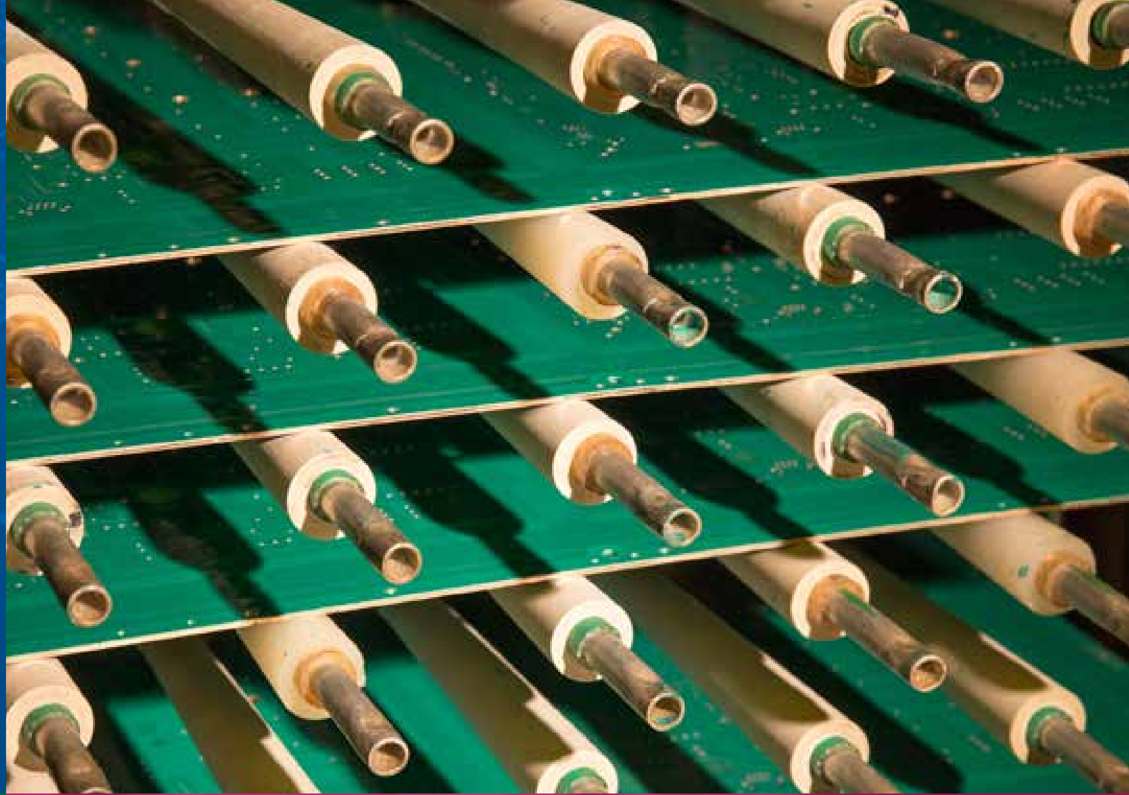
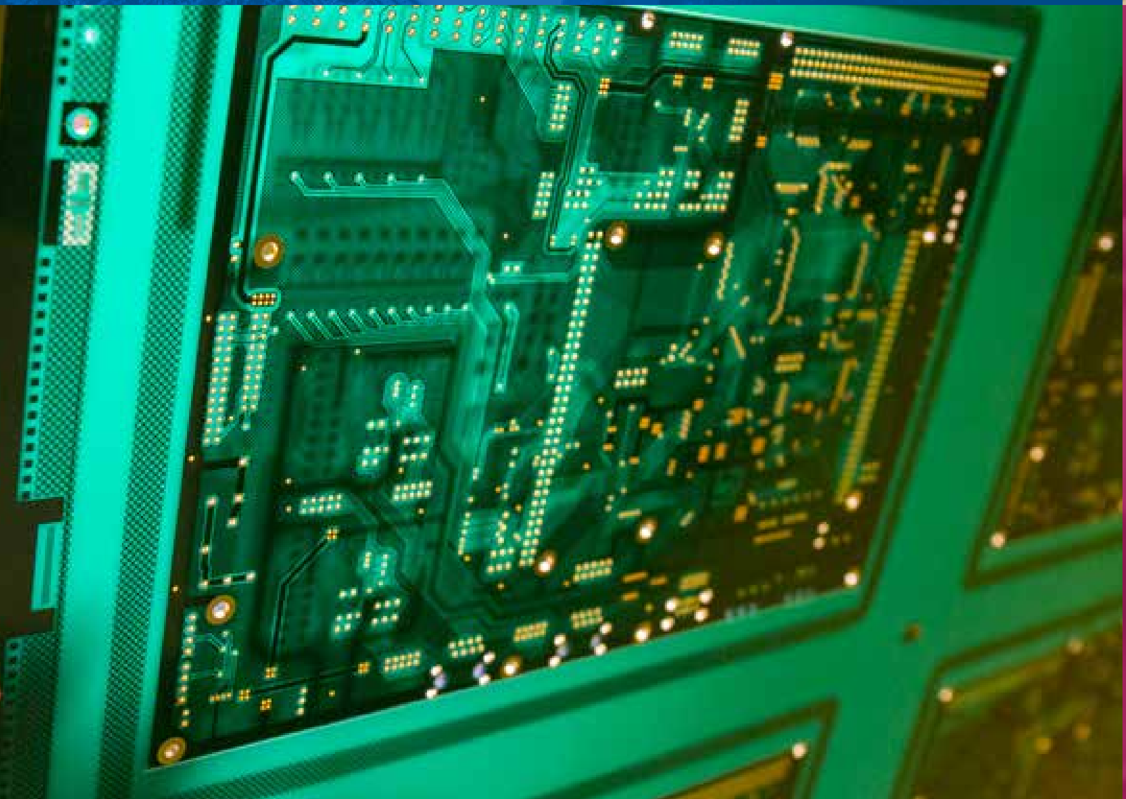
How to pay less for PCBs?

