

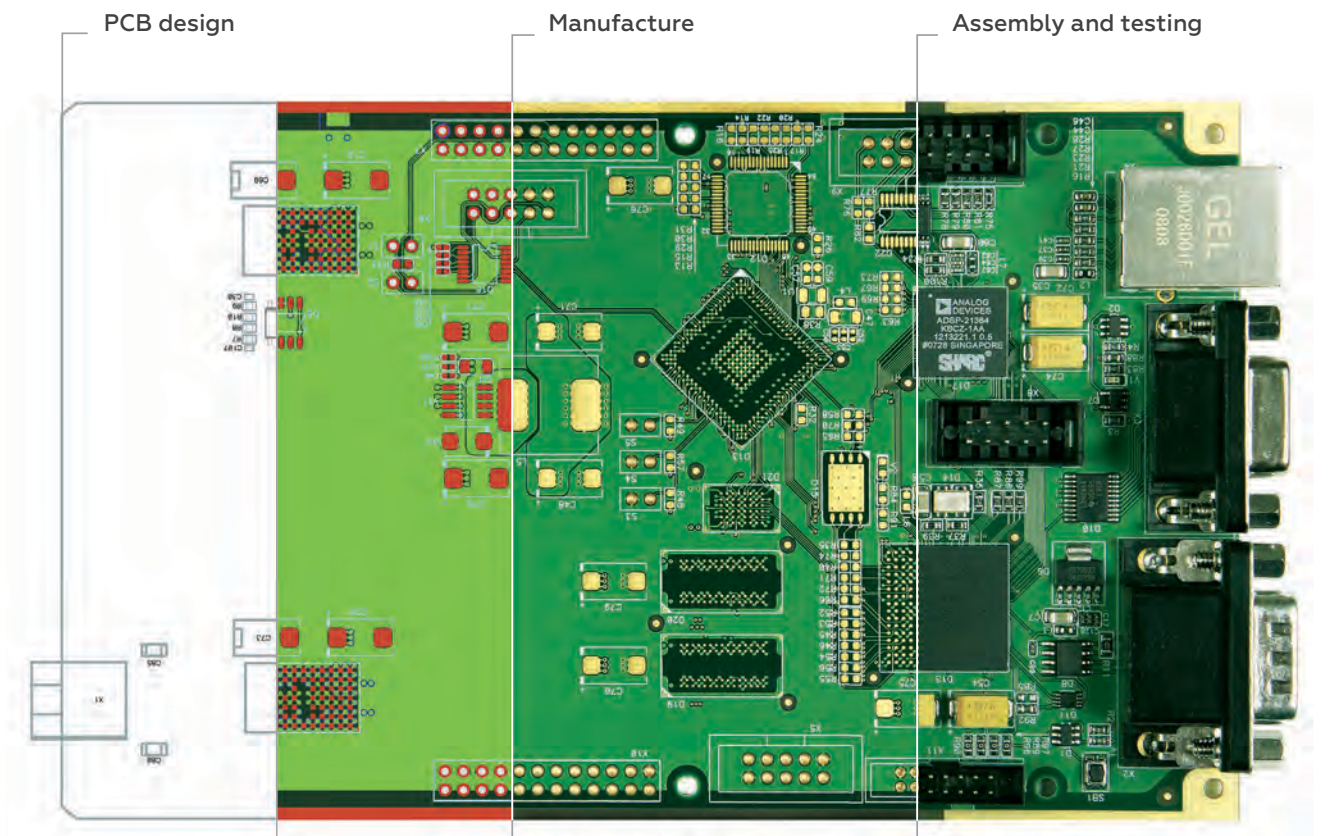
PCBtechnology



Reliable technology

Design, manufacture
and assembly of the multilayer
printed circuit boards

Services of the holding



The first integrated design made by PCB Technology according to the customer's specification. The electronic radio telescope control module, designed and manufactured for the Special Astrophysical Observatory of the Russian Academy of Sciences

PCB technology is a production holding company which includes the contract manufacturer of printed circuit boards and electronic modules, the design bureau, and the supplier of the eCAD software.

1997

- ✓ Supply of multilayer PCBs from USA

2000

- ✓ Start of partner relations with the Russian NPO – PCB manufacturer
- ✓ Supply of multilayer PCBs from the South Korea

2005

- ✓ Supply of large quantities of the flexible and flex-rigid printed boards
- ✓ Establishment of the office in China to ensure the PCB quality control

