ELECTRONICS
DESIGN AND MANUFACTURING
SERIES

GUIDE TO
PRINTED CIRCUIT BOARDS

Nanotech Elektronik is an EMS company with a wide range of services

- PCB Services
- SMT and THT assembly
- Electronic components
- BOM Services
- Prototypes
- Turnkey manufacturing

Our technological capabilities in the scope of assembly

Production and assembly of printed circuit boards	
Minimum order quantity	from 1 piece upwards
Maximum PCB size (X x Y)	automatic SMT assembly - 610 mm x 510 mm; THT assembly - no restrictions
Minimum PCB Size (X x Y)	automatic SMT assembly - 50 mm x 50 mm; THT assembly - no restrictions
SMD components assembly	
Component size range	from 0,4 mm x 0,2 mm (01005) to 45 mm x 100 mm
Component height (max)	15 mm
Types of components	Chips: 01005, 0201, 0402, 0603, 0805, 1206, 1210, 1812, 2010, 2225, 2512
	IC: PLCC18-PLCC84, LCC20-LCC84, SO, HSOP, SOJ18-SOJ44, MSOP8-MSOP10, SSOP8-SSOP64, HSOP20-HSOP44, TSSOP8-TSSOP80, TSOP28-TSOP56, TQFP32-TQFP176, LQFP32-LQFP256, QFP44-QFP304, CSP40-CSP56 (0,5), BGA46-BGA100 (0,75-0,8), LBGA48-LBGA280 (0,75-0,8), BGA81-BGA324 (1,0) up to LBGA1936 (1,0), BGA208 (1,27) up to LBGA1225 (1,27), BGA169 (1,5) up to LBGA400, CBGA121 - CBGA1089
Assembly accuracy (X, Y)	50µm for chips 01005, 0201, 0402
	75µm for chips > 0402, SOIC
	30µm for QFPs

Product quality is assured by a multi-level control system at every stage of the production cycle. The manufactured product will fully comply with the provided technical requirements and standards of the international association of electronics manufacturers (Institute of Printed Circuits - IPC).

Contents

Contacts

Feel free to contact us if you have further questions. You will always obtain comprehensive information both in the scope of designing and producing printed circuit boards, as well as practical information specifying the product manufacture and delivery time. We are always happy to share our knowledge and experience, in addition to taking care of the highest quality the projects implemented by us, which can be confirmed by the line-up of our clients in the EU and worldwide.

We are always willing to prepare a detailed cost estimate for the production of printed circuit boards, purchase of electronic components, assembly works consisting in mounting components on PCBs and other additional works. Owing to this, you will be able to find out about the production cost of both the first prototype batch and the cost of serial production after sending us the technical documentation of the project.

You can also contact us by phone: **+48 338 338 338**
or write to the e-mail address: **office@nanotech-elektronik.com**
(we communicate in English, German and Polish).

Sincerely,
The Team of Nanotech Elektronik.

1. General information and tips for designing a PCB layout

When designing a printed circuit board layout, it is very important to understand the technological capabilities of the manufacturer who will fabricate it later.

Different manufacturers of printed circuit boards have their own technological capabilities, which may differ widely.

Before proceeding to the implementation of PCB layout, the designer must have a good understanding of the production capabilities at which it will be manufactured and be properly guided by them when designing PCB.

Usually, the technological capabilities of a particular manufacturer can be found on its website. In case of any questions or doubts, it is recommended to contact the technical department of the manufacturer to get comprehensive information directly from its specialists.

We draw attention to two important issues:

Most likely, many manufacturers will describe their technological capabilities as typical and advanced. In most cases, advanced parameters mean a higher cost of the manufactured printed circuit board. Therefore, if you could use typical technological parameters when designing your PCB, it is better to stick to them and avoid advanced ones.

Another point is that if you need to use advanced technological parameters in your design, it is better to use them where they are really required and keep typical technological parameters where there is no need for advanced ones.

As a result, you will get a cheaper and more reliable in terms of performance PCB, rather than one that is designed on the verge of technological limitations and has a higher price due to lower technological yield because of possible rejections at the quality control stages during fabrication.

It is an important to keep in mind the DFM (Design for Manufacturability) approach in your designs. There are a lot of technical articles about DFM principles, and unfortunately it is beyond the scope of this booklet to mention all of them here. Next, we give the most relevant tips based on our practical experience in preparing for production and fabrication of printed circuit boards:

- It is better not to place the vias in the soldering pads of the SMD components (except for micro vias made using laser - micro via / Via-in-pad). Otherwise, the solder paste will leak out through them during reflow process.
- Always check the width of the copper ring around the hole (annular ring). It should be according to the parameters specified by the manufacturer (see Chapter 2 - PCB manufacturing capabilities).
- In the project files, use the final diameters of the holes, not the diameters of the drills - each manufacturer has its own algorithm for calculating the compensation of the hole diameter during metallization.
- It is better to open mounting holes without metallization in the solder mask with a swell of at least 0,1 – 0,2 mm from the hole edge.
- We recommend to open soldering pads in solder mask with a swell of 0,1 mm. For components with a small raster – 0,05 mm (minimum mask bridge should not be smaller than 0,1 mm).
- Always provide information if vias need to be tented (hidden under the solder mask).
- Mind a minimum distance from topology elements (tracks, solder pads, copper polygons) to the edge of the printed circuit board. We recommend a minimum distance of 0,5 mm, and certainly not less than 0,3 mm.
- On the inner layers, it is worth to keep a minimum distance from the edge of the copper polygon to the edge of the board, not less than 0,3 mm. Otherwise, copper burrs will occur when the PCB outline is routed.
- Remember that the actual dimensions of the printed circuit board are set by the geometric center of the line that draws the outline of the PCB in your CAD program.
- If polygons are used, always remove "dead copper" - areas of copper not connected to any signals.
- If there are large copper polygons on the PCB, it is best not to fill them in solid but in the form of hatching (mesh or grid). The electrical properties of the polygon will be the same, but the board will be less willing to twisting (bending). We recommend that you evenly distribute large areas of copper on the PCB. All this is especially important for large-sized printed circuit boards.