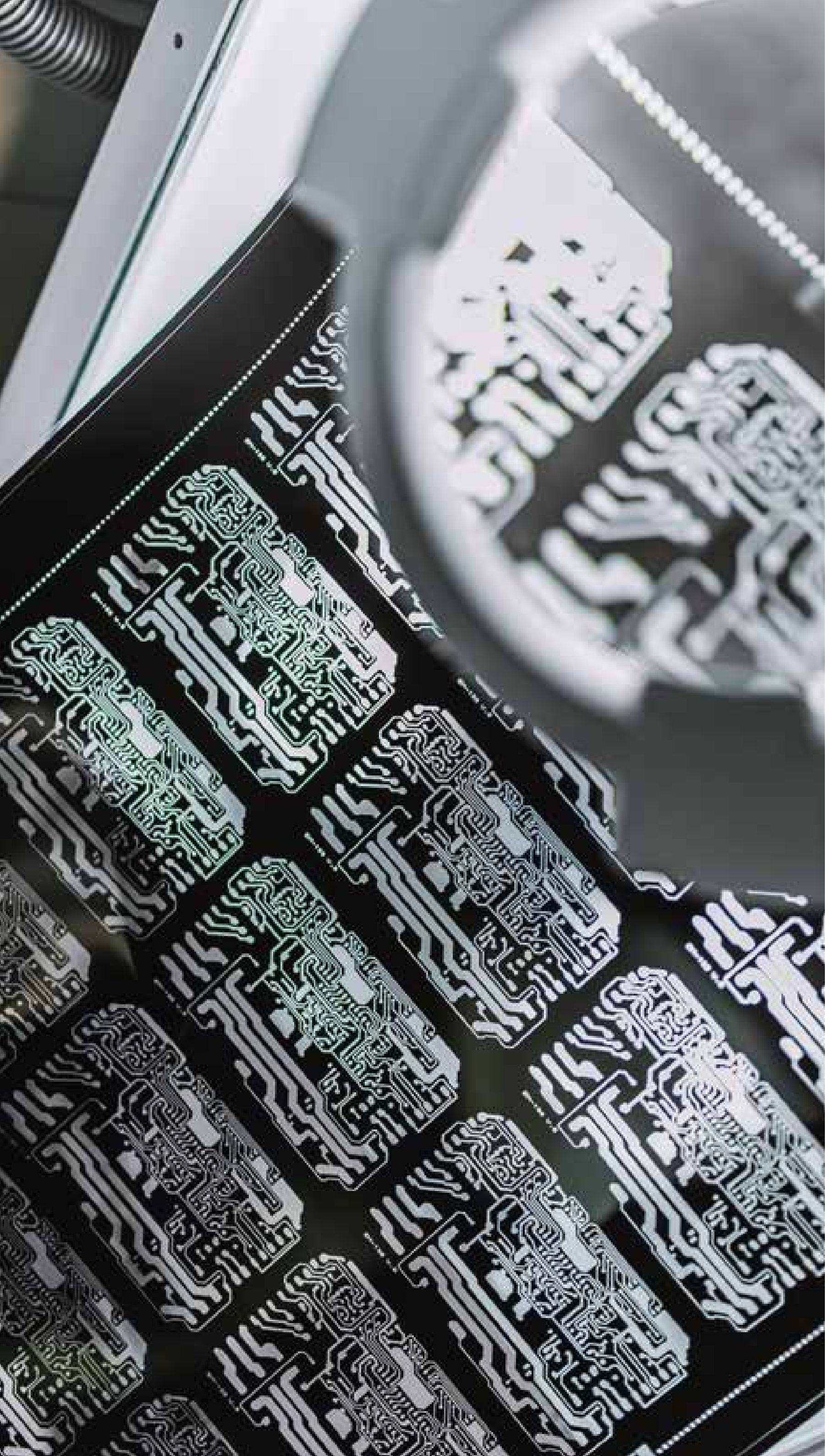




Proper design

TS PCB

tspcb.pl



Elaboration of technological documentation is first and one of the most important elements of the entire PCB production process.

The way the circuits are designed translates directly into the speed of their manufacture, final quality and cost.

Already at the design stage, designers decide whether the PCB production process will take place without the need for additional consultations and explanations with the manufacturer's technical team. It is also at this stage, that the risk of quality issues can be minimized.

By introducing minor changes to the design, many potential errors can be eliminated, thus ensuring a higher quality of the end product. Ambiguities and imprecise information in the documentation often cause big issues with the correct interpretation of customer expectations and may lead to the production of circuits that do not meet the customer's requirements.

