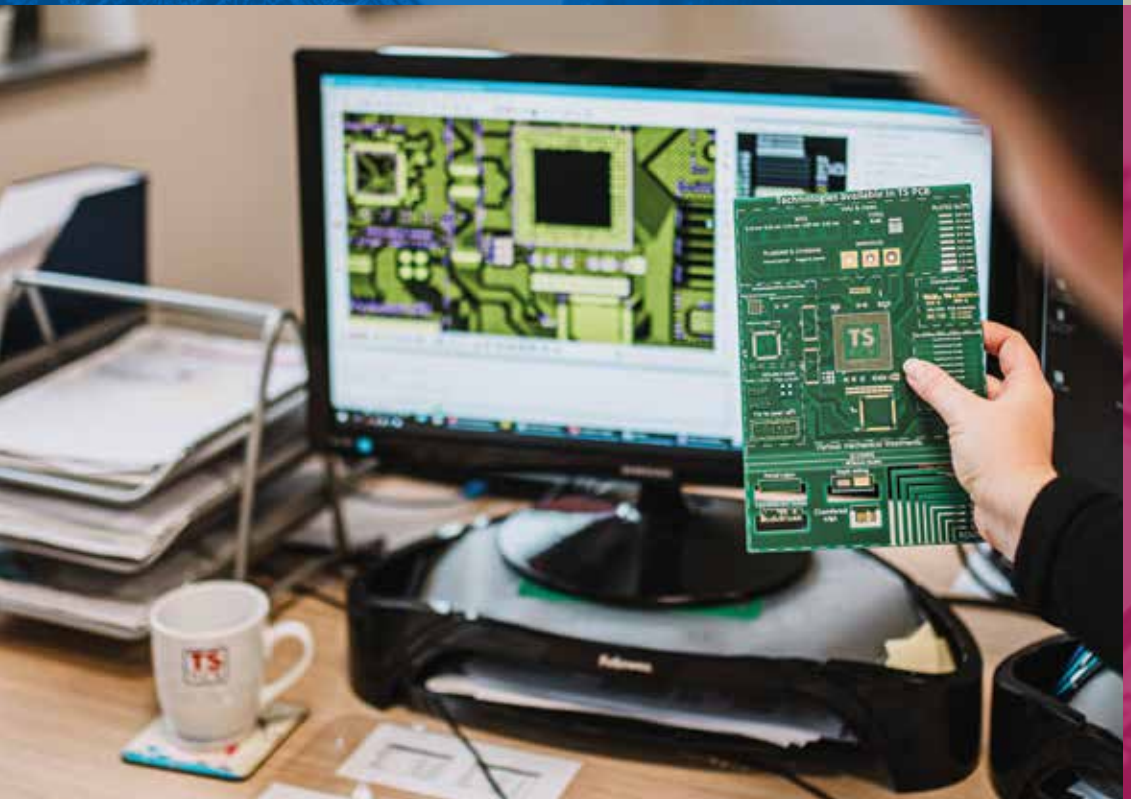