- In addition, when filling in copper polygons in your CAD, we recommend not choosing too small primitives and grid spacing – this will increase the file size and make it difficult to process.
- Soldering pads of SMD components with small sizes (0402 and less) we recommend with rounded corners. This contributes to a better application of solder paste when printing it with a stencil.
- Make sure that the component designators are not on the soldering pads and open copper areas.
- Do not use a too small font size for designators (we recommend a minimum height of 1,0 mm and a minimum line width of 0,15 – 0,2 mm).
- To draw the topology of a printed circuit board, always use the standard layers assigned to it in your CAD. If the PCB design contains additional layers, their description and explanations should be attached to the design documentation.
- When designing a multilayer PCB, we recommend checking with the manufacturer the stack-ups they can produce. Sometimes we receive from customers a printed circuit board with an incorrect stack-up, which cannot be implemented even theoretically.
- The most important rule is that in case of any doubts about the technological parameters of the designed printed circuit board, always consult with the manufacturer.

Upon completing the PCB layout design, it is recommended to perform DRC (Design Rules Check) analysis in your CAD program. For the parameters for DRC analysis, you can use the manufacturer technological capabilities. Sometimes manufacturers provide ready-to-use .drc files that you can import into your CAD.

2. PCB manufacturing capabilities

Here we list our technological capabilities for different types of printed circuit boards.

2.1 Single / two-sided PCB

Single / two-sided PCB technological capabilities

Parameter	Typical	Advanced
Minimum trace width[1], mm	0,15	0,1
Minimum spacing, mm	0,15	0,1
Trace to board edge distance, mm	0,5 (V-cut)	0,25 (routing)
Minimum drill hole size, mm	0,3	0,2
Minimum annular ring, mm	0,2	0,15[3]
Aspect ratio	1:8	1:12
Via-in-Pad technology	yes	yes
Solder mask opening/ expansion, mm	0,1	0,05
Solder bridge, mm	0,2	0,1
Minimum width of marking line (silkscreen), mm	0,15	0,15
Minimum height of marking text (silkscreen), mm	1	0,8

2.2 Multilayer PCB

Multilayer PCB technological capabilities

Parameter	Typical	Advanced
Number of layers	4-14	4-28
Minimum trace width[1], 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 / 0,3 (routing)
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
Buried (hidden) holes	yes	yes

Blind holes	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

2.3 HDI PCB

HDI PCB technological capabilities

Parameter	**Typical**	**Advanced**
Number of layers	4-16	4-28
Minimum trace width[1], 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 / 0,3 (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

2.4 Flexible PCB

Flexible PCB technological capabilities

Parameter	Typical	Advanced
Number of layers	1-2	4
Material	Polyimide, PET	
Minimum trace width[1], 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)[2]	

2.5 Rigid-flex PCB

Rigid-flex PCB technological capabilities

Parameter	Typical	Advanced
Number of layers	4-16	4-28
Minimum trace width[1], 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 / 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)[2]	

2.6 Metal core PCB

Aluminum core PCB technological capabilities

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

[1]Only for copper thickness 9µm and 18µm
[2]Stiffener thickness upon request
[3]Final PCB thickness should be no more than 1,2mm

Notes:
Standard copper via wall thickness is up to 20µm.
Gold thickness for IG coating — 0,05-0,11µm, for Hard Gold (Gold Fingers) — 0,07-1,27µm

3. Materials for PCB

The main elements of the PCB are dielectric substrate (rigid or flexible) with copper conductors on it, vias and holes both plated and non-plated.
As the dielectric substrate the glass-epoxy laminates or composite materials are used. Basic types and parameters of materials used for the production of printed circuit boards are given in the table:

Type	Description	Glass transient temp. T_g	Dielectric constant D_k	Main suppliers
FR4	FR stands for Fire Retardant. FR4 is a glass fiber epoxy laminate. It is the most commonly used PCB material.	135°C	3,8-4,7	Shengyi, Isola, Nanya, KB, Goldenmax
FR4 halogen free	This laminate type does not contain halogen, antimony, phosphorus, etc., does not emit hazardous substances when burning.	140°C	4,5 -4,9	Shengyi, Nanya
FR4 High T_g, FR5	These laminate types have excellent performance in Pb-free soldering.	170°C	3,8-4,6	Nanya, Nelco, Panasonic
RCC	RCC is electrolytic copper foil coated with a layer of special epoxy resin	130°C	4,0	Shengyi, Nelco
PD	Polyimide resin with aramid basis	260°C	4,4	Arlon, Nelco
High frequency (PTFE)	High Frequency laminates are used in PCBs that require a low dissipation factor (D_f) and very stable dielectric constant (D_k)	240–280°C	2,2–10,2	Taconic, Rogers
High frequency (non PTFE)		240–280°C	3,5	Rogers
Polyimide	Material for the production of flexible and rigid-flex PCB	195-220°C	3,4	Dupont, Thinflex

Most often printed circuit boards are made of standard glass-epoxy laminate namely FR4 type, with an operating temperature from -50 to + 110°C, glass transition temperature Tg of 135°C. The dielectric constant D_k can range from 3,8 to 4,6, depending on the supplier and type of material. For compliance with the lead-free technology assembly usually the laminates FR4 High Tg or FR5 are used. When there is a requirement of continuous operation at the high temperature or under extensive temperature stresses the polyimide as a base material is used. Besides polyimide has a good electric strength and often used in military products or in high endurance applications. For printed circuit boards for high frequency or microwave range the special materials are used. The cost of these materials is higher than the basic material FR4.

