

Nanotech Elektronik Sp. z o.o., Aleje Jerozolimskie 214, 02-486, Warsaw,
Poland tel.: +48 338 338 338, +48 506 620 833, +48 510 980 835
e-mail: office@nanotech-elektronik.com, <http://www.nanotech-elektronik.com>

Printed Circuit Board Services



Nanotech Elektronik is a supplier of printed circuit boards (PCB) of any type and complexity level.

We deliver the printed circuit boards of any kind:

- Multilayer (up to 28 layers) PCB
- HDI PCB (with laser microvias)
- Flexible PCB (polyimide, PET)
- Rigid-Flex PCB
- Metal core PCB (aluminum, copper)
- RF/microwave PCB (Rogers, Arlon etc.)

For manufacturing the printed circuit boards we use the FR4 laminates from the world leading suppliers such as Nanya, Isola, KB, Shengui.

We also have in stock another materials such as Polyimide, PET, Rogers, Arlon, PTFE to produce any specific PCB products.

We offer a wide selection of surface finishes: HASL RoHS, HASL SnPb, Immersion Gold (ENIG), Gold Fingers, Immersion Tin, OSP etc.

We operate in accordance with the IPC-A-600 (Acceptability of Printed Boards).

PCB Manufacturing Capabilities

Multilayer and HDI PCB

Parameter	Typical	Advanced
Number of layers	4-16	4-28
Minimum trace width, mm	0,1	0,075
Minimum spacing, mm	0,1 / 0,075	0,075 / 0,075
Trace to board edge distance (outer/inner layers), mm	0,5 / 0,5 (V-cut)	0,25 / 0,4 (routing)
Minimum laser hole size, mm	0,1	0,075
Minimum drill hole size, mm	0,2	0,15
Minimum annular ring (outer/inner layers), mm	0,15 / 0,1	0,127 / 0,1
Aspect ratio	1:8	1:12
Via-in-Pad technology	yes	yes
Buried (hidden) holes	yes	yes
Blind holes	yes	yes
Stacked and staggered microvias	yes	yes
Solder mask opening/ expansion, mm	0,05	0,05
Solder bridge, mm	0,1	0,1
Minimum width of marking line (silkscreen), mm	0,15	0,15
Minimum height of marking text (silkscreen), mm	1	0,8

Flexible PCB

Parameter	Typical	Advanced
Number of layers	1-2	4
Material	Polyimide, PET	
Minimum trace width, mm	0,15	0,1
Minimum spacing, mm	0,15	0,1
Trace to board edge distance, mm	0,5	0,25
Minimum drill hole size, mm	0,3	0,2
Coverlay opening/expansion, mm	0,15	0,15
Possibility of manufacturing a stiffener for flex PCB	Yes (Polyimide or FR4) ⁶	

Rigid-flex PCB

Parameter	Typical	Advanced
Number of layers	4-16	4-28
Minimum trace width, mm	0,1	0,075
Minimum spacing, mm	0,1	0,075
Trace to board edge distance (outer/inner layers), mm	0,5 / 0,5 (V-cut)	0,25 / 0,4 (routing)
Minimum drill hole size, mm	0,25	0,2
Minimum annular ring (outer/inner layers), mm	0,15 / 0,1	0,127 / 0,1
Via-in-Pad technology	yes	yes
Buried (hidden) holes (rigid part)	yes	yes
Blind holes (rigid part)	yes	yes
Solder mask (coverlay) opening/ expansion, mm	0,05 / 0,15	0,05 / 0,15
Solder bridge, mm	0,1 / 0,2	0,1 / 0,2
Minimum width of marking line (silkscreen), mm	0,15	0,15
Minimum height of marking text (silkscreen), mm	1	0,8
Possibility of manufacturing a stiffener for flex PCB	Yes (Polyimide or FR4) ⁶	