Recommendations of manufacturers

Correct documentation

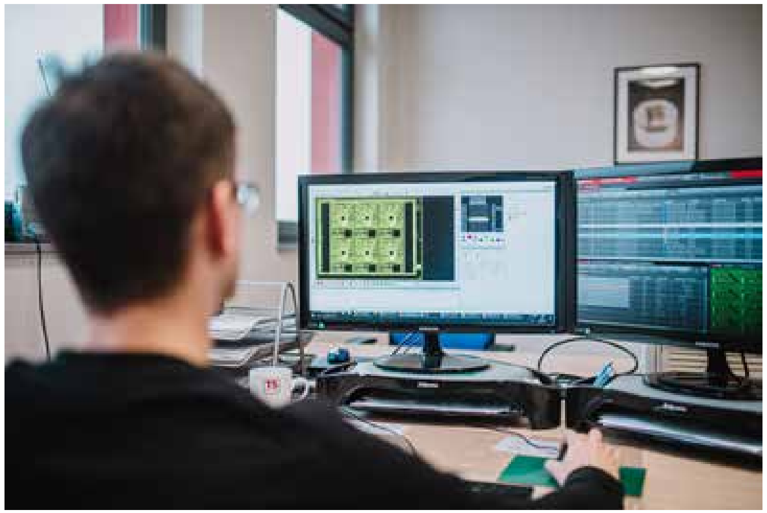
Irregularities

Final laminate thickness

Recommendations of manufacturers of printed circuits

The most important group of parameters of a printed circuit are its electrical characteristics. Before starting the design, however, it is worth getting acquainted with the production capabilities of the supplier with whom the order is placed. Each manufacturer defines their principles and parameters according to which the design should be prepared. Among others, the following are recommended:

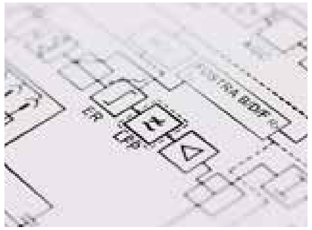
- minimum distances between conductive elements,
- minimum widths of conductors,
- minimum hole diameters and annual ring sizes,
- minimal exposure on the mask,
- maximum number of layers, etc.



Correct production documentation

should contain a minimum number of files to ensure its easy and unambiguous interpretation.

