ELECTRONICS
DESIGN AND MANUFACTURING
SERIES

PCB STACK-UP
PLANNING

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

1. **The advantages of multilayer PCB**
2. **What determines the cost of PCB**
3. **Recommended stack-ups**
   - 3.1 4-layer stackups
   - 3.2 6-layer stackups
   - 3.3 8-layer stackups
   - 3.4 10-layer stackups
   - 3.5 12-layer stackups
   - 3.6 14-layer stackups
   - 3.7 16-layer stackups
4. **Via holes plugging**
5. **Copper thickness in multilayer PCB**
6. **PCB fabrication capabilities**

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 at **+48 338 338 338** or write to the following e-mail address: **office@nanotech-elektronik.com** (we communicate in English, German and Polish).

Sincerely,
The Team of Nanotech Elektronik.

1. The advantages of multilayer PCBs

Sometimes PCB layout designers believe that the use of multilayer PCBs is solely due to the high density of connections, and that additional layers are introduced when connection routing cannot be made on fewer layers.

However, in addition to this obvious fact that the number of routing layers depends on the number and density of connections, there are many other situations where the use of additional layers is not only recommended but often necessary, even in cases where the PCB can be designed with fewer layers or even as a double-sided PCB.

First of all, an important criterion for selecting the number of layers for routing is the electrical properties of the conductors of the PCBs at higher frequencies.
Today, more and more electronic designs include high-speed data transmission lines. Therefore, a conductor on a PCB isn't solely an electrical connection, but rather a transmission line with a specific impedance value. At the same time, to ensure the integrity (absence of distortion and reflections) of high-frequency digital signals, we need to control the exact impedance value so that it corresponds to the calculations "by design".

Impedance control mainly is possible in multilayer printed circuits, with internal reference (ground/power) planes, ensuring efficient isolation of the return currents.

Sometimes designers try to calculate the impedance on a two-sided printed circuit board, but, unfortunately, in practice it is not possible to achieve the required accuracy of 5% - 10% due to the spread of laminate parameters and the impossibility to adjust the distance between the signal layer and the reference plane.

Another important advantage of multilayer printed circuit boards is their resistance to electromagnetic interference, as well as their ability to reduce the level of their own electromagnetic radiation.

A properly designed layer stack-up allows not only to ensure optimal wave characteristics of the transmission lines on the PCB, but also to shield the signal paths most sensitive to interference, reduce the level of intrinsic noise, minimize parasitic energy losses and, thus, ensure optimal thermal operation of the electronic device.

It is possible to improve the stability of an electronic device and increase performance by changing from a double-sided printed circuit board to a four-layer printed circuit board, adding internal copper planes for power and grounding. Sometimes, in order to successfully certify a device to more stricter requirements for electromagnetic compatibility, it is enough to optimize the printed circuit board without significant changes in the circuit.

On the other hand, an increase in the number of layers of a multilayer printed circuit board leads to a higher cost, and also requires certain technological capabilities from the manufacturer. However, if you understand what factors and how increase the cost of a multilayer printed circuit board, you can design its layout in the most optimal way and achieve the desired solution at an affordable price.

In the next chapter, we will explain these factors and how they affect the cost of manufacturing printed circuit boards.

2. What determines the cost of PCB

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

It can be said that the cost of a printed circuit board depends on the number of cores - the more cores, the more expensive it is to manufacture a printed circuit board.

Depending on how the interlayer connections are designed, the PCB may contain metallized holes, or vias - through vias, blind and buried. In our previous issue "Guide to Printed Circuit Boards" we have already explained that usually metallized holes can be made mechanically only between layers of copper within the same core or between copper layers within the stack of several cores glued together using prepregs.

Making a stack of cores includes drilling cycles and subsequent metallization. And the number of such cycles is also one of the main price criteria affecting the cost of a multilayer printed circuit board.

Thus, we can formulate a rule: for the same number of layers, a PCB with fewer cores number and, correspondingly, fewer drilling and metallization cycles will be cheaper.

In the case of PCB with high connection density, several additional signal layers are usually used. This is especially common when using BGAs with a large number of pads. To route signals away from the BGA pads, designers use vias between the pads of the BGA chip and additional 2, 3 or 4 layers for signal routing.

Therefore, printed circuit boards with high-density BGAs usually utilize more layers and are mostly more expensive.

A good solution in such a situation could be the HDI printed circuit board technology, which allows to reduce the number of signal layers using micro-vias and vias in the pad (Via-in-pad).

Sometimes, the use of more expensive HDI technology can lead to a cheaper printed circuit board by using fewer layers than if the same printed circuit board was implemented using traditional technology.

Unfortunately, there is no well defined price dependence on the number of layers in multilayer printed circuit boards and HDI printed circuit boards, so we recommend to ask the manufacturer for a price quote for each approach at the early stages of layout design.

If you want to learn more about the design of HDI structures (build-ups), we recommend checking out our guide “High Density Interconnect Printed Circuits”, which can be downloaded from our website.