FR-4

The family of laminates under the name FR-4 by the NEMA (National Electrical Manufacturers Association, USA) classification. These are the most common materials for the single sided, double sided and multilayer printed circuit boards with strict requirements for mechanical strength. FR-4 is a composite material based on woven glass-epoxy compounds. As a rule, laminate has a matte yellow color, green color of the PCB gives it a solder mask which is applied to the surface. Flammability rating is UL94-V0.

Depending on the properties and applications, FR-4 laminates are divided into the following subclasses:

- standard, with a glass transition temperature T_g of ~ 130°C, with UV blocking or without it. The most common and widely used type, at the same time, the cheapest of FR-4;
- with a higher glass transition temperature, T_g ~ 170°C - 180°C, compatible with the lead free reflow technology;
- halogen-free, compatible with the lead free reflow technology;
- with normalized index of CTI ≥ 400, ≥ 600;

FR-1/FR-2

The family of laminates under the name FR-1 and FR-2 by the NEMA classification. These materials are made from paper and phenol compounds and are used only for the production of single side printed circuit boards. FR-1 and FR-2 has the similar parameters, the main difference is that FR-1 has a higher glass transition temperature T_g. Because of the similarity of the parameters and applications of FR-1 and FR-2, many of the material suppliers produce only one type of laminates, most often FR-1. Laminates have a good ability to mechanical processing (milling, punching). Flammability rating is UL94-V0.

FR-1/FR-2 laminates are divided into the following subclasses:

- standard;
- halogen-free, without phosphorus and antimony, non-toxic;
- with normalized index of CTI ≥ 400, ≥ 600;
- non hydrophobic;

CEM-1

The family of laminates under the name CEM-1 by the NEMA classification. These materials are made from paper and two layers of woven glass epoxy and phenol compounds and are used only for the production of single sided printed circuit boards. As a rule, they have a milky white or milky yellow color. Laminates are incompatible with the process of metallization in holes, therefore, they are used only for the production of single sided printed circuit boards. Dielectric properties close to that of FR-4, mechanical endurance is somewhat worse. CEM-1 is a good alternative to FR-4 when the price is the deciding factor. Flammability rating is UL94-V0.

CEM1 laminates are divided into the following subclasses:

- standard;
- with a higher T_g , compatible with the lead free reflow technology;
- halogen-free, without phosphorus and antimony, non-toxic;
- with normalized index of CTI ≥ 600;
- non hydrophobic, with good dimension stability;

CEM-3

The family of laminates under the name CEM-3 by the NEMA classification. Composite material based on glass-epoxy compounds, usually has a milky white color. Very widely used in the production of double-sided PCBs with plated holes. The properties are very similar to that of FR-4, except the lower mechanical endurance. CEM-3 is a cheaper alternative to FR-4 for most applications. Laminates have a good ability to mechanical processing (milling, punching). Flammability rating is UL94-V0.

Depending on the properties and applications, CEM-3 laminates are divided into the following subclasses:

- standard, with UV blocking or without it.
- with a higher T_g , compatible with the lead free reflow technology;
- halogen-free, without phosphorus and antimony, non-toxic;
- with normalized index of CTI ≥ 600;

RO3000

A family of laminates developed for wide use in the early 90s of XX century. These materials have excellent electrical properties at the high frequencies and high thermal stability. The CTE (coefficient of thermal expansion) along the X and Y axes is close to the CTE of copper and FR4, therefore it is possible to produce the reliable RO3000 / FR4 hybrid PCBs. Low dielectric losses (D_f = 0,0013 at a frequency of 10 GHz) provide great benefits when using these laminates in the applications for microwave range.

RO4000

This is a family of materials for a very high frequency range, which has been designed, on the one hand, to achieve the performance comparable with that of materials containing polytetrafluoroethylene (PTFE), and on the other hand, to simplify the technology of PCB production, that is, to make it more in line with the traditional technology used for reinforced laminates (FR4). Materials RO4000 contain reinforced fiberglass of a high glass transition temperature (T_g = 280 °C) with a thermosetting polymer as a bonding agent as well as additives ceramics.

Polyimide

Is a flexible polymeric film used as the substrate for the flexible printed circuit boards. The polyimide films produced under the trademarks Kapton, ThinFlex, Dupont. Advantages: excellent flexibility in the wide temperature range, good electrical properties, high chemical resistance (except for hot concentrated alkalies), very good tensile strength. Some types of polyimide have additional advantages (the coefficient of thermal expansion is similar to that of a copper, small internal stresses). Working temperatures range from -200°C to + 300°C. Disadvantages: high water absorption (up to 3% by weight), relatively high price.

Despite the high glass transition temperature ($T_g \geq 250$ °C), their high temperature properties are limited by the bonding compound layers.

The polyimide film thickness can vary widely, but in practice most of the flexible materials are available with the thicknesses in a narrow range from 12 to 125 µm. When designing the flexible printed circuit boards, it can be useful to remember the practical rule: the stiffness of the flexible materials is proportional to the third power of their thickness. That means if the material thickness is doubled, it becomes eight times tougher and with the same load will bend eight times less.

4. Surface finishing

To preserve the solderability of printed circuit boards for a considerable period of time, it is important to protect the copper surface of the soldering pads with a suitable coating or in other words, to provide a so-called surface finishing.

Surface finishing is made on soldering pads and other copper elements opened from solder mask. In modern productions, as a rule the several types of surface finishes with different properties are used.

For the appropriate selection of surface finishes and for their parameters specification there are a number of regulations, of which the most common are the following IPC standards:

- J-STD-003 Solderability Tests for Printed Boards - defines the solderability test methods;
- IPC 2221 Generic Standard on Printed Board Design - defines the basic requirements for the design of printed circuit boards;
- IPC-7095A Design and Assembly Process Implementation for BGAs - focuses on the BGA components;

We offer a wide range of surface finishes to make an appropriate selection regarding the PCB project requirements.