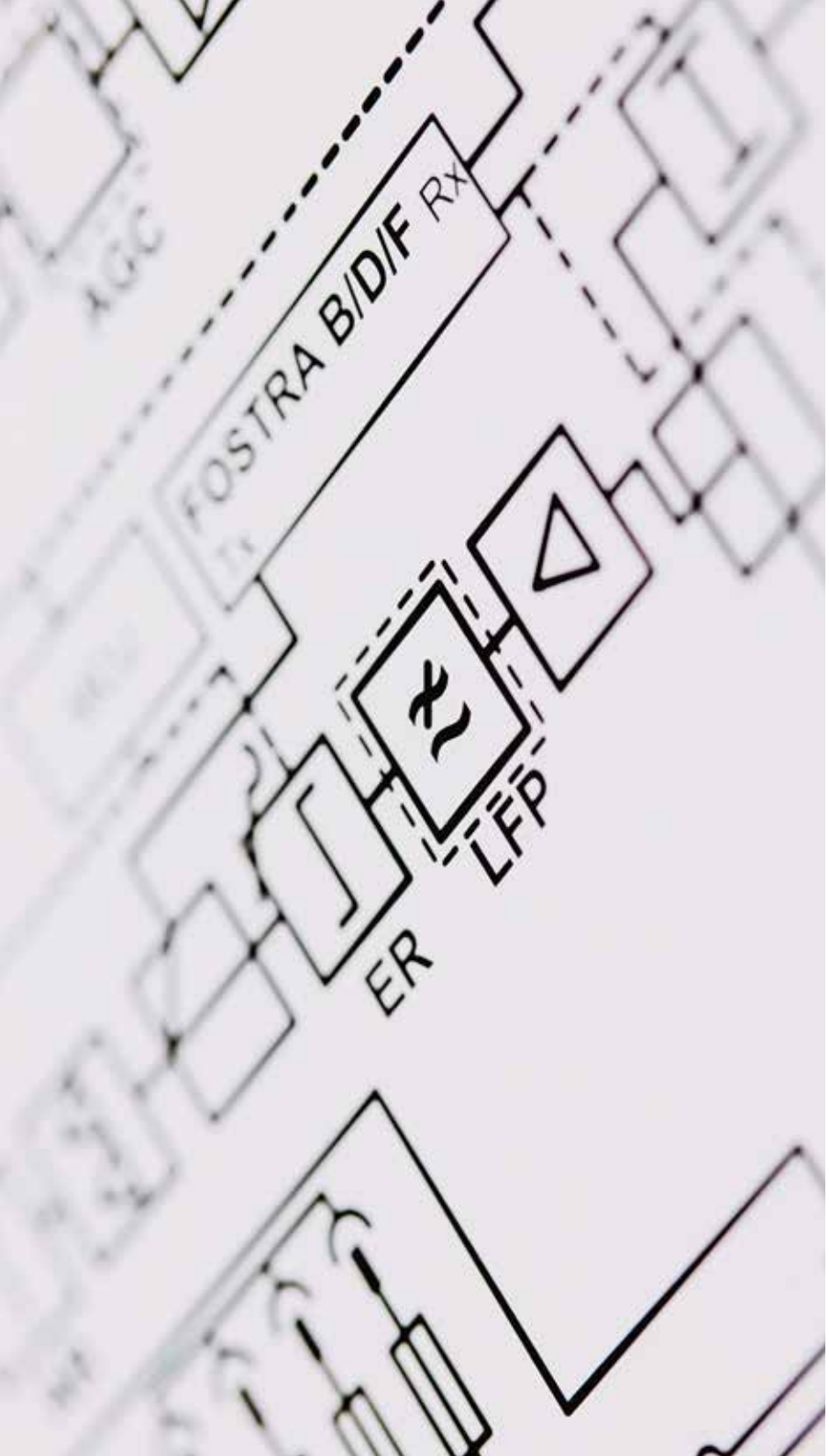
How to pay less for PCBs?



