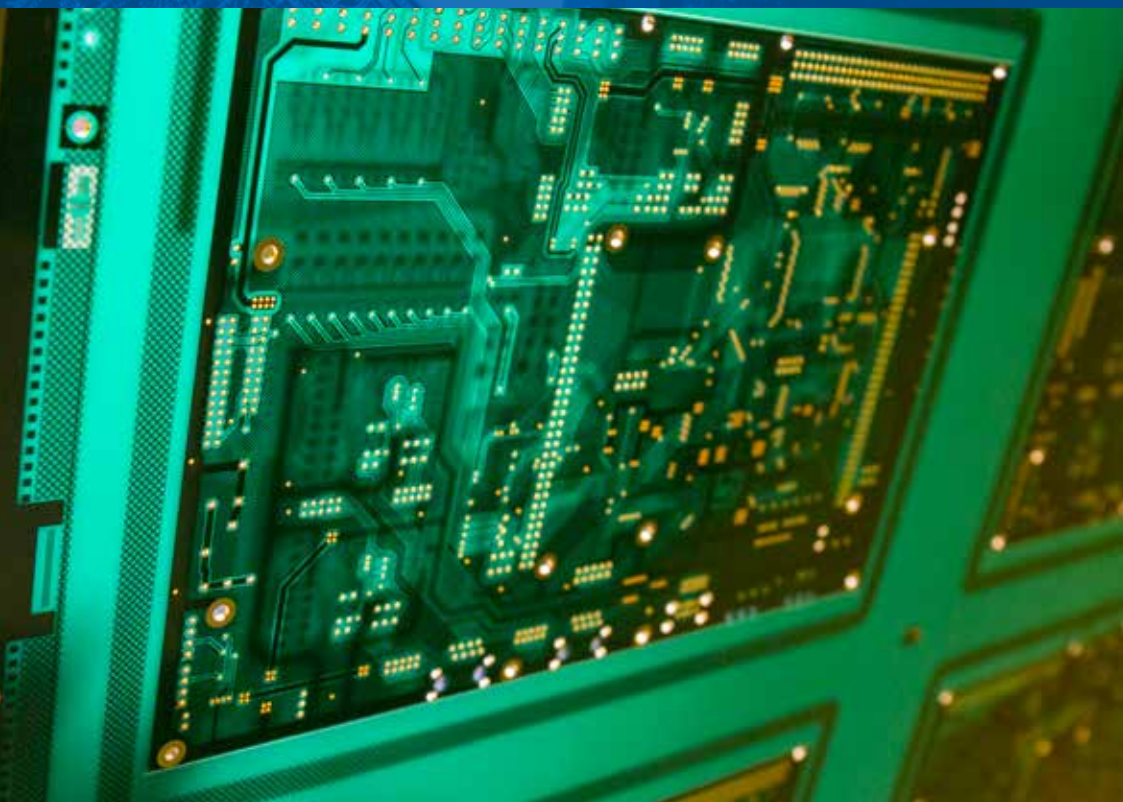
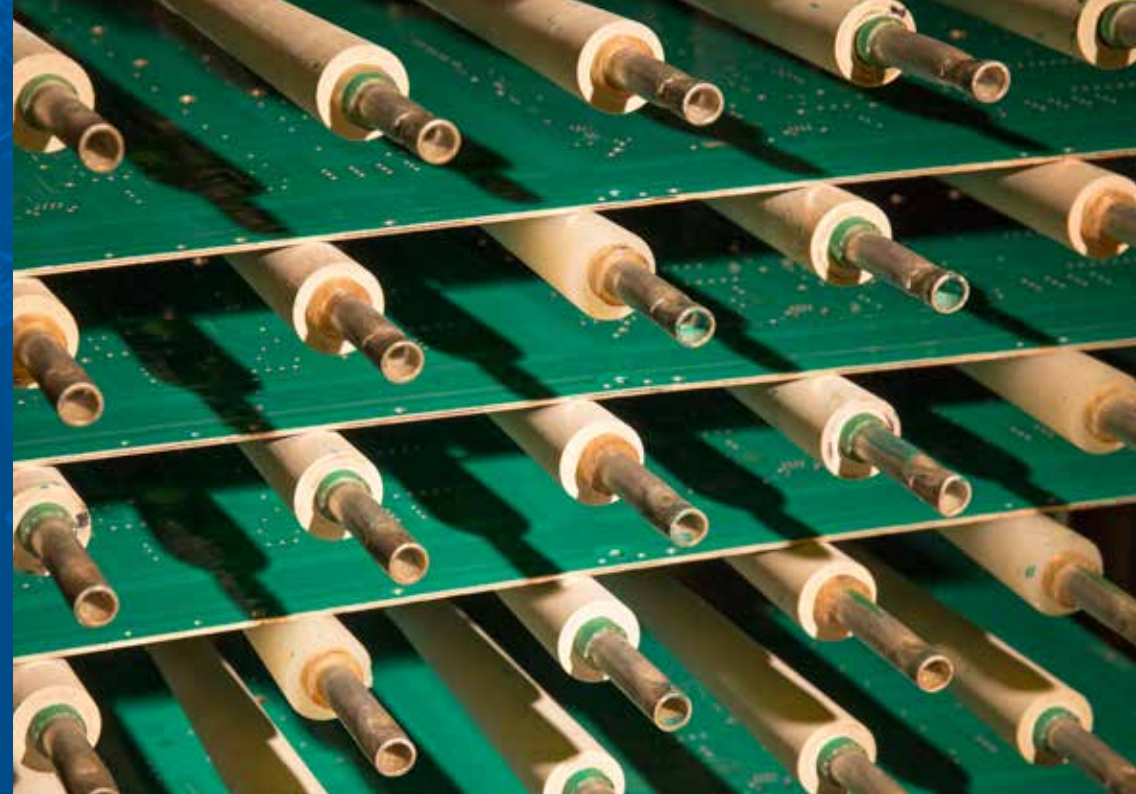