▪ **HAL or HASL** (Hot Air Leveling or Hot Air Solder Leveling) uses the tin-lead (Sn-Pb) alloys and the alignment by the hot air knife. This finishing is currently most commonly used due to its properties. It provides excellent solderability with substantial shelf life. HAL finishing is easy to manufacture and inexpensive. It is compatible with all methods of soldering or assembling - the manual soldering, wave soldering, reflow etc. The negative feature of this type of finishing is the presence of the lead - one of the most toxic metals that are prohibited for use on the territory of the European Union by the RoHS directive (Restriction of Hazardous Substances). Another limitation - it is surface non-uniformity that is not acceptable in case of fine pitch components. Besides it is not compatible with the COB technology (Chip on Board).

▪ **Lead free HASL** finishing is similar to the normal HASL, except that it could comprise of different alloys like Sn100, Sn96,5/Ag3,5, SnCuNi, SnAgNi and does not contain lead. The finishing is fully RoHS compliant and meets all the requirements of safety and solderability. However, due to the fact that finishing is applied at the considerably higher temperatures, more rigorous requirements are imposed on PCB base materials. Lead free HASL is compatible with all methods of soldering or assembling both in lead and lead free technologies, but it requires the appropriate temperature profiles during the soldering. As compared to SnPb HAL, it is more expensive due to the higher prices of alloys as well as higher energy consumption.

▪ **Immersion Gold or ENIG** (Electroless Nickel/ Immersion Gold) the finishing from a Ni/Au family. The thickness of the finishing: Ni 3,0 - 5,0 µm, Au 0,06 - 0,1 µm. The finishing is made by chemical method. The main function of thin gold layer is to protect the nickel layer from oxidation, and the nickel layer prevents a mutual diffusion of gold and copper. The excellent flatness of the finishing makes it suitable to use in case of fine pitch components. The finishing is fully RoHS compliant. Compatible with all methods of soldering. The main limitation is a higher price. Besides there could be a risk of immersion gold defect due to the oxidation, so-called "black pad" that is critical for BGA assembly.

▪ **ENEPIG** (electroless nickel electroless palladium immersion gold) - the finishing from a Ni/Au family but this surface made of three metallic layers (a layer of electroless nickel, a layer of electroless palladium and a layer of immersion gold). The thickness of the finishing: Ni 3,5 - 5,0 µm, Pd 0,2 - 0,4 µm (although sometimes 0,1 - 0,2 µm), Au 0,03 - 0,05 µm. ENEPIG is suitable for all common types of wire-bonding. ENEPIG is a type of surface finish done on a copper layer to protect it from corrosion and oxidation in printed circuit boards. It is suitable for high-density SMT designs. Furthermore, it follows RoHS compliance. ENEPIG is an excellent finish for BGA substrates since it is both solderable and bondable.

▪ **Gold Fingers** - the finishing from a Ni/Au family. Plating thickness: Ni 5,0 - 9,0 µm, Au 0,2 - 1,0 µm. It is applied by electrochemical deposition (i.e. electroplating). Most often the finishing is used for PCB edge connectors. It has high mechanical strength, resistance to abrasion and adverse environmental effects. Indispensable where it is important to ensure excellent electrical contact with long service life.

▪ **Immersion Tin** - the finishing made chemically. It is compatible with all methods of soldering or assembling. The finishing has an acceptable shelf-life period - up to one year. This is achieved by using organic compounds to make the barrier to intermetallic bonds that affect the oxidation of the surface. Such kind of insulating also prevents tin from crystallization. The finishing with the thickness of 1 µm has a good flatness and suitable for fine pitch components.

▪ **Immersion Silver** - is a non-electrolytic chemical finish applied by immersing the copper PCB into a tank of silver ions. The finishing has an acceptable for circuit boards with EMI shieldingand is also used for bonding. The average surface thickness of the silver is 0,13 - 0,46 µm. The Immersion Silver finish provides good electrical properties and maintains excellent weldability for soldering even when exposed to heat, moisture, and contamination.

▪ **OSP** (Organic Solderability Preservatives) - a family of organic coatings that applied directly onto the bare copper to protect it from oxidation during storage and soldering. This inexpensive finishing has a flat surface and is suitable for SMD assembly. It complies with the RoHS directive. As a result, it is a cheap alternative to the HASL finishings. Unfortunately, it has a limited shelf life (months) and quickly degrades during the soldering process.

▪ **Carbon** (graphite coating) - mechanical load-resistant coating. Used to apply on contact pads of buttons. The minimum distance between pads of different circuits should be at least 0,4 mm.

5. Design guidelines for different types of PCB

Typical PCB designs are based on a standard FR4 glass-epoxy laminate with operating temperature of typically -50 to +110 °C, with glass transition temperature Tg of about 135 °C. Dielectric constant D_k can be from 3,8 to 4,6 from, depending on supplier and material quality. In case of higher heat environment work needs or pcb assembly following lead-free technology (temperature up to 260 °C), High Tg FR4 or FR5 is used.

In the case of continuous operation in conditions of high ambient temperature or rapid temperature changes, polyimide is used as the base material. In addition, polyimide is also used for the manufacture of high-reliability printed circuit boards for military applications, as well as in cases where flexible or rigid-flex printed circuit boards are required.

If there are electronic components on the printed circuit board that emit significant thermal power (for example, power LEDs), printed circuit boards on a metal substrate made of aluminum or copper are used.

In case of PCB for very high frequency range - above 1 GHz, there is an option to fabricate the outer layers from laminates dedicated for RF or microwave applications, meanwhile the inner core(s) could be a normal FR4 (so called hybrid PCB), or entire stackup (core and prepregs) could be built from high frequency laminates.