3. Recommended stack-ups

In order to properly design a multilayer printed circuit board, the construction of the layer stack-up must be carefully considered. Layer stack-up planning depends, on the one hand, on the technological capabilities of manufacturer and, on the other hand, on the requirements for the electrical properties of the PCB.

You need to remember that a multilayer PCB consists of a combination of cores, prepregs and copper foil layers. Below you can find some of the materials that

our suppliers have in stock and we recommend to use these materials in your designs when orering PCB from us:

Materials used for the multilayer PCB fabrication

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

Standard prepreg thickness	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

Below, you can find a list of the most common variants of PCB layer stack-ups. The most cost-effective variants we marked with the stamp "Recommended".

4-layer stackups

Required board thickness: **0,5 +/- 0,1 mm**

Nr.	Layer				Thickness, um	Material
Nr. 1	SMT				20	Solder mask
	TOP	1			18-25	Copper Plating
					18	Copper Foil
				1x2116	105	Prepreg
	INT 1	2			35	Copper
					130	Core
	INT 2	3			35	Copper
				1x2116	105	Prepreg
	BOTTOM	4			18	Copper Foil
					18-25	Copper Plating
	SMB				20	Solder mask

Approximate board thickness: 0,46 mm

Required board thickness: **0,8 +/- 0,1 mm**

Nr.	Layer				Thickness, um	Material
Nr. 2	SMT				20	Solder mask
	TOP	1			18-25	Copper Plating
					18	Copper Foil
				1x1080	75	Prepreg
				1x2116	105	Prepreg
	INT 1	2			35	Copper
					250	Core
	INT 2	3			35	Copper
				1x2116	105	Prepreg
				1x1080	75	Prepreg
	BOTTOM	4			18	Copper Foil
					18-25	Copper Plating
	SMB				20	Solder mask

Approximate board thickness: 0,71 mm

Required board thickness: **1,0 +/- 0,1 mm**

Nr.	Layer			Thickness, um	Material
Nr. 3	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			2x1080	150	Prepreg
	INT 1	2		35	Copper
				510	Core
	INT 2	3		35	Copper
			2x1080	150	Prepreg
				18	Copper Foil
	BOTTOM	4		18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 0,89 mm

Required board thickness: **1,6 +/- 0,15 mm**

Nr.	Layer			Thickness, um	Material
Nr. 4	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x1080	75	Prepreg
			1x7628	185	Prepreg
	INT 1	2		35	Copper
				1000	Core
	INT 2	3		35	Copper
			1x7628	185	Prepreg
			1x1080	75	Prepreg
				18	Copper Foil
	BOTTOM	4		18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 1,54 mm

Required board thickness: **2,0 +/- 0,2 mm**

Nr.	Layer			Thickness, um	Material
Nr. 5	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			2x1080	150	Prepreg
	INT 1	2		35	Copper
				1600	Core
	INT 2	3		35	Copper
			2x1080	150	Prepreg
	BOTTOM	4		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 1,92 mm

Required board thickness: **2,5 +/- 0,2 mm**

Nr.	Layer			Thickness, um	Material
Nr. 6	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x1080	75	Prepreg
			1x2116	105	Prepreg
	INT 1	2		35	Copper
				2000	Core
	INT 2	3		35	Copper
			1x2116	105	Prepreg
			1x1080	75	Prepreg
	BOTTOM	4		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 2,41 mm

Required board thickness: **3,0 +/- 0,2 mm**

	Layer				Thickness, um	Material
Nr. 7	SMT				20	Solder mask
	TOP	1			18-25	Copper Plating
					18	Copper Foil
				1x1080	75	Prepreg
				1x7628	185	Prepreg
	INT 1	2			35	Copper
					2400	Core
	INT 2	3			35	Copper
				1x7628	185	Prepreg
				1x1080	75	Prepreg
	BOTTOM	4			18	Copper Foil
					18-25	Copper Plating
	SMB				20	Solder mask

Approximate board thickness: 2,95 mm

4-layer PCB stack-up (recommended)

Required board thickness: **1,6 +/- 0,1 mm**

	Layer				Thickness, um	Material
Nr. 8	SMT				20	Solder mask
	TOP	1			18-25	Copper Plating
					18	Copper Foil
				1x2116	105	Prepreg
	INT 1	2			35	Copper
					1100	Core
	INT 2	3			35	Copper
				1x2116	105	Prepreg
	BOTTOM	4			18	Copper Foil
					18-25	Copper Plating
	SMB				20	Solder mask

Approximate board thickness: 1,5 mm

6-layer stackups

Required board thickness: **0,7 +/- 0,1 mm**

Stackup	Name	Layer	Prepreg type	Thickness, um	Material
Nr. 1	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x2116	105	Prepreg
	INT 1	2		18	Copper
				130	Core
	INT 2	3		18	Copper
			1x2116	105	Prepreg
	INT 3	4		18	Copper
				130	Core
	INT 4	5		18	Copper
			1x2116	105	Prepreg
	BOTTOM	6		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 0,68 mm