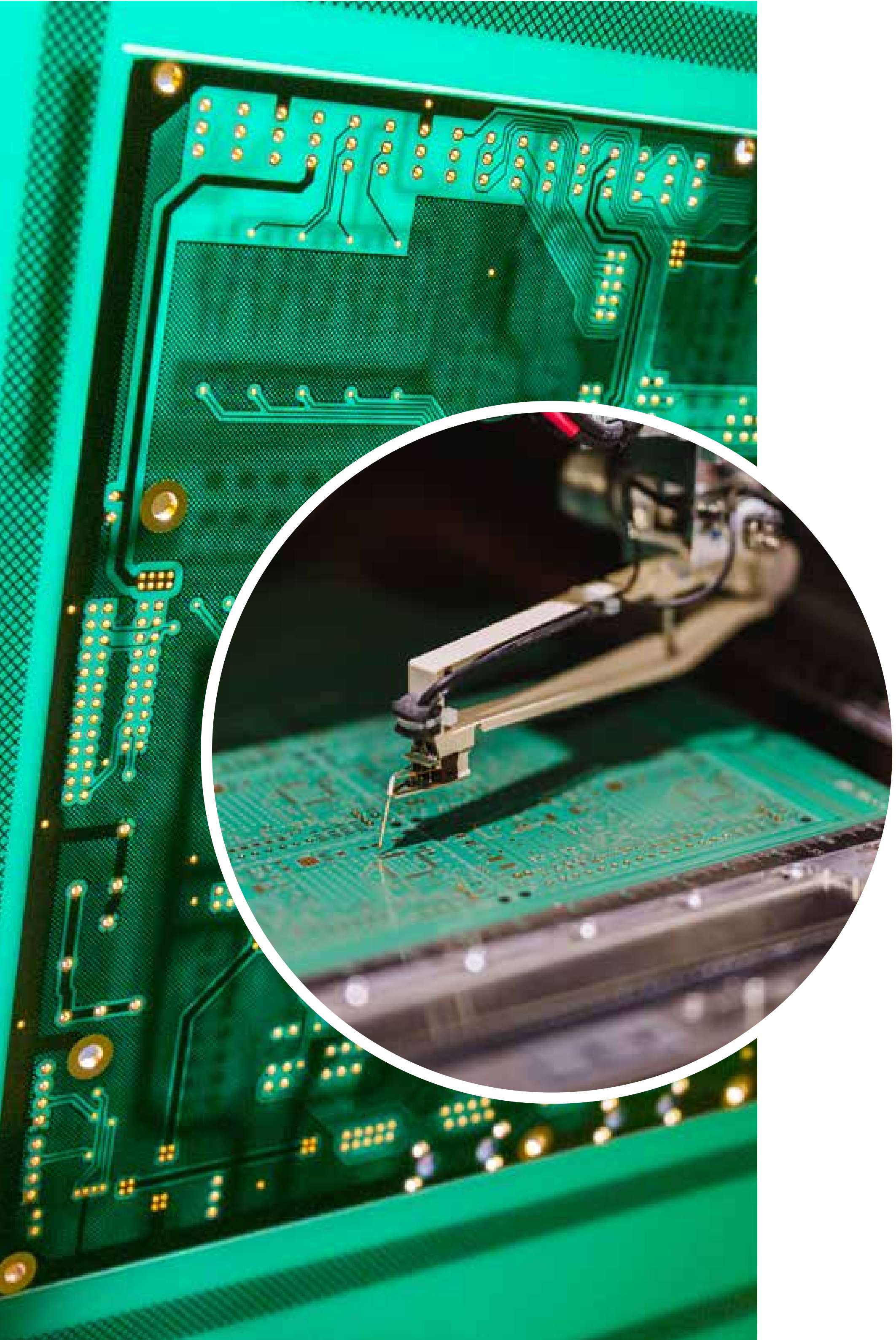
This rule applies both to the accepted file formats and the language accepted by the manufacturer. It is usually so that files in English or German are generally deemed acceptable. From the point of view of PCB manufacture inclusion of redundant or useless files in the documentation renders it difficult to analyze.



This applies in particular to a large groups for automatic assembly (BOM, Pick & Place), component catalogue notes, commissioning instructions and functional testing of the assembled packages, as well as documents informing about changes introduced in individual design revisions.

It is unacceptable to provide, in single documentation, various design revisions and several technological specifications, which may be mutually exclusive.