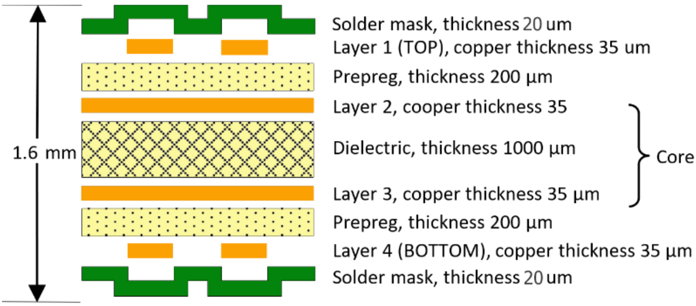
5.1 Multilayer and HDI printed circuit boards

A multilayer printed circuit board consists of one or more cores (thin glass-epoxy laminate with two copper layers), several layers of prepregs and outer layers of copper in the form of a copper foil. Prepregs are used to glue all layers together.

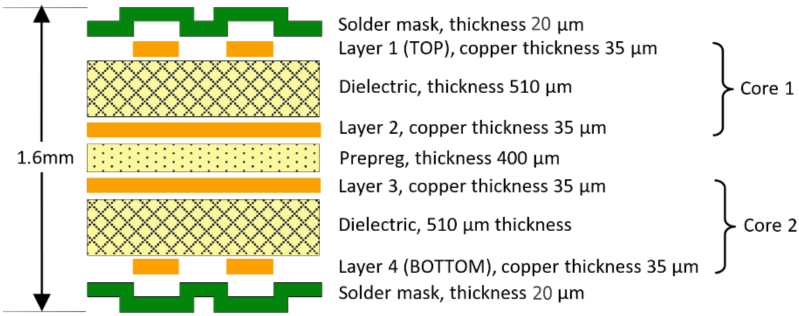
After pressing all the layers at high temperature, we get the basis for a multilayer printed circuit in which holes are drilled and then metallized.

Below we can see the typical designs of multilayer printed circuits:

Four layers, type I



Four layers of type II



A type I multilayer printed circuit board consists of one core and two additional layers of copper. With this design, holes can only be made between layers 1-2-3-4 (through) and 2-3 (buried). This is the most common variant of four-layer boards and at the same time the cheapest.

A type II printed circuit board, also called a two-core multilayer printed circuit board, is somewhat more difficult to manufacture, it consists of two cores. The holes can connect layers 1-2-3-4 (through), 1-2 (blind) and 3-4 (blind).

In case of more layers, PCB can consist of two, three, four or more cores, the appropriate number of copper layers and adhesive prepregs. The main rule to remember is that the holes can be made exclusively between the copper layers within one core or in a prepared stack of cores and copper layers glued together by prepregs.

To the stack of layers with drilled and metallized holes we can subsequently add outer layers in form of copper foil or another cores (of course along with adding prepregs) and then, after pressing it all together we can make more drill types (vias), connecting inner and outer layers.

For various stack-ups of multilayer PCB it is recommended to use standard cores and prepregs:

Standard copper thickness	9 µm
	18 µm
	35 µm
	50 µm
	70 µm

Standard prepregs	106 (0,050 mm)
	1080 (0,075 mm)
	2116 (0,105 mm)
	7628 (0,185 mm)
	7628 (0,216 mm)

Standard core thickness	0,1 mm
	0,13 mm
	0,21 mm
	0,25 mm
	0,36 mm
	0,51 mm
	0,71 mm
	1,0 mm
	1,2 mm
	1,6 mm
	2,0 mm
	2,4 (2,5) mm
	3,2 mm

Theoretically, it is possible to design very complex stack-ups of printed circuit boards with many cores and types of vias, but please keep in mind that quite often the fabrication of such structures can be very expensive.

For internal layers, it is better to use copper at least with thickness of 35 µm or more.

The design of multilayer printed circuit boards begins with the planning of the stack-up, which can be fabricated by manufacturers without complications. In case of complex stack-ups, we strongly recommend to discuss it with the manufacturer in advance.

In our **Electronics Design and Manufacturing series** we have an issue **PCB Stackup Planning,** dedicated to the construction of stack-ups and materials used for this. You can download this issue from our website.

When electronic components with a very small pitch of pads are used (for example micro BGA with the raster smaller than 0,5 mm), high density interconnections (HDI) technology is often used, which means the implementation of very thin tracks (up to 0,076 mm wide) and holes with a diameter of up to 0,1 mm. These holes can be made directly in the BGA soldering pads (via in pad) on the PCB without causing soldering problems. Also, to increase the reliability of connections, these holes in pads can be additionally filled with copper or paste.

When there are holes with a diameter of less than 0,2 mm (0,15 mm) on the printed circuit board, they cannot be performed by mechanical drilling. In this case, it is necessary to use laser or plasma technology to create such holes. It should be remembered that such holes will not be through holes. They will be just blind micro-vias connecting the outer layer with the layer next to it. This requires the use of special prepregs between layers with micro-vias.

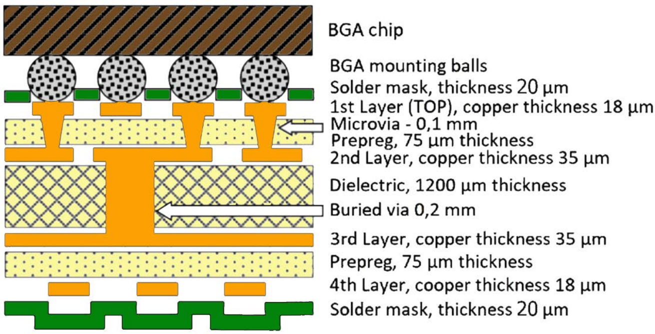
Generally speaking, a typical HDI printed circuit board is made of standard cores similar to a common multilayer PCB, but has outer layers made of special thin prepregs to make laser microvias possible.

It should be added that there can be up to three or even four outer layers with micro-vias. It is also possible to connect micro-vias between them by placing one above the other and filling them with copper for direct electrical contact.

In the technical guides, you can find a shortcut such as x-N-x, where x is the number of layers with micro vias, and N is a number of standard layers. For example, 2-4-2 is an eight-layer PCB with four HDI layers. The technological capabilities of many manufacturers are limited to the manufacture of HDI PCB up to 4-N-4, where N can be up to 14 layers.