Methods for optimizing prices of PCBs

TS PCB

tspcb.pl



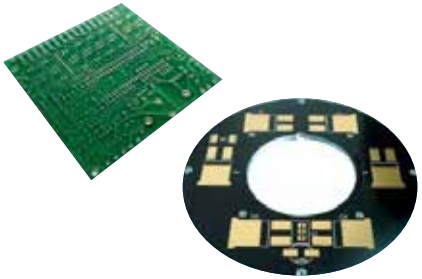
6 most significant technological parameters of PCBs that prove decisive for their price

- Type of material
- Through holes and surface finish
- Laminate thickness
- Number of layers
- Solder mask color
- Mechanical treatment

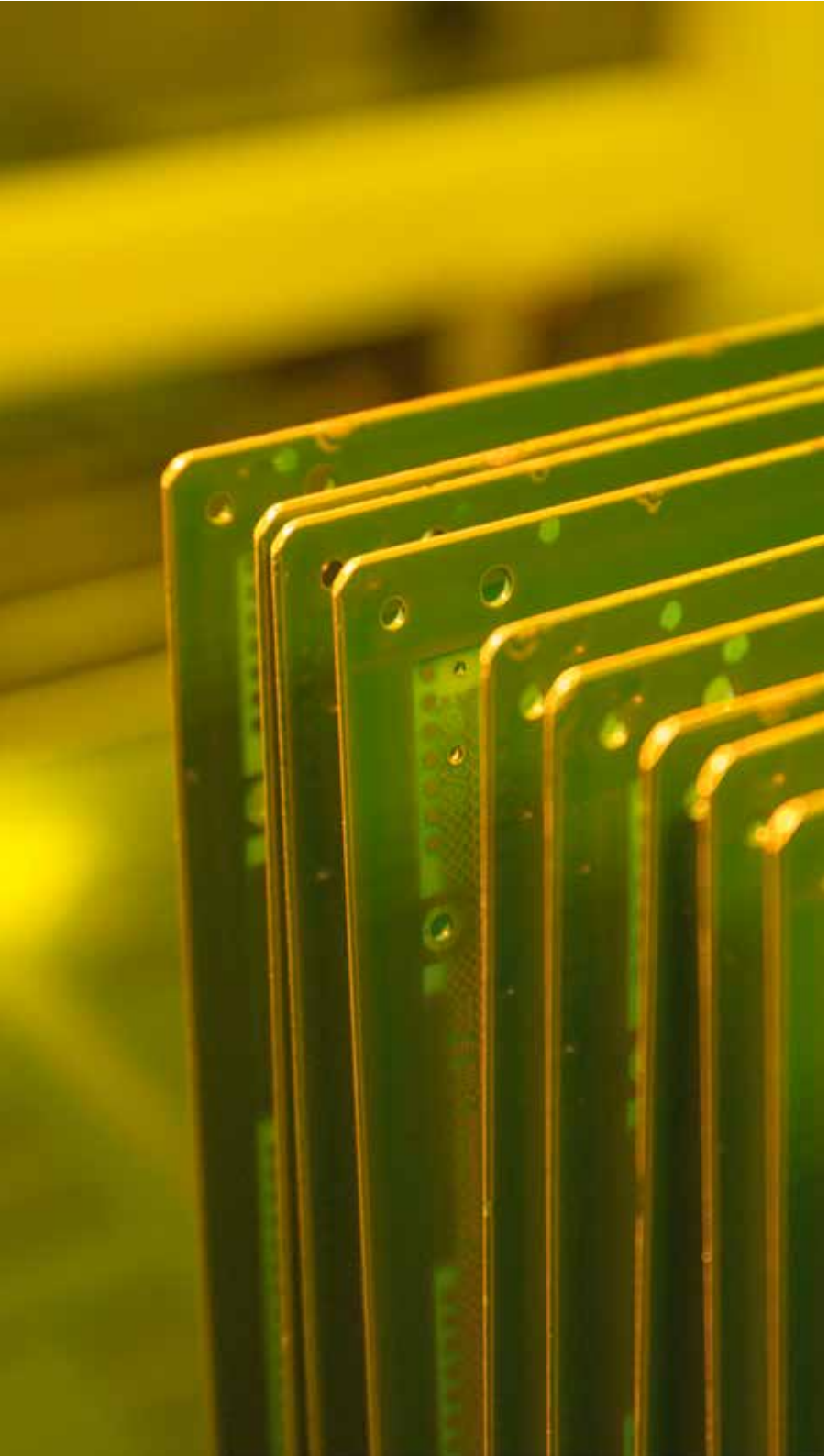
Type of material

The most popular
and the cheapest **laminate** is the **FR4**

Its universal electrical parameters, including the dielectric constant type 4.5, loss tangent type 0.02 and thermal characteristics - thermal conductivity of approx. 0.4 W/mK prove sufficient for most applications.



Rogers microwave laminates, which are characterized by a low loss tangent, are intended for **high frequency circuits**. However, their application is not always necessary. For systems operating at moderate speeds with clock speeds up to single GHz range, a cheaper solution will be to use the laminates from the FR408 (Isola) family with a loss tangent of 0.009 at 1 GHz.