It is a good practice to attach a completed technological card published on the manufacturer's website.



Irregularities in the technological documentation

The production documentation should contain one technological specification (the so-called technological card), the entries of which allow for an unambiguous interpretation of the information. The typical specification shortcomings relate to incorrect notations of thicknesses of layers of copper, laminate and the descriptions of design of multilayer circuits.

1.

Thickness of copper layers

The thickness of the copper layers specified in the technological card is interpreted by PCB manufacturers as the target (final) one, unless the customer clearly states that it refers to the baseline copper, which is the copper foil thickness of the base laminate used for production.

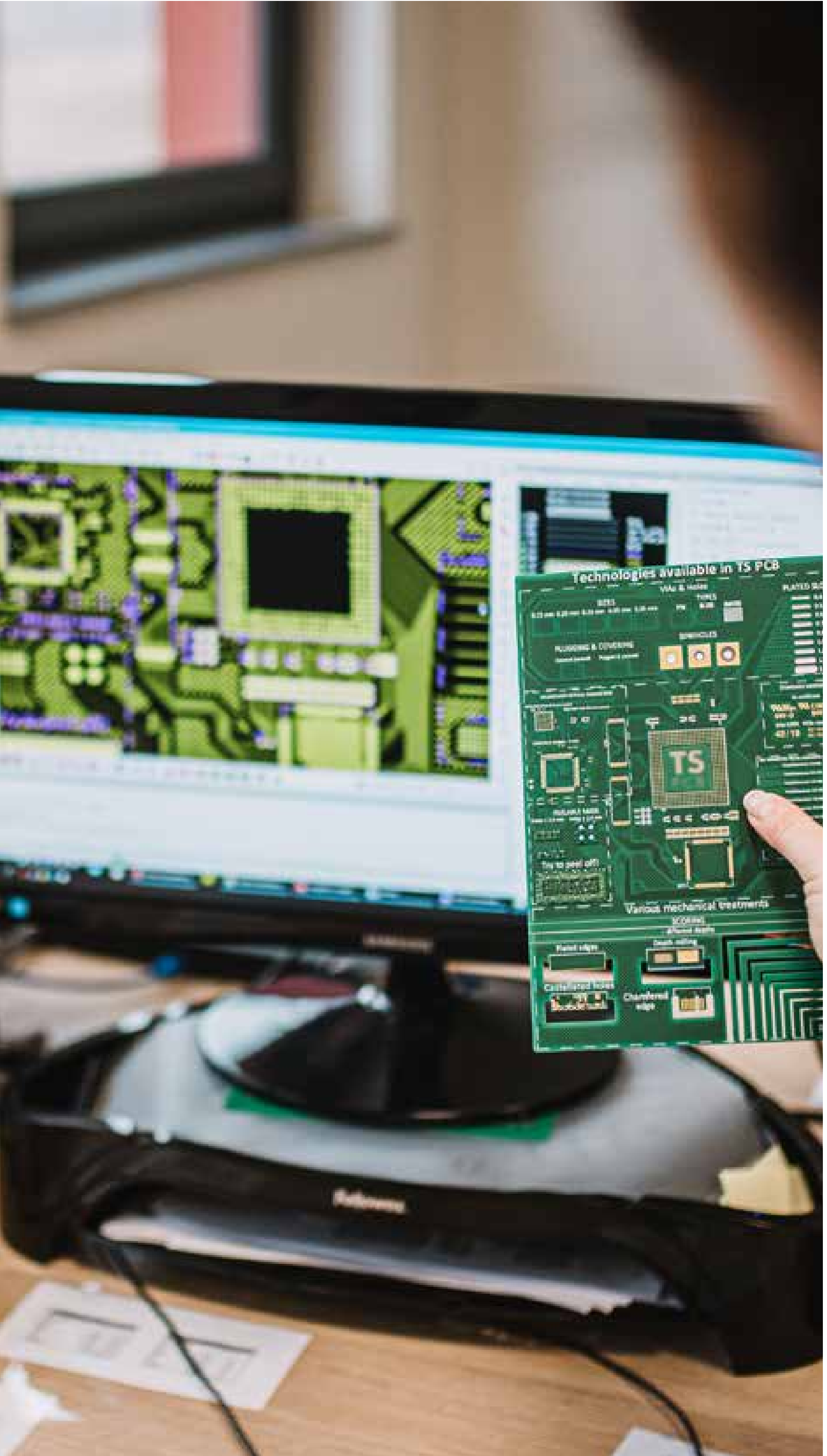
The final thickness of the copper layers is the final thickness of the layout in the manufactured circuit and in the case of circuits with metallization (double-sided and multilayer boards) it is the sum of the base and electroplated copper thickness of the laminate (25–30 μm range). This means that for manufacture of double-sided circuits with a standard final copper thickness of 35 μm, a base laminate with 18 μm copper is used and that it is not possible to obtain a final copper thickness of 18 μm for this type of circuits.