2010

- ✓ Start of Cadence Allegro PCB design software sales

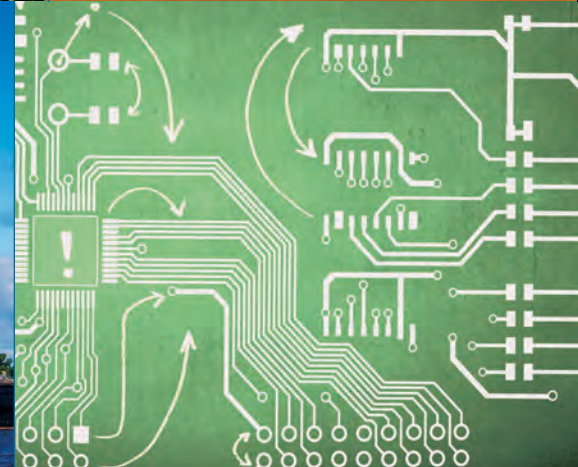
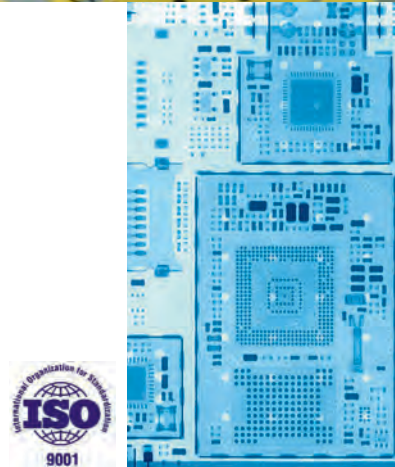
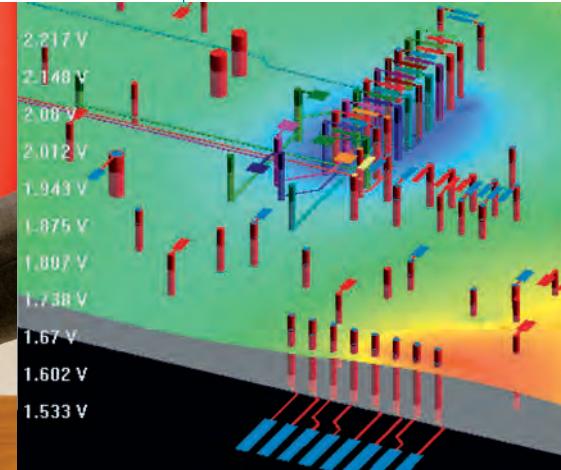
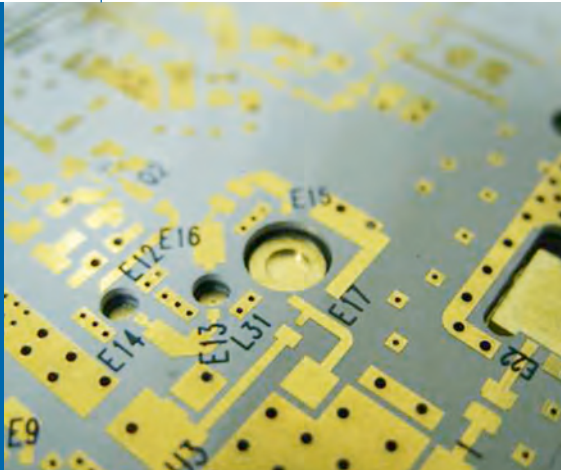


2011

- ✓ New assembly shop is opened
- ✓ IPC Engineer Certification
- ✓ IPC membership acquired

2014

- ✓ The quality management system is certified to comply with ISO9001 standard



2006

- ✓ ISO 9001 Certification
- ✓ ERP SAP introduction

2007

- ✓ Full cycle of PCB contract manufacturing
- ✓ Establishment of the second office in Saint-Petersburg