Required board thickness: **0,8 +/- 0,1 mm**

Stackup	Name	Layer	Prepreg type	Thickness, um	Material
Nr. 2	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x2116	105	Prepreg
	INT 1	2		35	Copper
				130	Core
	INT 2	3		35	Copper
			1x2116	105	Prepreg
	INT 3	4		35	Copper
				130	Core
	INT 4	5		35	Copper
			1x2116	105	Prepreg
	BOTTOM	6		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 0,75 mm

Required board thickness: **1,0 +/- 0,1 mm**

	Layer				**Thickness, um**	**Material**
Nr. 3	SMT				20	Solder mask
	TOP	1			18-25	Copper Plating
					18	Copper Foil
				2x1080	150	Prepreg
	INT 1	2			35	Copper
					210	Core
	INT 2	3			35	Copper
				1x2116	105	Prepreg
	INT 3	4			35	Copper
					210	Core
	INT 4	5			35	Copper
				2x1080	150	Prepreg
	BOTTOM	6			18	Copper Foil
					18-25	Copper Plating
	SMB				20	Solder mask

Approximate board thickness: 0,96 mm

Required board thickness: **1,6 +/- 0,1 mm**

	Layer				**Thickness, um**	**Material**
Nr. 4	SMT				20	Solder mask
	TOP	1			18-25	Copper Plating
					18	Copper Foil
				1x1080	75	Prepreg
				1x7628	185	Prepreg
	INT 1	2			35	Copper
					360	Core
	INT 2	3			35	Copper
				1x7628	185	Prepreg
	INT 3	4			35	Copper
					360	Core
	INT 4	5			35	Copper
				1x7628	185	Prepreg
				1x1080	75	Prepreg
	BOTTOM	6			18	Copper Foil
					18-25	Copper Plating
	SMB				20	Solder mask

Approximate board thickness: 1,56 mm

Required board thickness: **2,0 +/- 0,2 mm**

Stack	Layer		Construction	Thickness, um	Material
Nr. 5	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x1080	75	Prepreg
			1x7628	185	Prepreg
	INT 1	2		35	Copper
				510	Core
	INT 2	3		35	Copper
			1x7628	185	Prepreg
			1x1080	75	Prepreg
	INT 3	4		35	Copper
				510	Core
	INT 4	5		35	Copper
			1x7628	185	Prepreg
			1x1080	75	Prepreg
	BOTTOM	6		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 1,92 mm

Required board thickness: **2,4 +/- 0,2 mm**

Stack	Layer		Construction	Thickness, um	Material
Nr. 6	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x1080	75	Prepreg
			1x7628	185	Prepreg
	INT 1	2		35	Copper
				710	Core
	INT 2	3		35	Copper
			1x7628	185	Prepreg
			1x1080	75	Prepreg
	INT 3	4		35	Copper
				710	Core
	INT 4	5		35	Copper
			1x7628	185	Prepreg
			1x1080	75	Prepreg
	BOTTOM	6		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 2,32 mm

Required board thickness: **3,0 +/- 0,2 mm**

Nr.	Layer		Glass style	Thickness, um	Material
Nr. 7	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x1080	75	Prepreg
			1x2116	105	Prepreg
	INT 1	2		35	Copper
				1100	Core
	INT 2	3		35	Copper
			1x2116	105	Prepreg
	INT 3	4		35	Copper
				1100	Core
	INT 4	5		35	Copper
			1x2116	105	Prepreg
			1x1080	75	Prepreg
	BOTTOM	6		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 2,9 mm

6-layer PCB stack-up (recommended)

Required board thickness: **1,6 +/- 0,15 mm**

Nr.	Layer		Glass style	Thickness, um	Material
Nr. 8	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x7628	185	Prepreg
	INT 1	2		35	Copper
				400	Core
	INT 2	3		35	Copper
			2x2116	210	Prepreg
	INT 3	4		35	Copper
				400	Core
	INT 4	5		35	Copper
			1x7628	185	Prepreg
	BOTTOM	6		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 1,55 mm

Required board thickness: **1,6 +/- 0,15 mm**

	Layer			Thickness, um	Material
Nr. 9	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x2116	105	Prepreg
	INT 1	2		35	Copper
				500	Core
	INT 2	3		35	Copper
			1x7628	185	Prepreg
	INT 3	4		35	Copper
				500	Core
	INT 4	5		35	Copper
			1x2116	105	Prepreg
	BOTTOM	6		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 1,57 mm

8-layer stackups

Required board thickness: **1,0 +/- 0,1 mm**

	Layer			Thickness, um	Material
Nr. 1	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x2116	105	Prepreg
	INT 1	2		18	Copper
				130	Core
	INT 2	3		18	Copper
			1x2116	105	Prepreg
	INT 3	4		18	Copper
				130	Core
	INT 4	5		18	Copper
			1x2116	105	Prepreg
	INT 5	6		18	Copper
				130	Core
	INT 6	7		18	Copper
			1x2116	105	Prepreg
	BOTTOM	8		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 0,96 mm