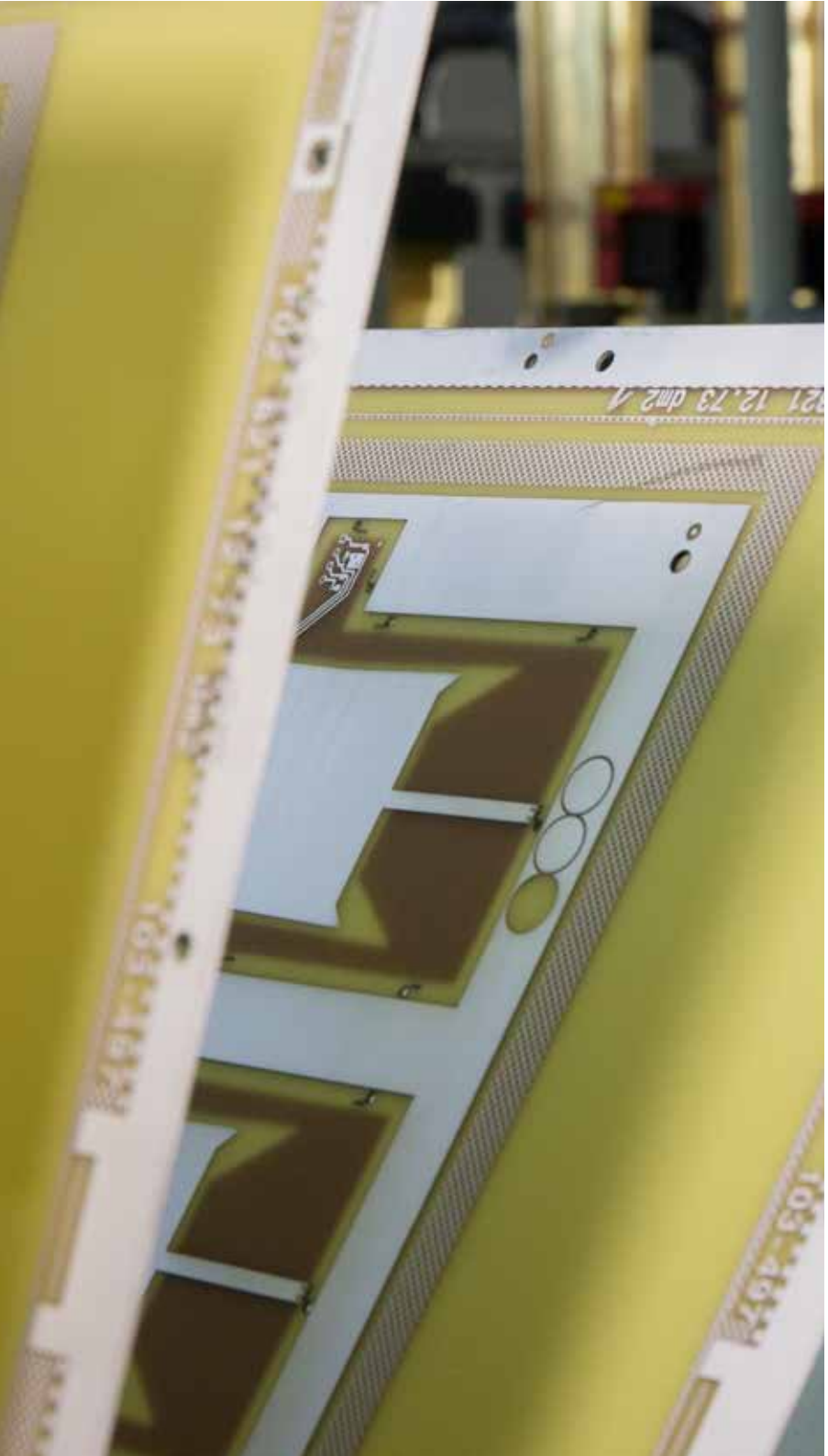
Through holes and surface finish

The cheapest option, in comparison for the use of blind or buried holes, is the execution of all holes as PTH through holes (Plated-Trough Holes).