2.

Laminate thicknesses

Similarly as for layout, the concepts of base and final thickness are also used for laminates.

The base thickness is the initial thickness of the laminate used for production, which for single- and double-sided circuits, with the exception of multilayer circuit cores, includes the thickness of copper foil. The final thickness of the laminate is increased by the thickness of the following coatings: electroplated copper, surface finish (tin, gold and nickel) and solder masks.

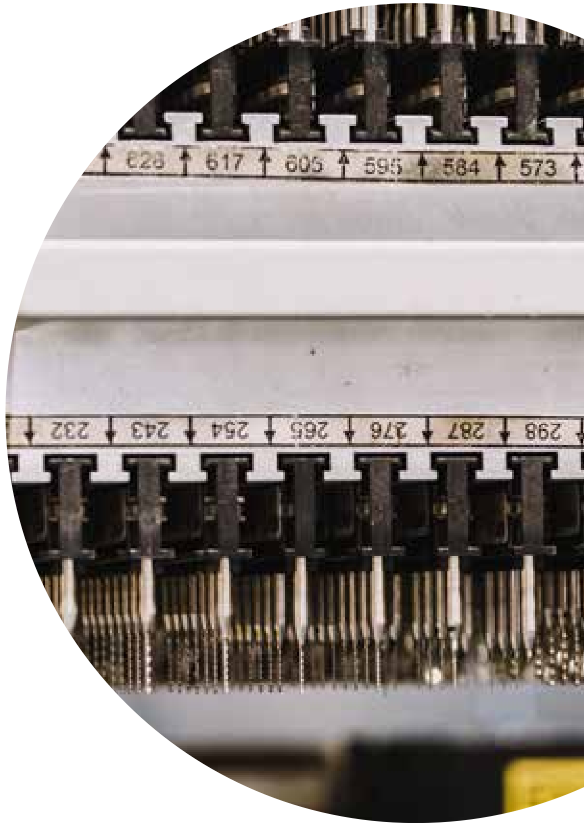


In the event that the final thickness the laminate is non-critical it is the best option to use the baseline thickness of laminate and its tolerance in the card.

Design of multilayer circuits

The parameters for the design of the multilayer circuit, i.e. the cross-section informing about the layout and type of respective layers, should take into account the technological possibilities and materials available to the PCB manufacturer already at the design stage. The use of specific cores and prepregs (pre-impregnated materials – a mixture of glass fiber and resin) and their atypical mutual arrangement may prevent the production of such circuit.

Whenever possible, it is worth designing structures symmetrical to the center of the cross-section, which will allow to obtain similar surface tensions on both sides of the laminate. Sometimes it proves impossible to design a symmetrical structure, e.g. due to the specific impedance of the layout, which may contribute to warping of ready-made circuits during their soldering on the assembly line. This phenomenon is due to the different surface stresses of the laminate due to their asymmetrical design.



Another design issue is the number and types of prepregs used. The majority of PCB manufacturers recommends using at least two prepregs separating adjacent conductive layers, as the use of a single greatly increases the risk of delamination and uncovering tracks. On the other side, the sole limit of the maximum number of prepregs is the thickness of the laminate after its pressing.

In the case of multilayer circuits with blind holes, their diameter depends on the permissible drilling depth and, consequently, the possible number of layers to be joined. The reduction of size of blind holes is facilitated by the lowest possible distance between the joined layers.

Another shortcoming in the multilayer board documentation is the **lack of information on the order of the internal layers**. Such information can be provided by the appropriate naming of layout files, numbering of layers on the layout itself or a written description in the specification.

TS PCB

Customer
Shaped Service

Manufacturer of printed circuits
with over 35 years of experience.

www.tspcb.pl
office@tspcb.pl
+48 58 340 42 54