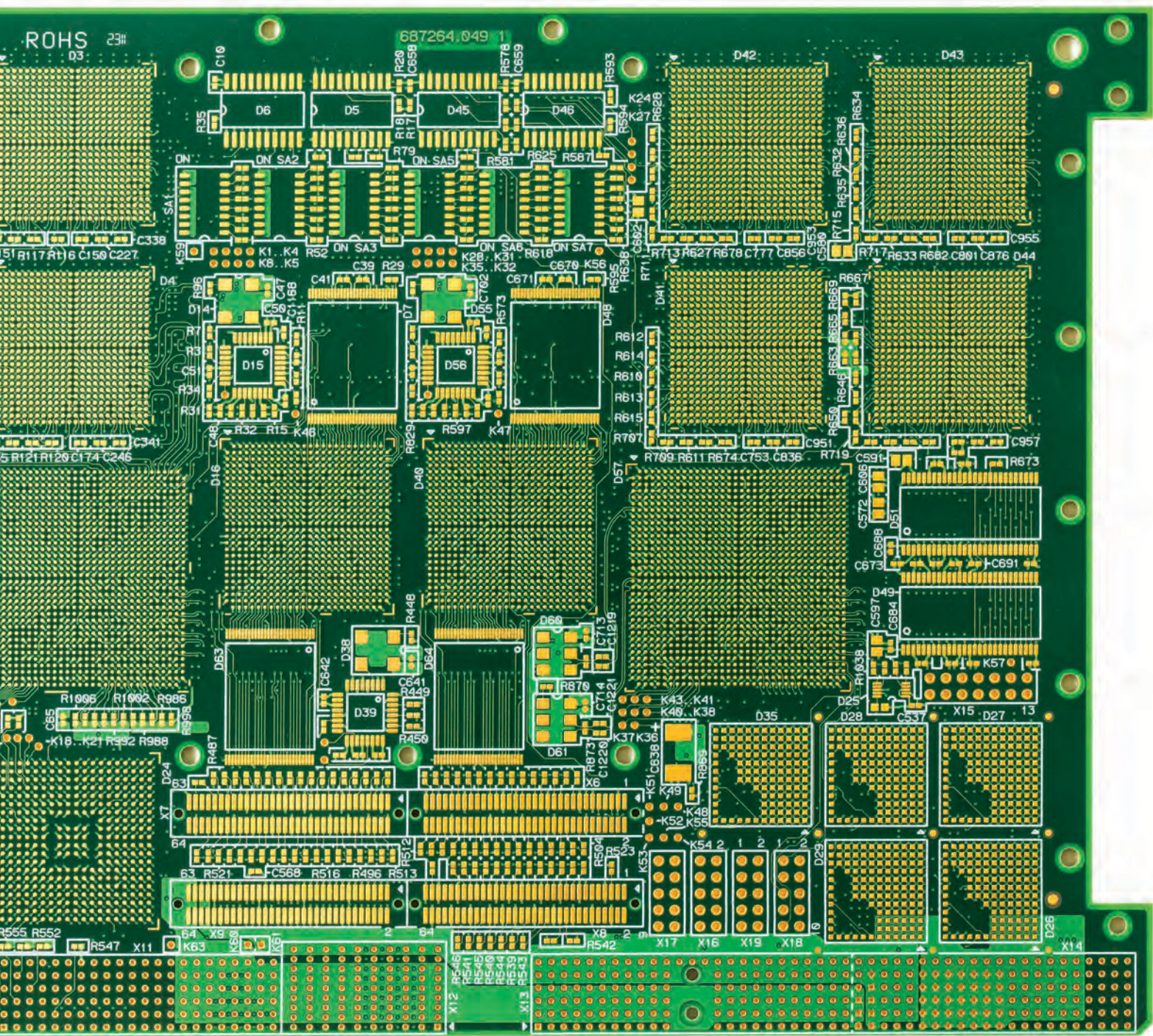
2009

- ✓ Settlement of PCB routing design bureau

2013

- ✓ Launch of a second automatic surface mount line with a capacity of 100,000 elements per hour

We don't rest on our laurels. Every year brings some changes, such as new technologies, new needs from our customers. We learn and we teach others to work with new materials and technologies.



Multilayer PCB 16 layers on FR4 High Tg, 3 mil trace/space
Immersion gold, Aspect Ratio 1:10

Tolerance for Press-Fit holes: +/- 0,05 mm.

Control of the differential pair impedance 100 Ohm +/- 7%.

Capacities of the Holding

PCB manufacturing

- Complex multilayer printed boards and HDI-boards
- Flexible and flex-rigid printed boards
- Multilayer boards on Rogers, Arlon and Taconic microwave-materials
- Special printed boards, including printed on metal base

PCB Assembly

- The assembly is focused on fulfilling the most complex orders with high demands for product quality
- BGA assembly/ dismounting, X-ray control, press-fitting and forming of "gull wing" terminals

Purchase of BOM

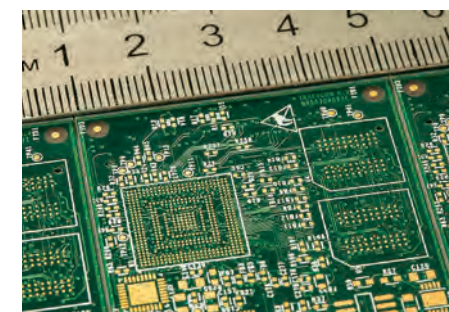
- Complete sets of electronic components are purchased for complex orders

Printed board design

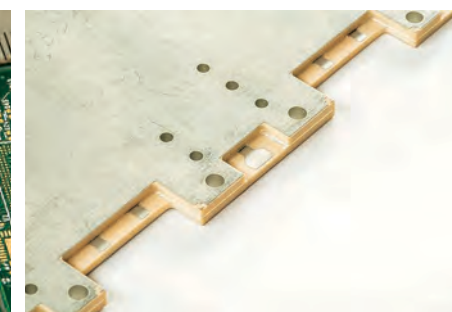
- We route printed boards according to the customer's circuit layout with guaranteed quality and in short time

Supply of the eCAD for printed circuit boards layout

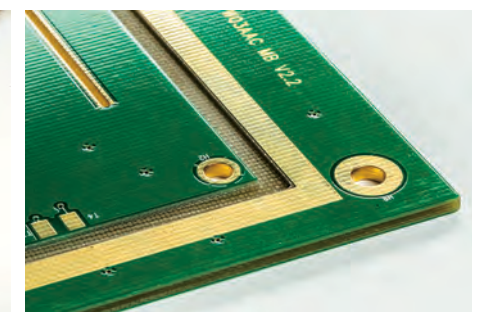
- We supply the Cadence Allegro CAD to our customers in Russia to design the top level printed circuit boards



Multilayer PCB 12 layers HDI with micro-BGA (pitch 0.4 mm)



Slots on the edge of the microwave board. Material Ro4350, 3 layers



Milling at a given depth



The Europlacer IINEO Tornado SMT-mounting machine allows for simultaneous installation of up to 250 part numbers

The holding specialists are professionals in the field of complex printed boards and electronic modules supply. This is what makes us different from our competitors who are engaged in orders of "any" type.

Advantages of working with PCB technology holding

PCB technology customers have the following benefits:

- Technical support by engineers with many years of experience
- Specialization in the implementation of complex projects
- Guaranteed product quality and reliability
- Guaranteed order fulfillment in time
- Own assembly shop is located in Moscow, so the logistics is easy
- Consultations and regular seminars on designing of complex printed circuit boards

Reliability and durability of printed circuit boards are ensured due to full compliance with the requirements of IPC and GOST standards and strict quality control.

Our printed circuit boards have excellent solderability and maintainability, which is confirmed by the samples and control protocols attached to each order. At the same time, we offer very short delivery times and guarantee their fulfillment.

PCB technology holding is an official distributor of the world's leading factories - manufacturers of complex multilayer printed circuit boards. Our company is one of the few direct legal importers of printed circuit boards, according to customs statistics, it is in third place in terms of imports of printed circuit boards in Russia. This ensures that client enterprises do not have problems with tax and counter checks and confirms the full payment of VAT and customs duties.

