

Table 2, typical trace width and spacing values

<i>PCB type</i>	<i>Trace width</i>	<i>Spacing</i>	<i>Via pad and hole size</i>
One layer board, no traces between DIPs	0,5...1 mm (20-40 mils)	0,35 mm (15 mils)	(jumper wire - 1,6 mm, 1 mm)
One layer board, one trace between DIP	0,33 mm (13 mils)	0,3 mm (12 mils)	(jumper wire - 1,6 mm, 1 mm)
Two layer board, 1 trace between DIP	0,33 mm (13 mils)	0,3 mm (12 mils)	1,4 mm, 0,7 mm (55, 28)
Two layer board, two traces between DIP	0,2 mm (8 mils)	0,2 mm (8 mils)	0,9 mm, 0,6 mm (36, 24)
SMT boards	0,2 mm (8 mils)	0,2 mm (8 mils)	0,8 mm, 0,5 mm* (32, 20)
Dense SMT boards	0,1 mm (4 mils)	0,1 mm (4 mils)	0,6 mm, 0,4 mm* (24, 15)
Boards with uBGA-packages	75-100 $\mu$ m (3-4 mils)	50-75 $\mu$ m (2-3 mils)	laser vias, 350 $\mu$ m, 150 $\mu$ m

\*)The Aspect Ratio must be taken into account as well, see "How to select a via size, Aspect Ratio" on page 113.

The values shown are just a starting point. They cannot be applied if there are high currents or frequencies. Reducing the via size under 0,5 mm will increase the manufacturing cost.

Also, it is desirable to limit trace widths less than 100 $\mu$ m to a small area such as under an **n**BGA package where it is absolutely necessary. To manage this you