Required board thickness: **1,6 +/- 0,15 mm**

	Layer		Prepreg type	Thickness, um	Material
Nr. 2	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			2x1080	150	Prepreg
	INT 1	2		35	Copper
				250	Core
	INT 2	3		35	Copper
			1x7628	185	Prepreg
	INT 3	4		35	Copper
				210	Core
	INT 4	5		35	Copper
			1x7628	185	Prepreg
	INT 5	6		35	Copper
				250	Core
	INT 6	7		35	Copper
			2x1080	150	Prepreg
	BOTTOM	8		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 1,56 mm

Required board thickness: **2,0 +/- 0,2 mm**

	Layer		Prepreg type	Thickness, um	Material
Nr. 3	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x1080	75	Prepreg
			1x2116	105	Prepreg
	INT 1	2		35	Copper
				360	Core
	INT 2	3		35	Copper
			2x1080	150	Prepreg
	INT 3	4		35	Copper
				360	Core
	INT 4	5		35	Copper
			2x1080	150	Prepreg
	INT 5	6		35	Copper
				360	Core
	INT 6	7		35	Copper
			1x2116	105	Prepreg
			1x1080	75	Prepreg
	BOTTOM	8		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 1,92 mm

Required board thickness: **2,4 +/- 0,2 mm**

	Layer			Thickness, um	Material
Nr. 4	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			2x1080	150	Prepreg
	INT 1	2		35	Copper
				510	Core
	INT 2	3		35	Copper
			1x7628	185	Prepreg
	INT 3	4		35	Copper
				510	Core
	INT 4	5		35	Copper
			1x7628	185	Prepreg
	INT 5	6		35	Copper
				510	Core
	INT 6	7		35	Copper
			2x1080	150	Prepreg
	BOTTOM	8		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 2,37 mm

Required board thickness: **3,0 +/- 0,2 mm**

	Layer			Thickness, um	Material
Nr. 5	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			2x1080	150	Prepreg
	INT 1	2		35	Copper
				710	Core
	INT 2	3		35	Copper
			1x7628	185	Prepreg
	INT 3	4		35	Copper
				710	Core
	INT 4	5		35	Copper
			1x7628	185	Prepreg
	INT 5	6		35	Copper
				710	Core
	INT 6	7		35	Copper
			2x1080	150	Prepreg
	BOTTOM	8		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 2,98 mm

10-layer stackups

Required board thickness: **1,3 +/- 0,15 mm**

		Layer			Thickness, um	Material
Nr. 1	SMT				20	Solder mask
	TOP	1			18-25	Copper Plating
					18	Copper Foil
				1x2116	105	Prepreg
	INT 1	2			18	Copper
					130	Core
	INT 2	3			18	Copper
				1x2116	105	Prepreg
	INT 3	4			18	Copper
					130	Core
	INT 4	5			18	Copper
				1x2116	105	Prepreg
	INT 5	6			18	Copper
					130	Core
	INT 6	7			18	Copper
				1x2116	105	Prepreg
	INT 7	8			18	Copper
					130	Core
	INT 8	9			18	Copper
				1x2116	105	Prepreg
	BOTTOM	10			18	Copper Foil
					18-25	Copper Plating
	SMB				20	Solder mask

Approximate board thickness: 1,24 mm

Required board thickness: **1,6 +/- 0,15 mm**

	Layer			Thickness, um	Material
Nr. 2	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			2x1080	150	Prepreg
	INT 1	2		35	Copper
				130	Core
	INT 2	3		35	Copper
			1x2116	105	Prepreg
	INT 3	4		35	Copper
				210	Core
	INT 4	5		35	Copper
			1x2116	105	Prepreg
	INT 5	6		35	Copper
				210	Core
	INT 6	7		35	Copper
			1x2116	105	Prepreg
	INT 7	8		35	Copper
				130	Core
	INT 8	9		35	Copper
			2x1080	150	Prepreg
	BOTTOM	10		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 1,58 mm

Required board thickness: **2,0 +/- 0,2 mm**

		Layer			Thickness, um	Material
Nr. 3	SMT				20	Solder mask
	TOP	1			18-25	Copper Plating
					18	Copper Foil
				2x1080	150	Prepreg
	INT 1	2			35	Copper
					210	Core
	INT 2	3			35	Copper
				1x7628	185	Prepreg
	INT 3	4			35	Copper
					210	Core
	INT 4	5			35	Copper
				1x7628	185	Prepreg
	INT 5	6			35	Copper
					210	Core
	INT 6	7			35	Copper
				1x7628	185	Prepreg
	INT 7	8			35	Copper
					210	Core
	INT 8	9			35	Copper
				2x1080	150	Prepreg
					18	Copper Foil
	BOTTOM	10			18-25	Copper Plating
	SMB				20	Solder mask