How to pay less for PCBs?



Documentation optimization



tspcb.pl



5

The first stage of contract implementation is the elaboration of the PCB design documentation. Precise and clear documentation guarantees that the circuits will be in line with the design assumptions, as any inaccuracies extend the time required to prepare the circuits for production. Clarifying doubts requires additional consultations, both on the part of the contractor and the customer.

File formats

DRC parameters

Preparation of the outline and silkscreen layers

Mechanical treatment

Preparation and naming of layer files

File formats

The **Gerber** format is the standard for documentation of **PCB manufacturing process**.

It allows to save the image of design layers: layout, masks, descriptions in vector format.

It has two versions: the older Standard Gerber (RS-274D and the modern Extended Gerber (RS-274X/ RS-274X2).

The Excellon format, version 2, is most often used to save the machining parameters (drilling, milling). A complete program for CNC drilling or milling operations should include:

- definitions of the diameters of applied tools,
- information about the coordinate format,
- information on the measurement units used.

The most popular **CAD PCB**

PCB software (Altium Designer, Eagle, Protel) allows us to export project design to Gerber and Excellon formats.

Most PCB manufacturers prefer the following formats:

- Gerber RS-274X/ RS-274X2,
- Excellon2,
- RS-274D,
- Dynamic Process Format *.dpf,
- GraphiCode GWK,
- ODB++,
- BRD (Eagle).

Characteristics of correct documentation

Correctly prepared designs are those containing information that is unambiguous and consistent with the technological capabilities of the PCB manufacturer.

DRC



Areas that do not meet the permissible DRC parameters

Exceeding the permissible DRC (Design Rule Check) parameters is a design flaw that may prevent the circuit from being feasible. They determine the technological capabilities of the manufacturer in terms of the minimum allowable widths of the layout elements (tracks, inscriptions), the distance between them and the sizes of the rings surrounding the through holes.

Most **CAD PCB software suites** allow you to preset and follow DRC rules so that the elaborated design remains in line with the manufacturer's capabilities.

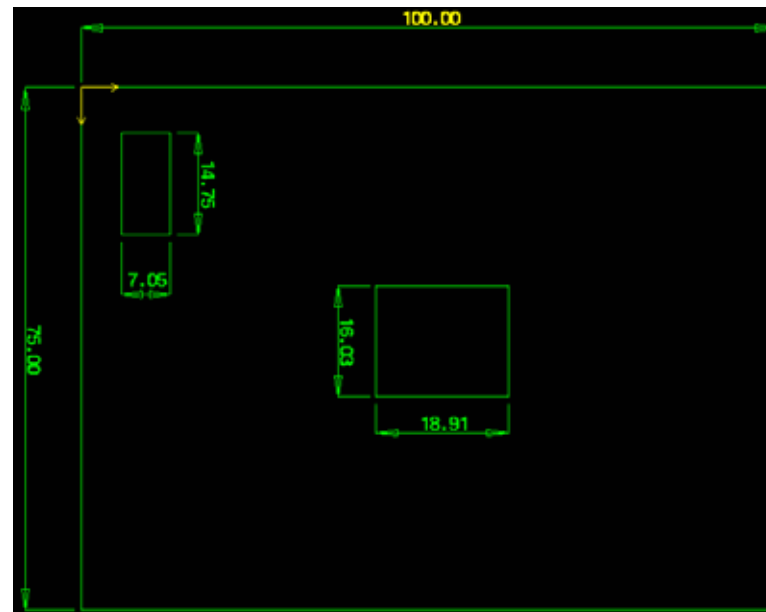
When designing the layout, we must remember that **narrow indentations on the surface of layout** are particularly tricky in the manufacturing process. These areas should be filled, because the principle of minimum permissible distances also applies to the layout elements belonging to the same network. DRC parameters also apply to inscriptions placed in copper layers. Characters that are too small can be etched and eventually become unreadable.

