

Copper balance between layers

Copper should be symmetrically distributed in layers both side of the core. In our example the layer pairs are 1-8, 2-7, 3-6 and 4-5. The layer pairs should have equal copper areas. If not, the board will twist.

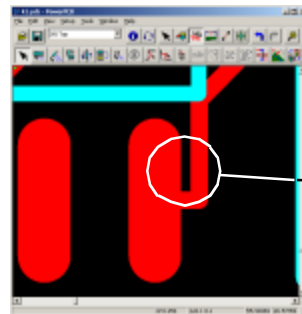
The most popular way is to use Pour copper to fill free areas on routing layers at the end of design. Hatched copper is not recommended.

It is a good idea to take a view copper distribution on each layer and look for small openings in middle of large copper area (2 to 5 cm), such as a short embedded trace or a pad. If you find one, adjust the spacing greater than elsewhere.

Same Net rules

"*Same Net*" rules affect to fabrication details. For example by setting the corner to pad rule you can prevent corners to be placed too close to pads in autorouting. This will reduce "*Acid Traps*".

In fabrication process photo-sensitive resist is applied on black areas (see the sample picture). In the development process, the narrow part (circled) may come off. When the board is lifted up from the liquid, the tip will fall back, but not necessarily to same location. This would lead a cut in either pad or trace.



Thermal Relief parameters

A thermal relief connection is required for mounted through hole parts. The purpose of the relief is to prevent heat conduction to plane, allowing the solder temperature to sufficiently rise inside the hole. If too much heat is dissipated to the plane, the solder will not fill the hole properly, and the joint will be unreliable.