Below you can see what the layer structure of a typical HDI printed circuit board looks like.

Typical four-layer HDI PCB structure



5.2 Flexible and Rigid-flex PCB

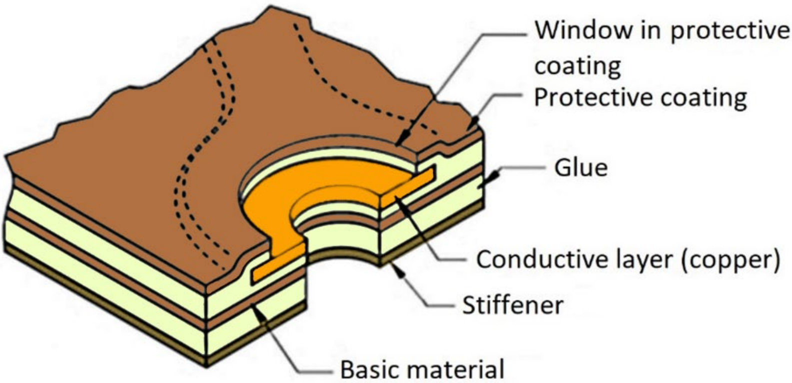
The main feature of flexible circuits is the ability to bend over a wide range (both static and dynamic) due to the low thickness and specific materials used in production.

Flexible printed circuits can be both single-sided and multi-layer (usually up to 4 layers).

The elements of the structure of flexible printed circuit boards are:

- **Basic material** - polyimide or polyester (PET) foil
- **Adhesive (bonding material)** - acrylic or epoxy polymers

- **Copper foil** - conductive material
- **Protective coating (coverlay)** - polyimide or polyester foils
- **Stiffeners** - additional inserts made of thicker material, for example FR4, polyimide or polyester.



The most common base material for flexible circuit boards is polyimide. Although PET is cheaper, manufacturers use it much less frequently due to its lower heat resistance. On the other hand, PET has advantages in terms of chemical resistance and moisture absorption. One-sided flexible circuits for automotive industry are most often produced from PET.

The thickness of polyimide foil can vary widely, although in practice most flexible materials offered by suppliers have a thickness in a narrow range from 12 to 125 microns.

The choice of material thickness may depend on the requirements for the flexible PCB itself, concerning its flexibility and mechanical strength. In any case, it is worth remembering that not only the base material will affect the final thickness, but also, importantly, a layer of copper, protective coatings and even glue.

Copper foil can be of two types - prepared by rolling or by electrochemical method. The first one has better mechanical properties and is more often used for flexible printed circuit boards with dynamic bending, but at the same time it is more expensive.

The protective coating is an analogue of a solder mask and is often made of the same basic material as the flexible printed circuit board itself, only of a smaller thickness. Due to the mechanical processing (drilling, stamping) of protective coatings, it is necessary to make significantly larger openings in it compared to the openings in the solder mask on a conventional rigid printed circuit board. That is why when designing a flexible PCB, it is necessary to make one large opening for several soldering pads with a small pitch.

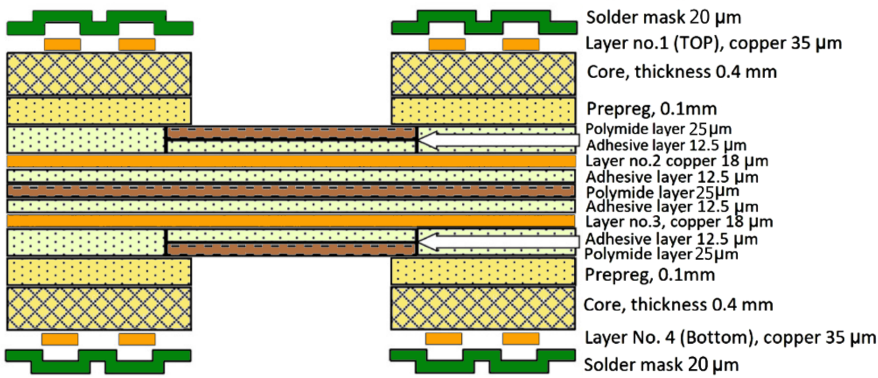
There are no such restrictions for medium and large series of flexible printed circuit boards, since in this case the openings in the coverlay will be made with a laser. Unlike small batches and prototypes, the use of laser cutting in mass production is economically justified.

Stiffeners are necessary when electronic components are mounted on a flexible printed circuit board or the edge of a flexible printed circuit board must be inserted into the connector and its appropriate thickness must be provided.

Rigid-flex printed circuits are the most difficult solution, since they use both rigid and flexible circuit manufacturing technologies.

Below we can see the layer structure of a typical four-layer rigid-flex printed circuit board:

Typical structure of a rigid-flexible board



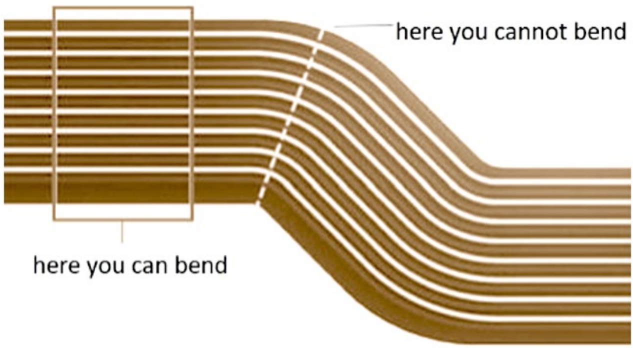
When designing flexible and rigid-flex printed circuit boards, it is worth following the rules:

- The bending radius depends on the thickness of the flex PCB as follows:

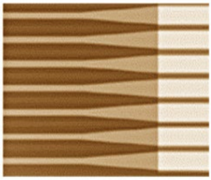
Flexible circuit board type	Minimum bend radius
One-sided	from 3 to 6 board thicknesses
Two-sided	from 6 to 10 board thicknesses
Multiple layer	from 10 to 15 board thicknesses
One-sided with dynamic bending	from 20 to 40 board thicknesses

- A practical rule may be useful: the rigidity of flexible materials is proportional to their thickness to the third power. This means that if the material is doubled in thickness, it will become eight times tougher and will deform eight times less under the same load.

