

## The design rules for 2/4 layers PCB

A. for the **inner layer**, the minimum clearance between the tracks should be 0.13mm, the minimum width of the tracks should be 0.1mm.

B. For the **outside layer**, the minimum clearance between the tracks should be 0.15mm, the minimum width of the tracks should be 0.13mm.

C. The minimum diameter of the vias should be 0.2mm, the minimum clearance between the vias should be 0.3mm, the minimum clearance between the through-holes should be 0.4mm

D. The minimum pad diameter should be 0.3mm, the minimum clearance between the pads should be 0.4mm

E. The minimum clearance between the tracks and the copper sheets should be 0.2mm, the outer via diameter should be 0.26mm larger than the inner via diameter.