Aluminum core PCB

Parameter	Typical	Advanced
Number of layers	1-2	1-4
Board Thickness, mm	0,5 – 3,2	
Copper thickness, μm	35	
Dielectric thickness, μm	50, 75, 100, 150	
Thermal conductivity, W/(m·K)	1-4	
Dielectric strength, kV	2-6	
Maximum size, mm	550,0 x 950,0	
Minimum trace width, mm	0,2	0,15
Minimum spacing, mm	0,2	0,15
Trace to board edge distance, mm	0,5	0,25
Minimum drill hole size, mm	0,9	0,6
Solder bridge, mm	0,1	0,05

¹Final copper thickness depends on clearances (see “Minimum clearances for different copper thicknesses” table)

²Other materials upon request

³Other colors upon request

⁴Solder bridge is 0,15 mm

⁵Upon request

⁶Stiffener thickness upon request

Notes:

Standard copper via wall thickness is up to 20 μm .

Gold thickness in IG coating — 0,05-0,11 μm ,

Hard Gold (GoldFingers) — 0,07-1,27 μm

Minimum clearances for different copper thicknesses

Outer layers					
Finished copper thickness	35um	70um	105um	140um	210um
Minimum trace width	0,1mm	0,20mm	0,23mm	0,30mm	0,60mm
Minimum clearance	0,1mm	0,20mm	0,24mm	0,35mm	0,70mm
Inner layers					
Finished copper thickness	35um	70um	105um	140um	210um
Minimum trace width	0,1mm	0,20mm	0,27mm	0,34mm	0,60mm
Minimum clearance	0,1mm	0,20mm	0,30mm	0,45mm	0,85mm

How to get a quote and place an order

To prepare for you a quote for printed circuit boards we will require:

- ① PCB files (Gerber or CAD files)
- ② Description of PCB (quantity, base material, final PCB thickness, copper thickness, surface finishing etc).

When you send us this data, it will first be analyzed by our technical department to accurately understand all your requirements. Next, the price offer will be calculated, which will be sent to you along with the delivery time and our notes.

When you place an order for printed circuit boards, our technical department will perform thorough analysis of entire project documentation (including DRC - Design Rule Checking, DFM - Design for Manufacturability, DFA – Design for Assembly control) at the stages of pre-production.

Please note that our engineers are ready to advise you in case of any questions that arise during the PCB layout design phase (e.g. stack-up planning, selection of appropriate materials etc).

Requests for quote or orders, please send to our email address: office@nanotech-elektronik.com

Quality assurance

During, as well as after the production of printed circuit boards, they are subjected to many quality tests, the results of which we store in a database for each batch of products.

For example the QA test reports include such an information as:

① Final inspection report

In this part of the report you will find the following information: materials used for PCB fabrication, copper and dielectric layer thickness, solder mask thickness, final PCB and panel dimensions and their tolerances.

② Cross-section report

Cross-sectioning (sometimes called micro-sectioning) is a metallographic technique used to characterize materials, perform a failure analysis and expose an internal structure of PCB. Also, we can fabricate a few extra circuit boards called PCB Test Coupons, which can be tested to evaluate the performance of the boards.

③ Electrical test report

All printed circuit boards are electrically tested for proper interconnections (we use two methods - Flying Probe & Bed-of-Nails).

④ Impedance control test report

Impedance control includes measuring the impedance of certain traces on PCB and checking if they are within the specified limits.

⑤ Solderability test report

The solderability test checks how effectively the exposed pads are wetted with molten solder. This is one of the most important PCB testing methods.

Packing and shipping

After the final inspection is completed, the finished products are packed in vacuume packaging and sent to the customer via express delivery service (DHL, DPD, UPS or other).

We use reliable and environmentally safe packaging materials to protect the printed circuit boards and to secure the delivery to our customers.

Do not hesitate to contact us if you need additional special packaging options for your products.



Get your technical guidance



Click the images above to get for free our technical guides and tutorials