PCB Technology LLC is a Russian legal entity with Russian founders, not a representative office of foreign firms or a one-day company.

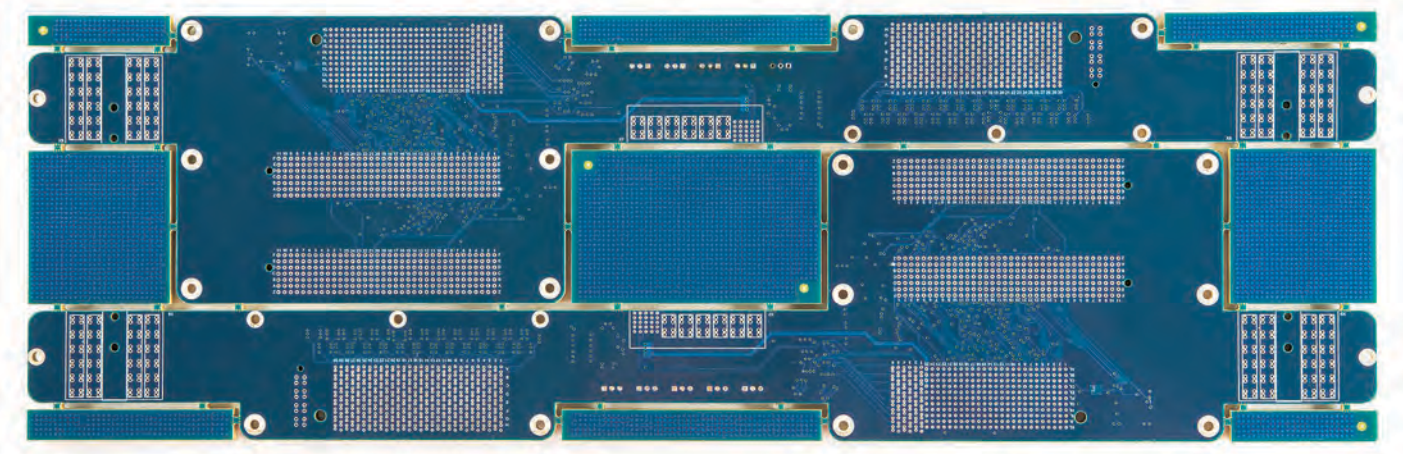
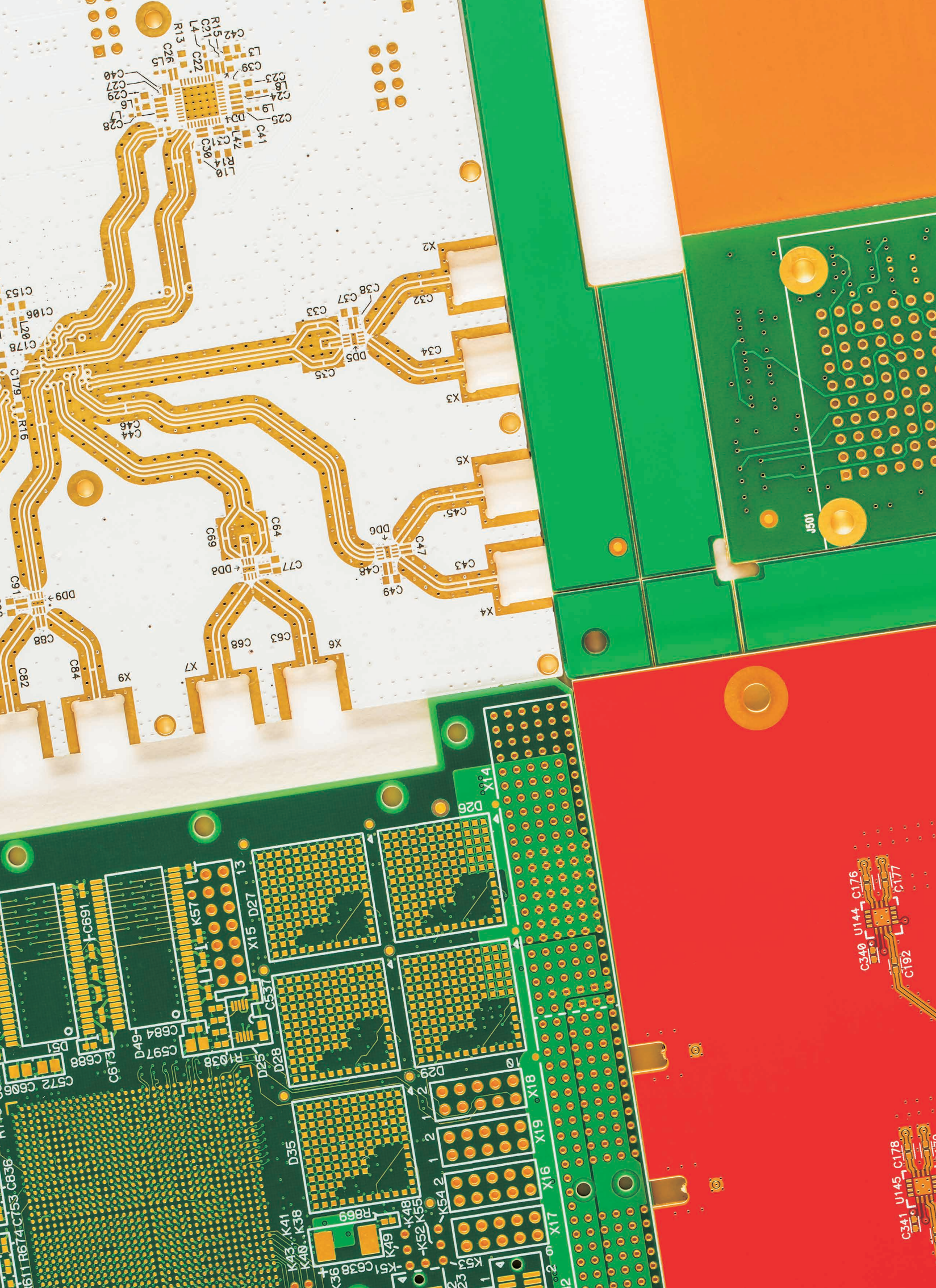
The technological capabilities of the holding allow us to successfully fulfill the most complex orders with the highest parameters.