Approximate board thickness: 1,94 mm

Required board thickness: **2,4 +/- 0,2 mm**

		Layer		Thickness, um	Material
Nr. 4	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x1080	75	Prepreg
			1x7628	185	Prepreg
	INT 1	2		35	Copper
				250	Core
	INT 2	3		35	Copper
			1x7628	185	Prepreg
	INT 3	4		35	Copper
				250	Core
	INT 4	5		35	Copper
			1x7628	185	Prepreg
	INT 5	6		35	Copper
				250	Core
	INT 6	7		35	Copper
			1x7628	185	Prepreg
	INT 7	8		35	Copper
				250	Core
	INT 8	9		35	Copper
			1x7628	185	Prepreg
			1x1080	75	Prepreg
				18	Copper Foil
	BOTTOM	10		18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 2,32 mm

Required board thickness: **3,0 +/- 0,2 mm**

	Layer			Thickness, um	Material
Nr. 5	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x1080	75	Prepreg
			1x7628	185	Prepreg
	INT 1	2		35	Copper
				360	Core
	INT 2	3		35	Copper
			1x7628	185	Prepreg
	INT 3	4		35	Copper
				360	Core
	INT 4	5		35	Copper
			2x7628	370	Prepreg
	INT 5	6		35	Copper
				360	Core
	INT 6	7		35	Copper
			1x7628	185	Prepreg
	INT 7	8		35	Copper
				360	Core
	INT 8	9		35	Copper
			1x7628	185	Prepreg
			1x1080	75	Prepreg
	BOTTOM	10		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 2,93 mm

12-layer stackups

Required board thickness: **1,6 +/- 0,15 mm**

	Layer			Thickness, um	Material
Nr. 1	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x2116	105	Prepreg
	INT 1	2		35	Copper
				130	Core
	INT 2	3		35	Copper
			1x2116	105	Prepreg
	INT 3	4		35	Copper
				130	Core
	INT 4	5		35	Copper
			1x2116	105	Prepreg
	INT 5	6		35	Copper
				130	Core
	INT 6	7		35	Copper
			1x2116	105	Prepreg
	INT 7	8		35	Copper
				130	Core
	INT 8	9		35	Copper
			1x2116	105	Prepreg
	INT 9	10		35	Copper
				130	Core
	INT 10	11		35	Copper
			1x2116	105	Prepreg
	BOTTOM	12		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 1,65 mm

Required board thickness: **2,0 +/- 0,2 mm**

	Layer			Thickness, um	Material
Nr. 2	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			2x1080	150	Prepreg
	INT 1	2		35	Copper
				130	Core
	INT 2	3		35	Copper
			1x2116	105	Prepreg
	INT 3	4		35	Copper
				210	Core
	INT 4	5		35	Copper
			1x2116	105	Prepreg
	INT 5	6		35	Copper
				210	Core
	INT 6	7		35	Copper
			1x2116	105	Prepreg
	INT 7	8		35	Copper
				210	Core
	INT 8	9		35	Copper
			1x2116	105	Prepreg
	INT 9	10		35	Copper
				130	Core
	INT 10	11		35	Copper
			2x1080	150	Prepreg
	BOTTOM	12		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 1,94 mm

Required board thickness: **2,4 +/- 0,2 mm**

	Layer		Type	Thickness, um	Material
Nr. 3	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x1080	75	Prepreg
			1x2116	105	Prepreg
	INT 1	2		35	Copper
				210	Core
	INT 2	3		35	Copper
			1x7628	185	Prepreg
	INT 3	4		35	Copper
				210	Core
	INT 4	5		35	Copper
			1x2116	105	Prepreg
	INT 5	6		35	Copper
				210	Core
	INT 6	7		35	Copper
			1x2116	105	Prepreg
	INT 7	8		35	Copper
				210	Core
	INT 8	9		35	Copper
			1x7628	185	Prepreg
	INT 9	10		35	Copper
				210	Core
	INT 10	11		35	Copper
			1x2116	105	Prepreg
			1x1080	75	Prepreg
	BOTTOM	12		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 2,32 mm

Required board thickness: **3,0 +/- 0,2 mm**

	Layer			Thickness, um	Material
Nr. 4	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper
			1x1080	75	Prepreg
			1x2116	105	Prepreg
	INT 1	2		35	Copper
				360	Core
	INT 2	3		35	Copper
			1x2116	105	Prepreg
	INT 3	4		35	Copper
				360	Core
	INT 4	5		35	Copper
			1x2116	105	Prepreg
	INT 5	6		35	Copper
				360	Core
	INT 6	7		35	Copper
			1x2116	105	Prepreg
	INT 7	8		35	Copper
				360	Core
	INT 8	9		35	Copper
			1x2116	105	Prepreg
	INT 9	10		35	Copper
				360	Core
	INT 10	11		35	Copper
			1x2116	105	Prepreg
			1x1080	75	Prepreg
	BOTTOM	12		18-25	Copper Plating
				18	Copper
	SMB			20	Solder mask