The standard surface finish is the HAL (Hot Air Leveling) which uses a lead-free alloy to cover the pads.

However, HAL is not always permissible. Usually, more complicated circuits with minimum **DRC parameters** and those with **BGA (Ball Grid Array) housed chips** must be ENIG plated - here the price most often depends on the surface area of such finish.

In order not to unnecessarily increase the price for the plating, it is recommended to prepare most of the holes in the design as covered with solder mask and leave, as unmasked, only those for which e.g. measurement access is necessary for testing the functional package.



Laminate thickness

The factor determining the price of the circuit is the thickness of the laminate and copper foil, the standard values of which are 1.00/1.55 mm and 18 µm.

While the production of thinner circuits, especially single- and double-sided, does not increase the production costs, as much as the production of circuits on **thicker laminates (2.4/3.2 mm)**, as well as with thicker **copper foils (of 70/140 µm and more)**, it is more complex, which translates into their price.

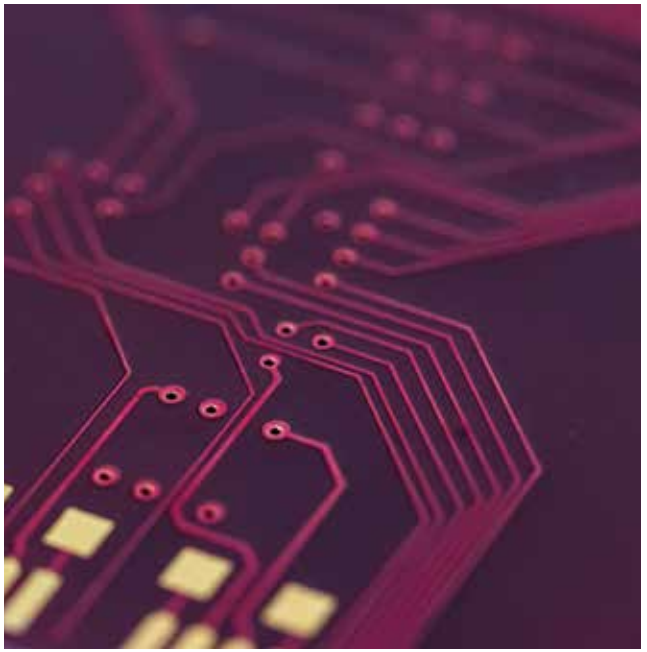
To get the lowest price one should:

- use standard thicknesses of laminates and copper foils,
- try to increase the width of the paths to maintain the permissible current density without the need to increase the thickness of the copper,
- unmask the tracks and subject them to the HAL process, which allows for an increase in thickness of about 20-40 um while maintaining the standard thickness of copper.

Number of layers

The number of copper layers has significant impact on price of PCBs.
Tangible benefits can be obtained especially by reducing the number of circuit layers in multi-layer circuits, e.g. from six to four.

To make the network more compact, the layout should be redesigned to make it fit the minimum permissible DRC parameters (Design Rule Check) for circuit technology. **Blind and buried** holes can also be used. A 4-layer circuit containing this type of through holes will still be cheaper than a 6-layer one without such holes.