Quality of BGA components assembly is confirmed by an X-ray inspection



Visual quality control and micro-soldering adjustments support the AOI system automated quality control



Production of multilayer printed circuit boards

Backplane with connectors for press-fit mounting and impedance control

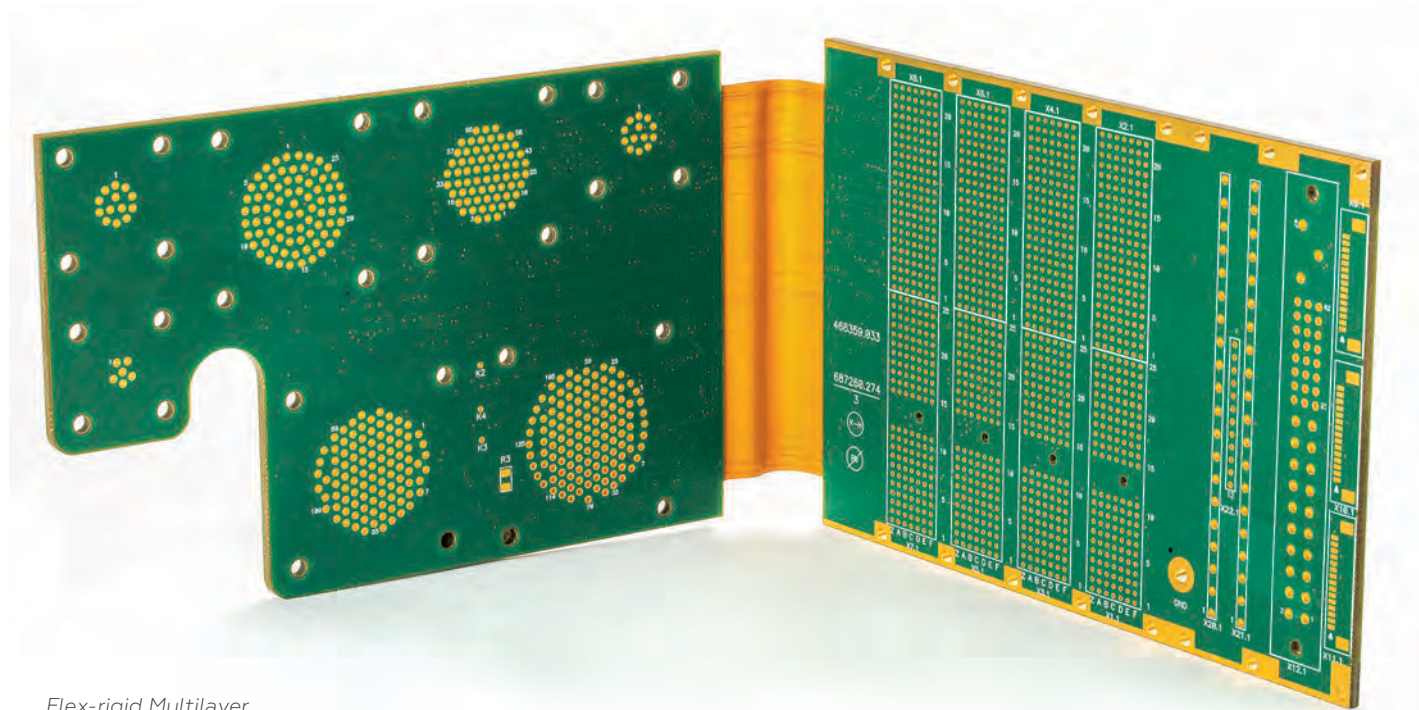
We deliver high-quality PCBs, paying maximum attention to their reliability and durability. With the help of a well-established production process, selection of the best materials, quality control and smooth logistics, we ensure short delivery times and competitive prices. All boards pass additional check for good solderability and flatness, as well as the strictest quality control of metallization in the hole.

Every week we deliver hundreds of different models of printed circuit boards to our Russian customers:

