Pad Design

Through-hole components

For through-hole components, the pad configuration is a compromise between several competing requirements. Board density and insulation resistance demand the use of the smallest outside diameter possible, whilst joint strength and reliability are best served if the pad is as large as possible to increase the copper peel strength. Easy soldering, however, requires a pad that is wide enough for repetitive wetting and not too wide for solder to drain away from the lead wire.

The following guidelines should be followed:

- Use round pads whenever possible. Oval pads are acceptable, but square ones should be avoided because the solder cohesive forces draw the solder into a sphere.
- Ensure a minimum annular ring of 0.5mm in all directions to provide the required strength.
- Avoid large pads with multiple holes. When this cannot be done, pads should be separated with solder resist, otherwise some parts of the common pads will not fillet properly, and icicles may form.
- Isolate large ground planes from the pad by using cartwheel spokes for uniform heat transfer.

Other areas of good practice include:

- Providing the right amount of through-hole clearance, as this affects the shape of the joint and its reliability (Figures 1 & 2)
- Preventing open non-plated holes from becoming totally covered by solder by measures such as the C-pad (Figure 3)

Figure 1: Over-large through hole

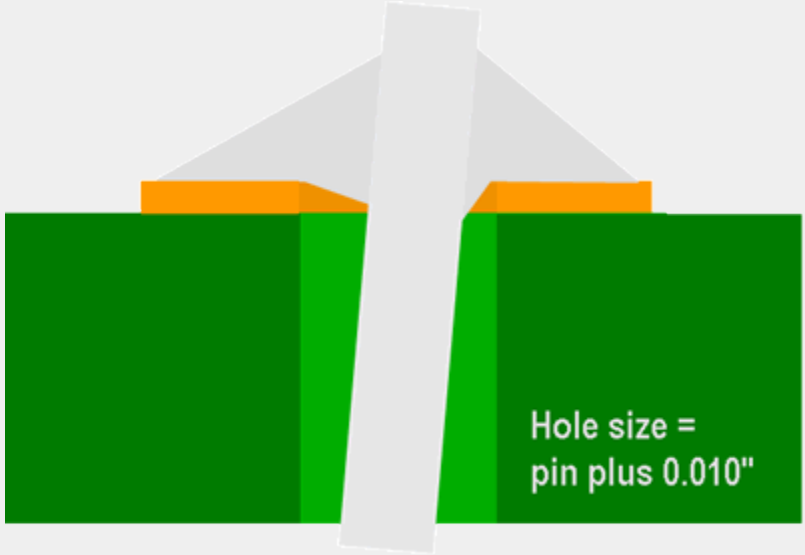


Figure 2: Cracks in weakened area caused by an over-large hole

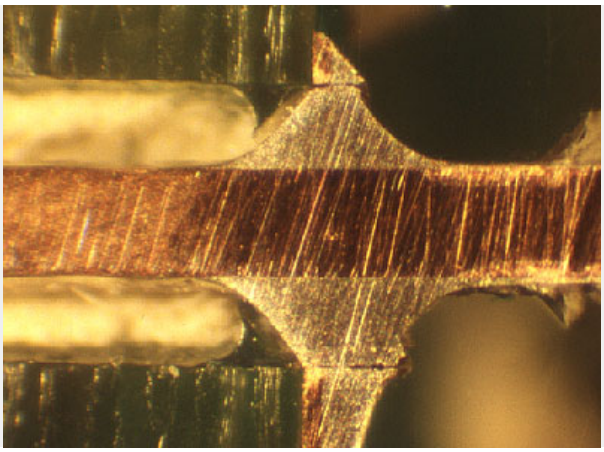
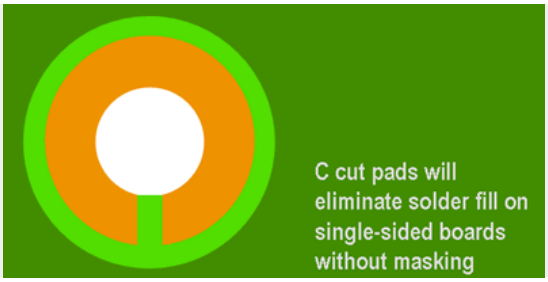


Figure 3: C-cut pad to eliminate solder fill



Surface-mount components

For SM components, as in through-hole designs, layout density favours the smallest area, while joint stability indicates the largest pad possible. Manufacturing tolerances, including adhesive application and component placement, also dictate the need for large pads. Pad dimensions for **wave-soldering** of surface-mounted assemblies should also ensure as large a contact area as possible, to reduce any problems of lack of consistent wetting. Because of the geometry of the components glued to the surface, there are areas where gaseous material can be trapped, causing skipping or misses on small pads: the larger the pad, the less likely this problem is to occur.

Printed wiring board layout rules for **reflow soldering** are controlled by three considerations:

- All components must stay precisely in their positions, so that the surface tension forces have to 'work for' the joint.
- All of the solder deposited must contribute to forming the joint, and none must be tempted to flow away from it.
- In order to make it easier for all joints of a component to reach the same pre-reflow temperature at the same time, the heat capacity of component pads should be uniform.

Other design considerations

Not only does the copper area of the pad need to be considered, but so must the design and quality of **solder resist**. Resist is used to define the solderable areas, and needs to be both well-defined and resistant to the soldering process conditions.

As well as being used to connect to components, contact areas may also be dedicated as **test pads**. The majority of automatic testing is carried out by placing the board on an array of spring-loaded-probes (bed of nails). These probes make contact with the test points which should preferably be separate pads, away from the component joint in order to prevent component damage.

Source : http://www.ami.ac.uk/courses/topics/0169_padd/index.html