Approximate board thickness: 2,94 mm

14-layer stackups

Required board thickness: **2,0 +/- 0,2 mm**

	Layer				Thickness, um	Material
Nr. 1	SMT				20	Solder mask
	TOP	1			18-25	Copper Plating
					18	Copper Foil
				1x2116	105	Prepreg
	INT 1	2			35	Copper
					130	Core
	INT 2	3			35	Copper
				1x2116	105	Prepreg
	INT 3	4			35	Copper
					130	Core
	INT 4	5			35	Copper
				1x2116	105	Prepreg
	INT 5	6			35	Copper
					130	Core
	INT 6	7			35	Copper
				1x2116	105	Prepreg
	INT 7	8			35	Copper
					130	Core
	INT 8	9			35	Copper
				1x2116	105	Prepreg
	INT 9	10			35	Copper
					130	Core
	INT10	11			35	Copper
				1x2116	105	Prepreg
	INT11	12			35	Copper
					130	Core
	INT 12	13			35	Copper
				1x2116	105	Prepreg
	BOTTOM	14			18	Copper Foil
					18-25	Copper Plating
	SMB				20	Solder mask

Approximate board thickness: 1,95 mm

Required board thickness: **2,5 +/- 0,2 mm**

	Layer			Thickness, um	Material
Nr. 2	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			2x1080	150	Prepreg
	INT 1	2		35	Copper
				210	Core
	INT 2	3		35	Copper
			1x2116	105	Prepreg
	INT 3	4		35	Copper
				210	Core
	INT 4	5		35	Copper
			1x2116	105	Prepreg
	INT 5	6		35	Copper
				210	Core
	INT 6	7		35	Copper
			1x2116	105	Prepreg
	INT 7	8		35	Copper
				210	Core
	INT 8	9		35	Copper
			1x2116	105	Prepreg
	INT 9	10		35	Copper
				210	Core
	INT10	11		35	Copper
			1x2116	105	Prepreg
	INT11	12		35	Copper
				210	Core
	INT 12	13		35	Copper
			2x1080	150	Prepreg
				18	Copper Foil
	BOTTOM	14		18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 2,48 mm

Required board thickness: **3,0 +/- 0,2 mm**

	Layer			Thickness, um	Material
Nr. 3	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x1080	75	Prepreg
			1x7628	185	Prepreg
	INT 1	2		35	Copper
				210	Core
	INT 2	3		35	Copper
			1x7628	185	Prepreg
	INT 3	4		35	Copper
				210	Core
	INT 4	5		35	Copper
			1x7628	185	Prepreg
	INT 5	6		35	Copper
				210	Core
	INT 6	7		35	Copper
			1x7628	185	Prepreg
	INT 7	8		35	Copper
				210	Core
	INT 8	9		35	Copper
			1x7628	185	Prepreg
	INT 9	10		35	Copper
				210	Core
	INT10	11		35	Copper
			1x7628	185	Prepreg
	INT11	12		35	Copper
				210	Core
	INT 12	13		35	Copper
			1x7628	185	Prepreg
			1x1080	75	Prepreg
	BOTTOM	14		18	Copper Foil
				18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 3,0 mm

16-layer stackups

Required board thickness: **2,2 +/- 0,2 mm**

	Layer				Thickness, um	Material
Nr. 1	SMT				20	Solder mask
	TOP	1			18-25	Copper Plating
					18	Copper Foil
				1x2116	105	Prepreg
	INT 1	2			35	Copper
					130	Core
	INT 2	3			35	Copper
				1x2116	105	Prepreg
	INT 3	4			35	Copper
					130	Core
	INT 4	5			35	Copper
				1x2116	105	Prepreg
	INT 5	6			35	Copper
					130	Core
	INT 6	7			35	Copper
				1x2116	105	Prepreg
	INT 7	8			35	Copper
					130	Core
	INT 8	9			35	Copper
				1x2116	105	Prepreg
	INT 9	10			35	Copper
					130	Core
	INT10	11			35	Copper
				1x2116	105	Prepreg
	INT11	12			35	Copper
					130	Core
	INT13	13			35	Copper
				1x2116	105	Prepreg
	INT14	14			35	Copper
					130	Core
	INT 15	15			35	Copper
				1x2116	105	Prepreg
					18	Copper Foil
	BOTTOM	16			18-25	Copper Plating
	SMB				20	Solder mask

Approximate board thickness: 2,18 mm

Required board thickness: **2,5 +/- 0,2 mm**

	Layer			Thickness, um	Material
Nr. 2	SMT			20	Solder mask
	TOP	1		18-25	Copper Plating
				18	Copper Foil
			1x1080	75	Prepreg
			1x7628	185	Prepreg
	INT 1	2		35	Copper
				130	Core
	INT 2	3		35	Copper
			1x2116	105	Prepreg
	INT 3	4		35	Copper
				130	Core
	INT 4	5		35	Copper
			1x2116	105	Prepreg
	INT 5	6		35	Copper
				130	Core
	INT 6	7		35	Copper
			1x2116	105	Prepreg
	INT 7	8		35	Copper
				130	Core
	INT 8	9		35	Copper
			1x2116	105	Prepreg
	INT 9	10		35	Copper
				130	Core
	INT10	11		35	Copper
			1x2116	105	Prepreg
	INT11	12		35	Copper
				130	Core
	INT13	13		35	Copper
			1x2116	105	Prepreg
	INT14	14		35	Copper
				130	Core
	INT 15	15		35	Copper
			1x7628	185	Prepreg
			1x1080	75	Prepreg
				18	Copper Foil
	BOTTOM	16		18-25	Copper Plating
	SMB			20	Solder mask