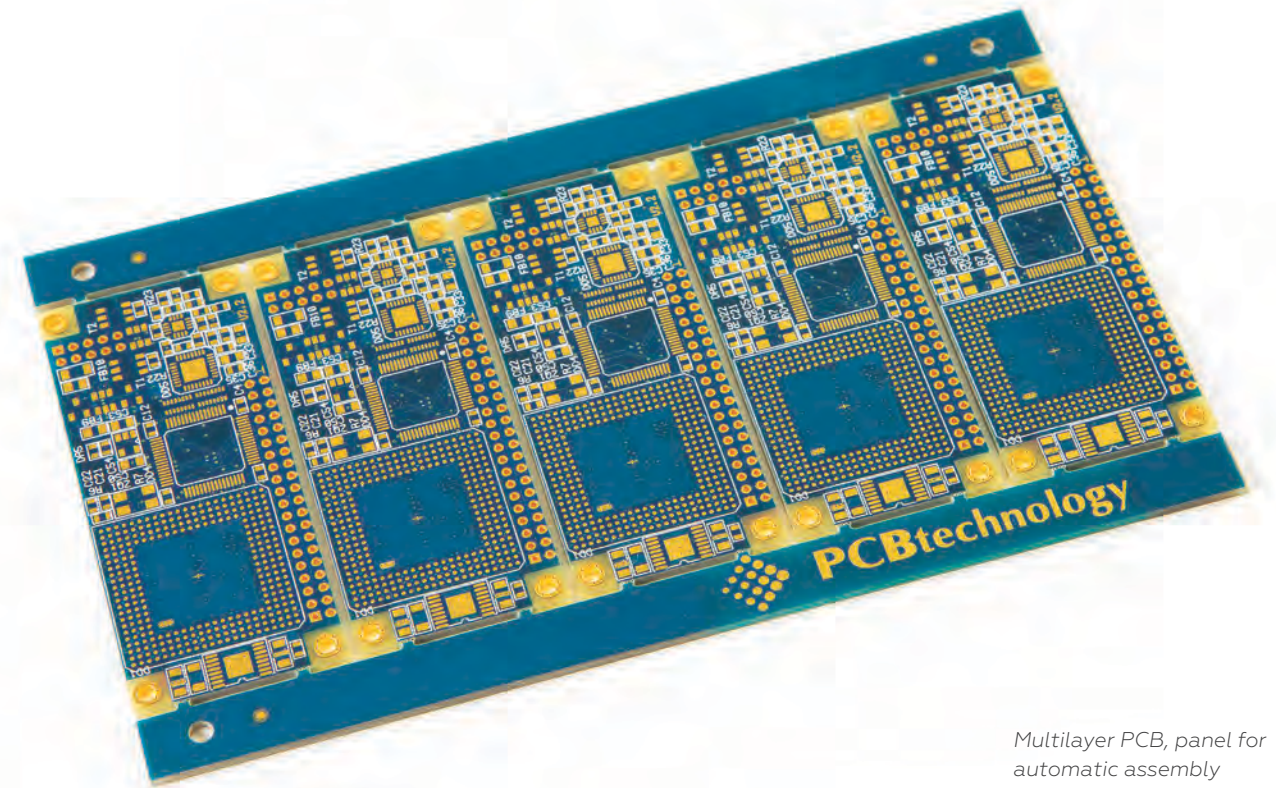
- with blind and buried vias;
- with impedance control;
- flexible and flex-rigid printed circuit boards;
- special PCB (RF, circuit boards on metal, etc.)

PCB lead time

Small batches	3 weeks
Large series	4-5 weeks
Urgent Prototypes	2,5-3 weeks
Super express	2 weeks



Flex-rigid Multilayer
Printed Circuit Board



Multilayer PCB, panel for
automatic assembly

We use RF materials, polyimide, as well as various special types of materials.

File check

Prior to production process, each order undergoes an engineering check to identify potential problem areas in the customer's design documentation.

Technical Advice

Calling us by phone or sending a message by e-mail, you can get a free consultation on any technical issues. Feel free to ask, especially if you are designing a new PCB structure or applying more stringent parameters than usual.

RF board with strip lines
coming out of the inner
layer



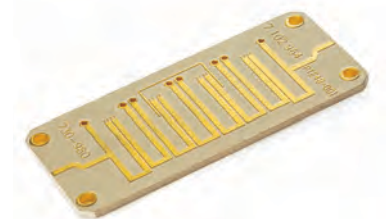
Prior to production process, every order undergoes an engineering check to identify potential problem areas in the customer's design documentation.

Materials

We use high-quality FR4 material for double-sided and 4-layer boards. For boards of 6 and more layers we use FR4 High Tg – a more stable material resistant to high soldering temperature, especially in lead-free reflow process. Such boards are much more reliable than conventional ones due to better performance of copper interlayer connections in vias, but their cost is higher than that of conventional boards.

Note. If the board has lead-free BGA components and it is planned to assemble it in lead-free mode (heating to 250-260°C), then the use of FR4 High Tg is mandatory! Conventional boards on FR4 material will not withstand such soldering temperature!

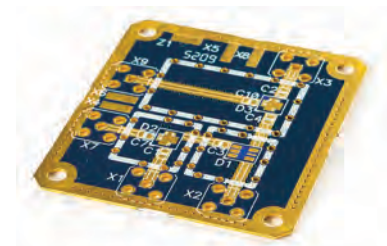
Special materials. In addition, we use microwave materials, polyimide, as well as various special types of materials. It is possible to combine several types of materials and ordinary FR4, making "composite" (hybrid) multilayer boards.