Outline

The outline of the circuit should be placed on a separate layer, in the form of a closed outline and without offsets in relation to the other layers.

It should include the shapes and locations of all the cuts and slots that need to be made in the circuit.

The circuit dimensioning, can be placed on the layout layer, including its tolerances, as long as they differ from the typical tolerance of the manufacturer.



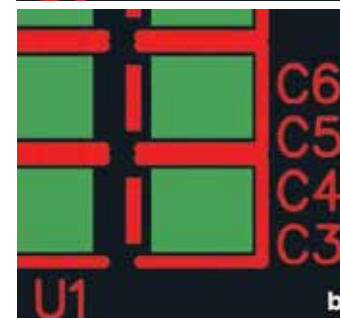
Silkscreen layers

In order to ensure good legibility of silkscreen layers, it is necessary to maintain the appropriate size of characters and the thickness of the lines.

Optimal height of the characters is no less than 1 mm, and the minimum thickness of the line is 0.1 mm.



In the case of silkscreen made with a line that is disproportionately thick to the height of the characters, reducing the thickness of the line improves the legibility and clarity of the silkscreen. By default PCB manufacturers remove silkscreen from pads and other areas of the circuit not covered with a mask, because paint left on the soldered elements prevents the correct assembly of the circuit. In order to reduce the loss of silkscreen caused by clipping, the designer should not place markings on the soldering locations and holes and keep at least 0.1 mm distance of the silkscreen lines from the pads exposed on the anti-soldering mask and the holes.



Silkscreen line thickness and description quality:

- a) a silkscreen with 0.25 mm line,
- b) a silkscreen with 0.1 mm line,
- c) view of the silkscreen cut by the SMD pad

Mechanical treatment

In a circuit board design, all holes and slots can be placed in a single drilling program. However, it is important to clearly designate their types, e.g. by describing them with comments in the drilling program or in an additional document.

Nevertheless it is recommended to divide the holes and slots into separate, appropriately named drilling programs. It is also good practice to provide the drill map in a separate layer for the design.

Things to remember:

- the diameters provided are treated as target, unless otherwise indicated in the documentation,
- in the case of blind and buried vias, their drilling programs should be placed in separate files, separately for each depth. In addition, it is necessary to indicate directly which layers are to be joined.
- the distance of layout from the edges and board cutouts must be kept at the appropriate values, as mechanical treatment is performed with preset tolerance. In this way, damage to the layout elements that are too close to the edge and possible discharges between the layers can be prevented.
- milled areas should be clearly illustrated on a separate drawing or on a layer with an outline or a separate layer, as well as defined on which side of the circuit and to what depth they are to be made. The presence of metallized edges should be determined in a similar way.