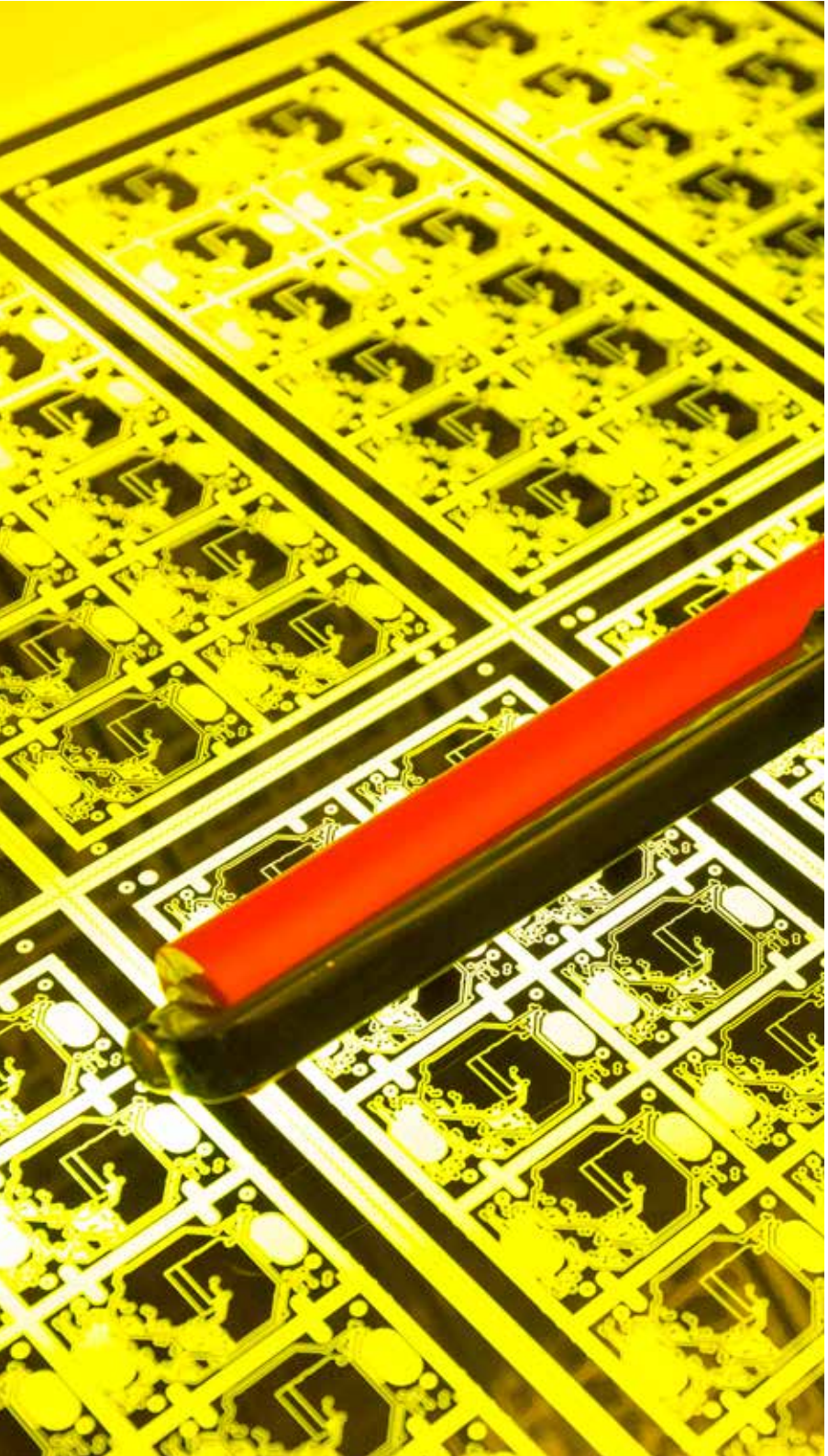


Solder mask color

Many PCB manufacturers offer a wide range of mask colors.

The cheapest solution is to use the green mask, which is applied automatically using the curtain method. This is a good choice especially when the color of the mask is irrelevant from the point of view of the design.

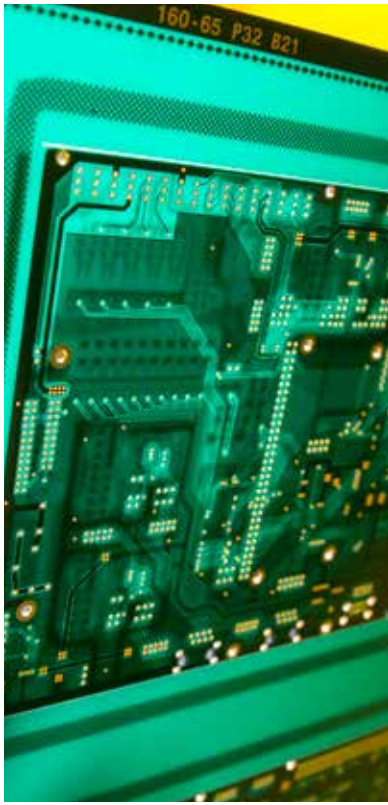
Other colors are applied by screen printing (semi-automatic), which is more timeconsuming and therefore more expensive.