RF 2 layers, Ro4350B



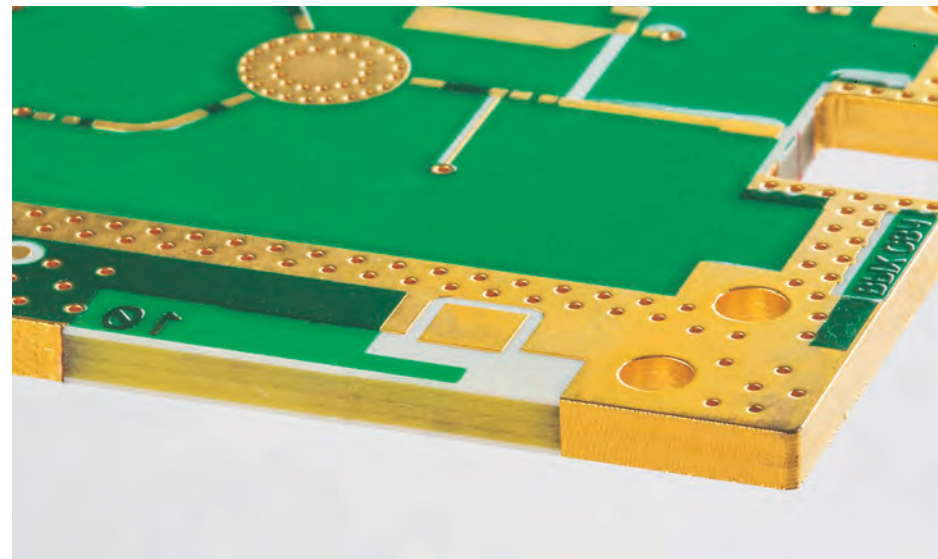
Flex cable with mounted
connectors



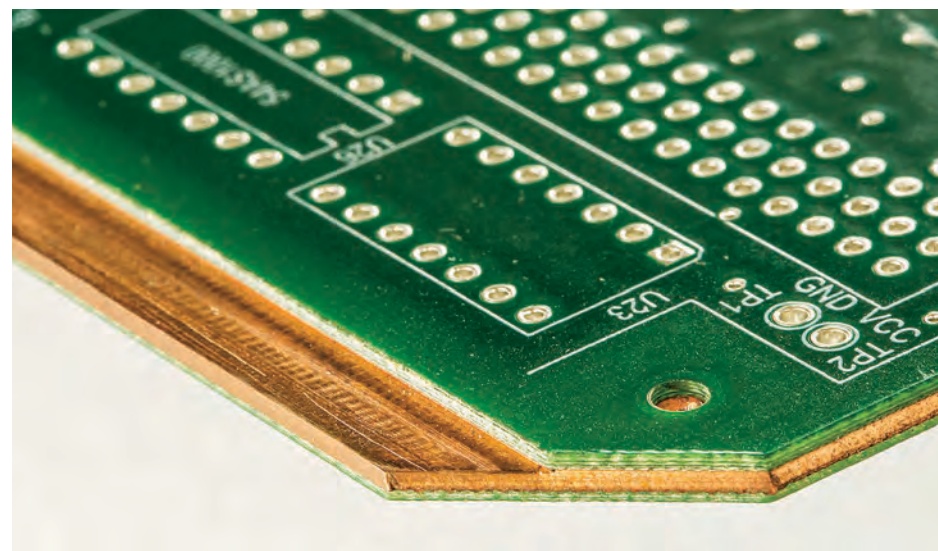
PCB with plated edges

Technology capabilities

Parameters and units	Standard	Advanced
Number of layers	24	64
Board thickness, mm	0,3...3,2	0,1...8,0
Maximum board size, mm	500x800	1000x1100
Ratio of the diameter of the metal vias to the thickness of the board	1 : 10	1 : 12...1 : 20
Minimum trace and space	0,1	0,075...0,05
Minimum annular ring of the hole (from inside diameter), mm	1 : 12...1 : 20	0,1



Multilayer PCB with layers of RF material



Multilayer PCB 6 layers with 1mm thick heat-trapping copper core between the 3rd and 4th layers. Blind holes L1-L2, L1-L3, L5-L6, L4-L6 and plated through holes via the copper core

Parameters and units	Standard	Advance
Minimum distance from hole to copper in inner layers (from the drill to the conductor / polygon), mm	0,2	0,15
The minimum diameter of the via, mm	0,2 (drill 0,25)	0,1 (drill 0,15)
The minimum diameter of the laser blind vias, mm	0,15	0,1...0,075
The ratio of the depth of laser hole to its diameter	1 : 1,2	1 : 1
Minimum opening from the pad to the mask (on the side), mm	0,075	0,025
Board Thickness Tolerance	+/- 10%	+/- 5%
Board size tolerance, mm	+/- 0,1	+/- 0,05
Impedance Control Tolerance	+/- 10%	+/- 7%...+/- 5%
Plated hole diameter tolerance, mm	+/- 0,1	+/- 0,1
Diameter tolerance of the press-fit hole, mm	+/- 0,05	+/- 0,05
Non-plated hole diameter tolerance, mm	+/- 0,1	+/- 0,1
Tolerance of the trace width	+/- 30%	+/- 10%
Minimum width of silkscreen line, mm	0,15	0,1