Flexible circuit must bend at right angles to the conductors:



- Copper conductors on the flexible printed circuit board should change direction smoothly (you have to avoid sharp angles), the same applies to changes in the width of the conductors - the transition from a wider conductor to a narrow one must be gradual:

- The openings for the soldering pads in coverlay should be slightly smaller (0.04 - 0.05 mm) than the size of pads themselves, this is especially important for pads with holes without metallization or soldering pads for SMD components.
- Conductors running in parallel in two adjacent layers should not be placed one above the other. You have to move them so that they do not overlap.
- Copper polygons should always be done as a large mesh, not solid.

5.3 Metal core PCB

Most often, printed circuits on a metal substrate are used if there are electronic components on it that emit significant thermal power (for example, power LEDs). Such printed circuit boards can be single-sided, double-layer and even multilayer - up to 4 layers.

Aluminum is most often used as the core (thermal conductivity 150-250 W/mK).

In the case of higher requirements, a copper core is used to dissipate more thermal power (thermal conductivity is about 400 W/mK), such PCB are much more expensive and, moreover, they are difficult to mechanically mill.

The thermal conductivity of the finished PCB on the metal core is significantly lower than the thermal conductivity of the material itself. It depends on the type and thickness of the dielectric between the layer with copper conductors and the metal core.

When designing a printed circuit board on a metal core, we must remember that we cannot make through holes with metallization. There may be through holes used as mounting holes, or blind and buried vias only for connecting layers. It means that PCB on a metal core is suitable only for automated SMT assembly.

Main characteristics of metal core printed circuit boards are:

- thermal conductivity (ranges from 1-4 W/mK)
- dielectric strength (2-10 kV)
- coefficient of thermal expansion (about 20 ppm/^oC)

5.4 RF and Microwave PCB

Printed circuit boards for the microwave range differ from common PCB in that additional measures must be taken at the design stage and during production to ensure signal integrity and immunity to electromagnetic interference. In the high-frequency range, the tracks of printed circuit boards are considered as transmission lines having a certain complex impedance, and the effects of signal energy absorption in the dielectric are also taken into account.

Nonlinear signal distortions (due to reflections and absorption) occur at frequencies of 500 MHz - 1000 MHz and higher. For many devices, the need to control the signal integrity arises even at lower frequencies.

Following the rules of signal integrity is achieved due to the signal paths are designed as transmission lines with given parameters (wave impedance of the line and delay time of the signal propagation).

The properties of a conductor as a transmission line depend mainly on its width and thickness, PCB stackup (distance between the signal layer and ground plane), as well as the dielectric properties of the material (dielectric constant).

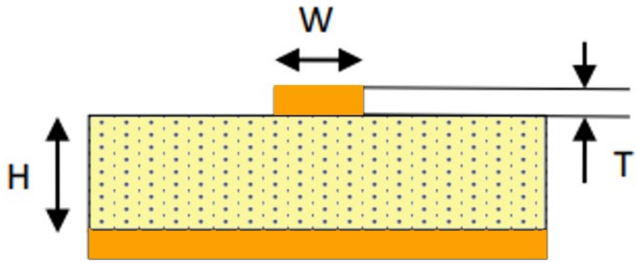
The transmission line is matching when it has the same wave impedance along its entire length as both the signal source and the signal receiver. For differential and parallel signals (e.g., digital buses), the delay time of signal propagation along all paths followed by these signals must be additionally equalized.

Fast and critical signals should be located in the layer next to the grounding layer (ground plane), which has a corresponding power layer (power plane) and the decoupling between which it is made by ceramic blocking capacitors. Transmission line should not have openings or interruptions in the ground plane over which it runs along, as this leads to significant changes in impedance relative to the calculated values.

Let's review the simplest types of transmission lines:

Note: The calculation below can only be used for a preliminary assessment.

1. **Microstrip line** - a conductor on board's surface, under which there is a ground plane.



Wave impedance of the line and propagation time of the signal front can be estimated from the following formula:

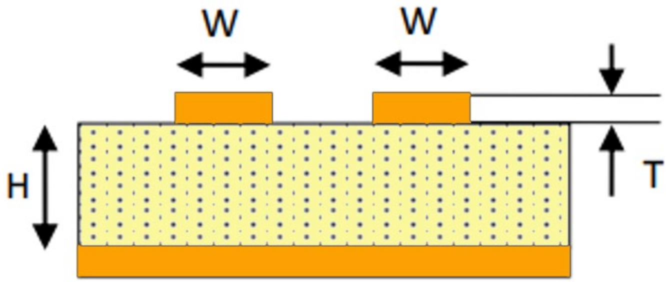
$$Z_0 = \left[\frac{87}{\sqrt{\varepsilon_r + 1{,}41}}\right] \times \ln\left[\frac{5{,}98H}{0{,}8W + T}\right], \text{[ohm]}$$

Provided that W≤ $2H$

$$t_0 = 3{,}3\sqrt{0{,}47\varepsilon_r + 0{,}67}\ , \text{[ns/m]}$$

where ε_r - dielectric constant of material, W - width of conductor, T - thickness of copper including metallization, H - thickness of dielectric (distance to the ground plane).