Approximate board thickness: 2,48 mm

Required board thickness: **3,0 +/- 0,2 mm**

	Layer				Thickness, um	Material
Nr. 3	SMT				20	Solder mask
	TOP	1			18-25	Copper Plating
					18	Copper Foil
				1x1080	75	Prepreg
				1x2116	105	Prepreg
	INT 1	2			35	Copper
					210	Core
	INT 2	3			35	Copper
				1x2116	105	Prepreg
	INT 3	4			35	Copper
					210	Core
	INT 4	5			35	Copper
				1x2116	105	Prepreg
	INT 5	6			35	Copper
					210	Core
	INT 6	7			35	Copper
				1x2116	105	Prepreg
	INT 7	8			35	Copper
					210	Core
	INT 8	9			35	Copper
				1x2116	105	Prepreg
	INT 9	10			35	Copper
					210	Core
	INT10	11			35	Copper
				1x2116	105	Prepreg
	INT11	12			35	Copper
					210	Core
	INT13	13			35	Copper
				1x2116	105	Prepreg
	INT14	14			35	Copper
					210	Core
	INT 15	15			35	Copper
				1x2116	105	Prepreg
				1x1080	75	Prepreg
	BOTTOM	16			18	Copper Foil
					18-25	Copper Plating
	SMB				20	Solder mask

Approximate board thickness: 2,95 mm

4. Via holes plugging

On a printed circuit board, the metallized holes, intended for assembling of through-hole components, always remain open from under the solder mask. But if we are talking about the vias, it is important to determine the method of their manufacture - whether they will be opened from under the mask or will be covered with a solder mask (tented) or they will be filled with the appropriate material (plugged).

Usually there are several possible approaches: leaving the vias open under the mask, tenting vias, filling the vias with mask or non-conductive resin (mask plugged via), filling the vias with conductive paste or copper (plated via). All methods have advantages and disadvantages.

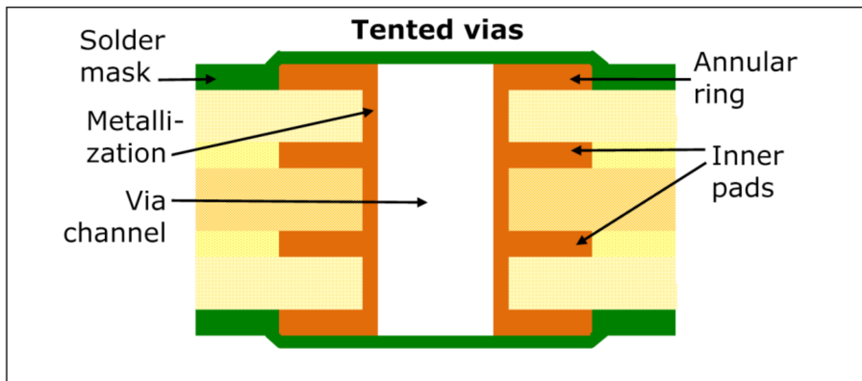
The most expensive option is to fill the vias with conductive materials or copper. The main disadvantage of this method is that such vias have an expansion coefficient different from laminate, which creates a risk of defects. Therefore, this technology is used only if strict requirements are imposed on the thermal conductivity of the vias interconnections.

Below there is a description of the technologies used.

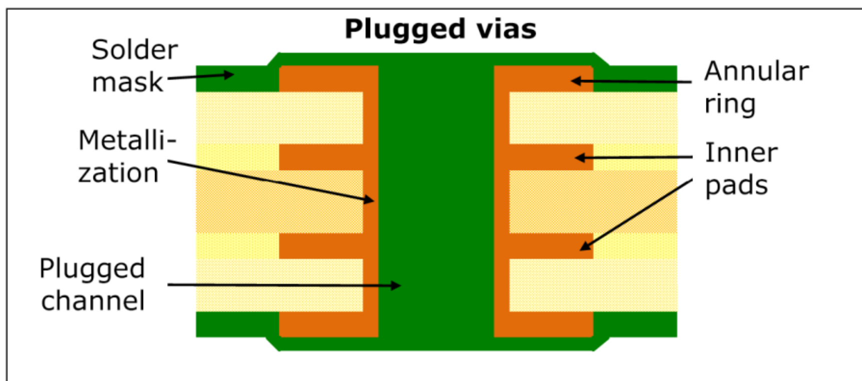
Tenting vias. This is the cheapest and simplest technology. According to this method, the via hole and the copper ring (annual ring) are covered by solder mask. When using this method, the hole channel is not filled. Vias with a diameter of 0,3 mm and less will certainly be covered, while vias with a diameter of 0,4 - 0,6 mm may also be covered, but with less probability. All holes above these diameters are most likely to have leaks and will only be partially concealed.