Preparation and naming of layer files

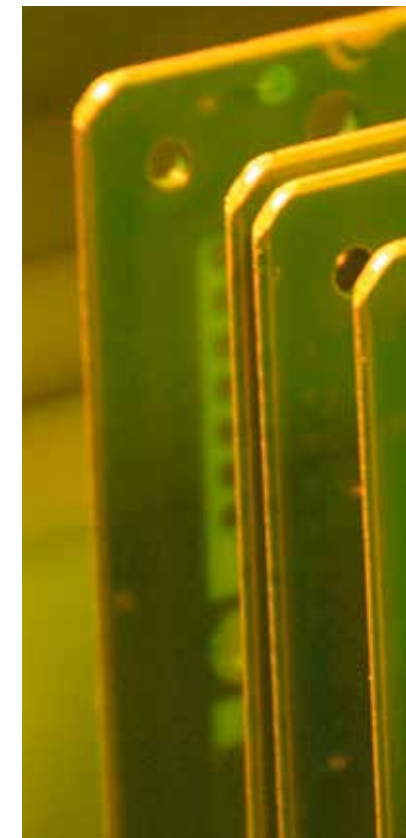
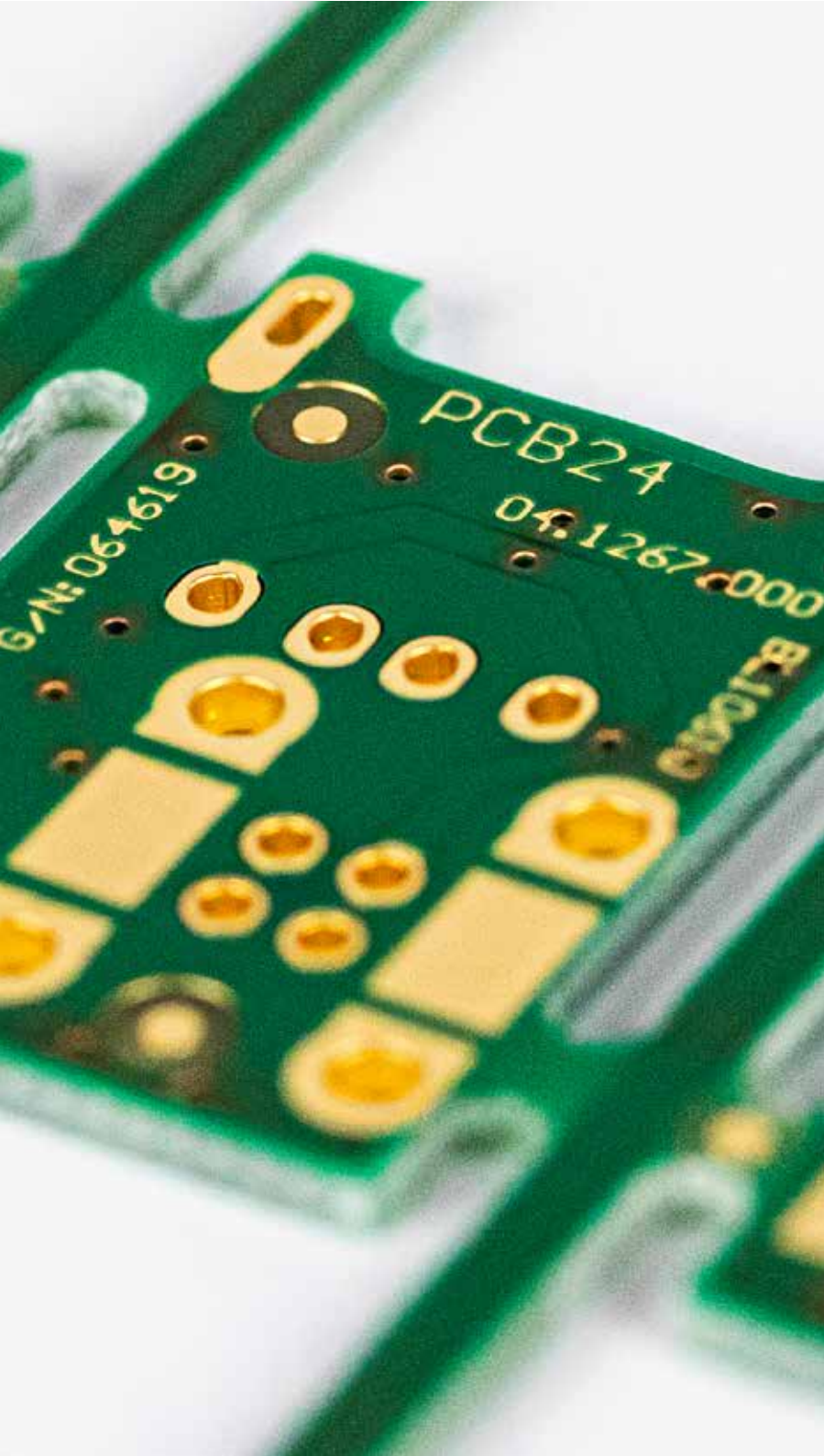
Images of individual layers should be generated in the same scale and without mutual shifts. The common standard is to place the layers in the view from the top side of the circuit. In simple terms, it means that all the inscriptions placed on the layers from the side of the elements should be legible, while the text placed on the soldering side will be visible in a mirror image.

It is recommended that you use such layer file names and/or extensions so that they can be interpreted unambiguously.

File nomenclature is usually defined in the **CAD PCB software**.

Most of this type of suites, apart from files in Gerber and Excellon formats, generates an additional text file (report) with a list of exported files and a description of the format applied for saving coordinates and units.

In the case of multi-layer circuits, it is necessary to provide information about the order in which the internal layers are arranged. Their order can be specified by numbering directly on the layers or the appropriate file naming (e.g. *.L1, *.L2, *.L3, *.L4, etc.).





Customer Shaped Service

Manufacturer of printed circuits
with over 35 years of experience

www.tspcb.pl

office@tspcb.pl

+48 58 340 42 54