Mechanical treatment

The **milling** operation allows to obtain almost any shape of printed circuit boards while maintaining dimensional tolerance on the level of ± 0.05 mm.

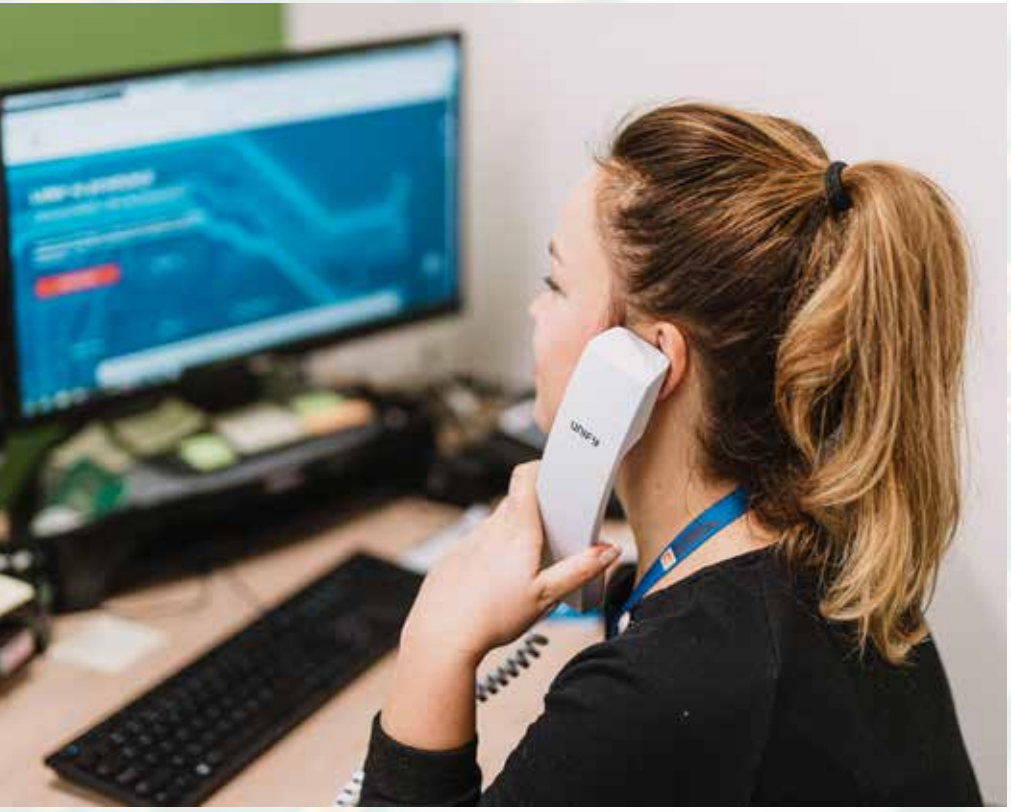
The time consumed by this operation and its impact on the price of the circumference increase with the decrease in the diameter of the tools used in the process below the one adopted as the standard - i.e. the tool with a diameter of 2 mm. This is mainly due to the necessity to lower the cutting speed for small tools. **Using tools larger than 2 mm in diameter is more advantageous as the milling can be performed at a higher speed.**



Scoring is a simpler and cheaper type of machining.

However, it has its limitations:

- its dimensional tolerance is worse than for milling, reaching up to ± 0.3 mm,
- triangular profile of the edge of the PCB,
- laminate thickness - panels on a laminate with a thickness of less than 0.36 mm must be made as milled.



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