production. Imagine a situation where the circuit is 20mm round, stepped 20 up on a panel. Problem is that when the tool tries to cut sideways through the material then the panel flexes away. If it's the first circuit then the panel is still fairly rigid so the result will be good but the opposite applies to the last circuit on the panel. By then, the panel will be much less rigid and the results less accurate and less predictable. For example, circles will become oval to some extent. The more material removed from the panel the more the acute the problem. On the plus side: the vertical edge of the cut will be clean, the route program easily modified. Don't consider a punch tool unless you're sure of the profile. Allow at least 0.010" copper free area around the circuit profile.
- c) Punching. Approx as accurate as routing but critically: more repeatable and much less expensive. Unlike routing, the first circuit will be the same as the last circuit. Also, the first panel will be the same as all the other panels. The down side is the punched edge is less neat than routing and the tool can be difficult / expensive / impossible to modify (punch tools in the Far East are approx 20% of European prices). Also, internal radii on punch tools can be 0.5mm without any adverse implications. To achieve the same with routing requires a 1.00mm tool which is very slow and expensive. Allow at least 0.010" copper free around the circuit profile.

In summary, the relevant factors are: shape of the circuit, panelisation requirements, thickness of material, design stability, cost.

14) Test Jigs. Come in 2 types: Permanent or rebuildable (grid based).

Permanent jigs cost more initially and perform better for large volumes. To a minor extent they are modifiable. They are more suitable for high density designs because of the flexibility in pin placement. Cost approx £0.50 per test point, minimum £200.

Rebuildable jigs are more suitable for (a) large panels with many test points, (b) where the design may change rapidly.

15) Materials. Many different types available but most popular as follows:-

FR2	<p>Paper / resin material.</p> <p>For: Lowest cost, lightweight, easy to profile, ideal for punching (profile and/or holes).</p> <p>Against: Poor adhesion between copper and laminate therefore doubtful for hand soldering especially with small pads. Danger of warping during production and/or flow solder. Susceptible to cracking. Poor dimensional stability (material shrinks during process). Suitable for simple single sided only.</p> <p>Available in usual thicknesses (mainly 0.8, 1.6mm, 2.4mm, 3.2mm) and weights (1/2 oz, 1 oz, 2oz).</p>
CEM1	<p>Composite material: Glass - paper – glass. CEM1 has outer layers of glass fibre, inner of paper/resin.</p>

	<p>For: Better copper adhesion. Less prone than FR2 to warping. Better dimensional stability than FR2.</p> <p>Against: More expensive than FR2. More expensive than FR2 to drill & route. Less suitable for punching holes than FR2. Heavier than FR2. Usually only considered for single sided.</p> <p>Thickness and copper weight as above.</p>
CEM3	<p>Composite: Glass fabric – glass felt – glass fabric (no paper)</p> <p>For: Suitable for DSPTH as a lower cost alternative to FR4.</p> <p>Against: More expensive than CEM1 to buy, drill & route.</p> <p>Thickness and copper weight as above.</p>
FR4	<p>Glass fibre / resin.</p> <p>For: Suitable for all normal uses.</p> <p>Against: Most expensive of the routine materials. Unsuitable for punching holes below 2.00mm. Most expensive to drill & route.</p> <p>Wider range of thicknesses eg 1.0, 1.2mm. Copper weight as above.</p>

TRACK WIDTH. 3mm @ 1oz = 10AMPS @ 20-30 DEGREES C

INCHES	THOU, MIL	MICRO INCH (u")	MM	MICRONS (uM)
1.0	1000		25.4	25400
0.1	100		2.54	2540
0.01	10		0.254	254
0.001	1	1000	0.0254	25.4
0.0394	39.4		1.0	1000
0.00394	3.94		0.1	100
0.0014	1.38		0.034	35
0.00067	0.67		0.017	17
0.000394	0.394		0.01	10
0.0000394	0.0394	39.4	0.001	1 (1 uM)
0.000001	0.001	1	0.0000254	0.0254