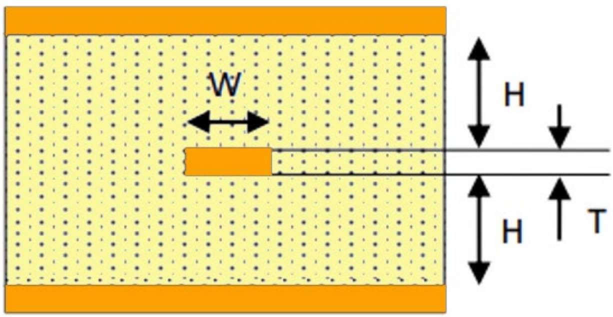
2. **Coplanar strip line** - two transmission lines in close parallel distance from each other, having a common ground plane.



$$Z_{diff} = 2Z_0\left[1 - 0{,}48\exp\left(-\frac{0{,}96S}{H}\right)\right], [ohm]$$

Where S – the distance between transmission lines.

3. **Symmetrical strip line** - a conductor in the inner layers of the printed circuit board, located symmetrically.

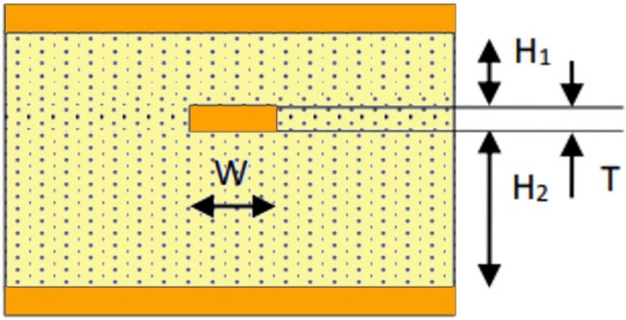


$$Z_0 = \left[\frac{60}{\sqrt{\varepsilon_r}}\right] \times \ln\left[\frac{1{,}9(2H+T)}{0{,}8W+T}\right], \text{[ohm]}$$

Provided that $W \leq 2H$, $T \leq H/2$

$$t_0 = 3{,}3\sqrt{\varepsilon_r}, \text{[ns/m]}$$

4. **Asymmetrical strip line** - a conductor in internal layers of the board, located with an offset (not symmetrically) relative to the ground planes.



$$Z_0 = \left[\frac{80}{\sqrt{\varepsilon_r}}\right] \times \ln\left[\frac{1{,}9(2H_1+T)}{0{,}8W+T} \times \left(1 - \frac{H_1}{4H_2}\right)\right], \text{[ohm]}$$

Provided that $H1 \leq H2$ $\quad 0{,}1H1 \leq W \leq 2H1$, $T \leq H/4$

$$t_0 = 3{,}3\sqrt{\varepsilon_r}, \text{[ns/m]}$$

In order to properly manufacture a printed circuit board with impedance control, the following must be specified in the design documentation:

- In which layer there is a transmission lines, its conductor widths and impedance value for each type of transmission lines
- In which layer is the ground plane relative to which the impedances are calculated

Before proceeding to the design of the printed circuit board layout, the designer must plan the stack-up of the printed circuit board and perform preliminary calculations of the impedance based on the average value of dielectric constant of materials, the distance between layers and the width of the conductors.

It is enough to make calculations with an accuracy of +/- 10%. In any case, during the manufacture of the printed circuit, adjustments will be made to all parameters to bring the real impedance values in line with those required by the customer. If it will be necessary to change something (e.g. the width of the tracks) in order to achieve the target values of impedance, the technical department will always ask you in advance to confirm all the changes in project.

In the production of printed circuit boards with impedance control, the impedance value is measured using the appropriate equipment, and only those printed circuits that exactly meet the requirements are shipped to the customer.

A typical FR4 material has a dielectric constant of about 4,6 at a frequency of 1 MHz, which decreases to 4 when the frequency increases to 1 GHz. However, its actual values may vary within +/- 30%. That is why there are materials whose dielectric constant value is normalized and does not change with frequency changes in a wide range.

For the manufacture of printed circuit boards operating in the high frequency range, materials produced by Rogers Corporation are most often used. You may find basic information about the properties of high frequency materials on their website.

We always have in stock the most popular types of materials 4350B and 4003C of various thicknesses.

6. IPC standards

IPC standards are documents used by manufacturers, suppliers and designers of electronic devices. Working on the basis of recognized IPC standards will help all specialists speak one language - the language of the international electronics industry. In addition, the use of IPC standards solves the misunderstanding among the participants of the electronic market, because in this case they know that they must adhere to the current industry standards.

Below is a list of the most commonly used standards in the electronics industry:

Acceptability of Electronics Assemblies IPC-A-610

- Rework, Modification and Repair of Electronic Assemblies IPC-7711/21
- Solderability Tests J-STD-002, J-STD-003
- BGA, CSP, HDI Flip Chip J-STD-030, IPC-7094, IPC-7095
- Test Methods IPC-TM-650, IPC-9691

- Requirements for Soldered Electronic Assemblies J-STD-001
- Stencil Design Guidelines IPC-7525
- Materials for Assembly J-STD-004, J-STD-005, IPC-HDBK-005, J-STD-006, IPC-SM-817, IPC-CC-830, HDBK-830
- Components J-STD-020, J-STD-033, J-STD-075

Acceptability of Printed Boards IPC-A-600

- Requirements for Electrical Testing IPC-9252

Performance Specification for Printed Boards IPC-6011, 6012, 6013, 6017

- Solder Mask and Flexible Cover Materials IPC-SM-840
- Surface Finishing IPC-4552, IPC-4553, IPC-4554

Base materials IPC-4101, 4104, 4202, 4203, 4204

- Metal Foil for Printed Board Applications IPC-4562

Generic Standard on PCB Design IPC-2221, IPC-2222 – 2226, IPC-7351

- Design Guide for High-Speed Controlled Impedance Circuit Boards IPC-2141, IPC-2251

Data Transfer and Electronic Product Documentation IPC-2581, IPC-2610

- Materials Declaration IPC-1751, IPC- 1752
- Requirements for marking, symbols and labels J-STD-609