The main purpose of covering the vias with a solder mask is to prevent short circuits between the vias pads and other conductive elements.

Please note that vias must be covered with a solder mask on both sides of the PCB, otherwise there is a risk of contamination of the hole channel with chemical residues.

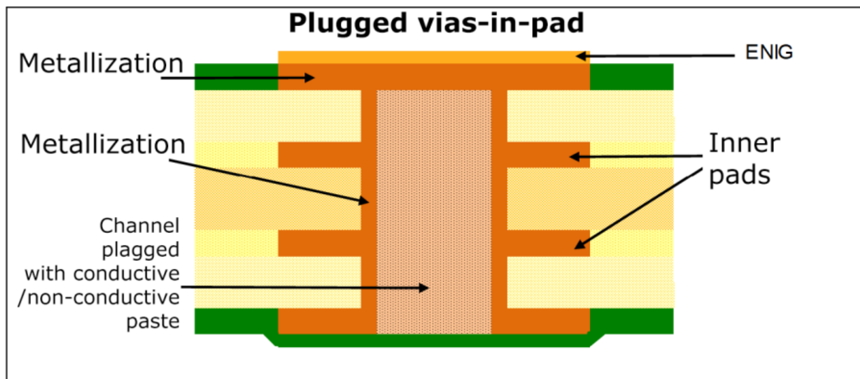


Vias filled with solder mask or non-conductive resin (mask plugged via). When using this technology, the vias are completely covered, and the holes are also plugged. This technology is very often used in BGA projects, where there is a risk of a short circuit during soldering between closely spaced vias and the BGA pads.

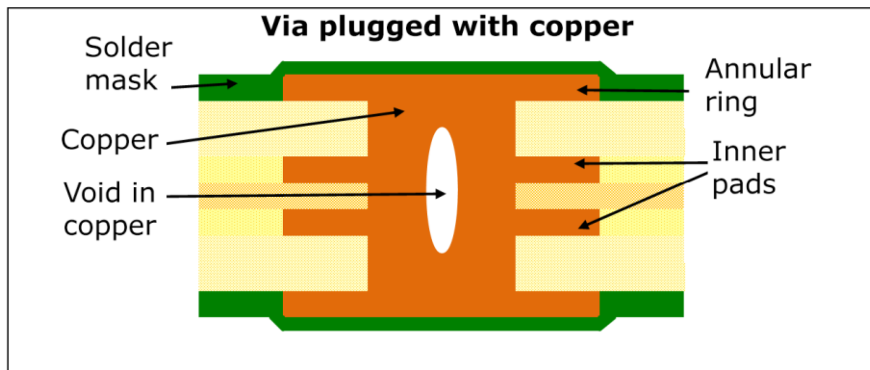


Vias-In-Pad (Active Pads). This technology is used in BGA designs with a small raster. The via holes are drilled directly in the solder pads of SMT components, which saves the PCB area and facilitates signal routing. The via holes are filled with conductive or non-conductive materials. After filling the holes, the surface of the vias is covered with copper, resulting in a good contact plane and facilitating the soldering of the most complex components.

The choice of conductive or non-conductive materials is based, on the one hand, on the requirements for heat dissipation and, on the other hand, on the maximum safety of interlayer connections, since non-conductive materials have a coefficient of thermal expansion more suitable for laminate than conductive materials. The difference in thermal coefficients leads to the risk of delamination in the case of excessively hot electronic components. The technology itself is not complicated.



Vias Plugged with Copper. The technology is based on via through holes plugged with copper, which improves their thermal and electrical conductivity. But the big disadvantage of this technology is the difficult-to-reach uniformity of the coating of through channels, which creates the risk of air voids that will outgas during soldering, compromising the uniformity of the soldering connections. In addition, the technology is more difficult to implement and, therefore, more expensive.



When ordering or requesting price quotes, we recommend to specify which technology is required, as they differ in price.

5. Copper thickness in multilayer PCB

Materials with a basic copper thickness of 18 or 35 microns are used for the inner layers of printed circuit boards. The thickness of copper does not change during the manufacture of the printed circuit board, since it is hidden internally during metallization of the stack of layers.

For the outer layers, materials with a copper thickness of 9...35 microns can be used. However, it must be borne in mind that during metallization, copper is applied to the holes, as well as to the entire surface of the panel, which increases the thickness of the outer layers by another 10 ... 25 microns. As a result, the final thickness of copper on the outer layers can reach 18...70 microns.

The choice of the outer copper foil thickness depends on the requirements for the width of the conductors and the spacings between the conductive elements on the printed circuit board.

Below there are the conductor/spacing restrictions on the PCB for different thicknesses of copper:

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

6. PCB fabrication 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)	

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 / 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]	

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

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