

## Minimum distance from drill to copper area in the inner layer

Minimum distance from drill to copper area in the inner layer – this is a parameter used in manufacturing, i.e. a value demonstrating the technological capability of the PCB manufacturer. During design, all hole sizes are defined as final, i.e. together with plating, which means that the PCB manufacturer must increase all initial drill sizes by 0.1 mm. Consequently, the minimum distance from the drill edge to a copper area decreases by 0.1 mm and if it is less than 0.15 mm, it is no longer technologically possible! This parameter is of great importance for ensuring the long-term functionality of the printed circuit board. When designing distances of less than 0.25 mm, the probability of a thermal expansion short circuit increases. In the case of particularly compact boards, it is possible not to increase the vias and use the minimum distance. In such case, the contracting entity/designer should add a corresponding remark in the design documentation or the order.