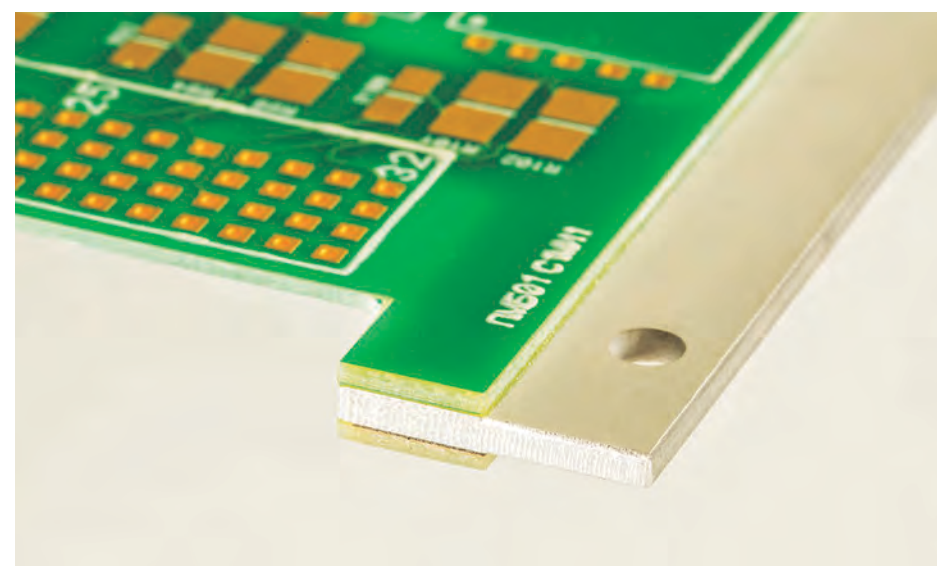


Composite PCB, top layer Ro4003, edge plating

Parameters and units	Standard	Advanced
Minimum height of text in silk-screening	1,25 mm	1 mm
Filling of the holes with resin	0,2...0,45 mm	0,2...0,55 mm
Filling of the holes and copper capping	0,2...0,45 mm	0,2...0,55 mm
PCB quality control standard	IPC-2221, class 2	IPC class 3, GOST 23752
Blind and buried holes, HDI structure	Laser vias, layer-by-layer build up	2+N+2, 3+N+3

Additional options

Series (sequence) numbers on each PCB	Height 2...2.5 mm	Height 1 mm
Plated blind slots	No	Yes
Plated edge	No	Yes
Types of PCB surface	HASL, immersion gold, gold plating, immersion tin, immersion silver	ENEPIG, combined (HASL+gold, gold =solder gold, etc.)

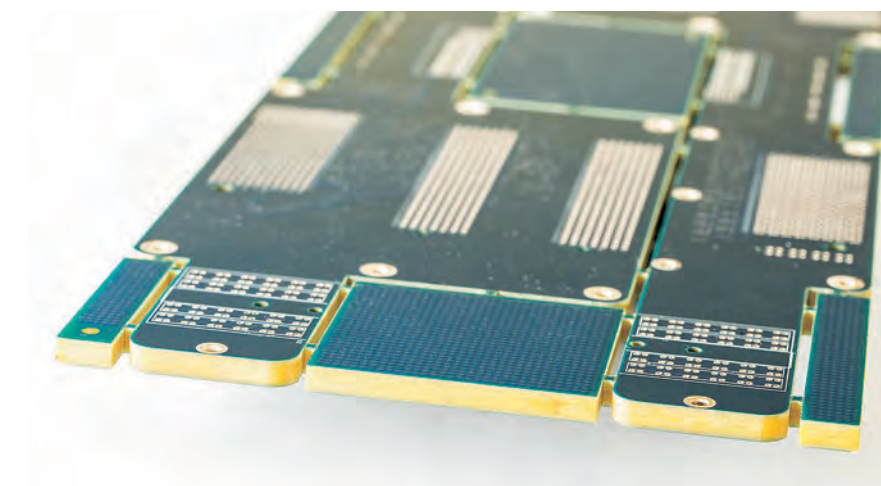


PCB with heat dissipation
Alu core

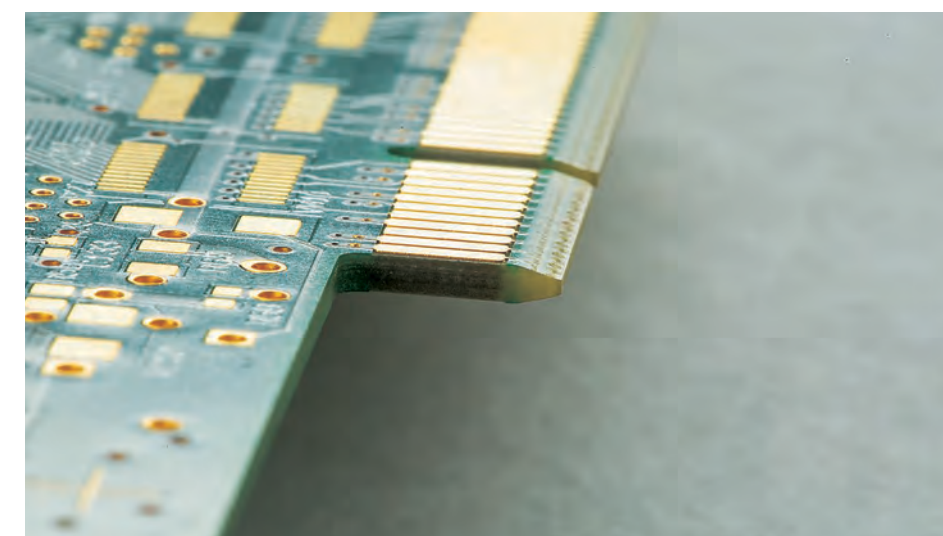
Available materials

Glass-fiber laminate FR4	ITEQ, Shengyi, Isola, TUC
Microwave materials	Rogers, Taconic, Arlon, TUC, Nelco, Megtron
For flexible PCB	DuPont series, TaiFlex, Panasonic
Polyimide for multilayer PCB	Arlon 35N, 85N

For PCB with 6 or more layers we apply FR4 High Tg – a stable material, resistant to high soldering temperature, especially in lead-free reflow modes.



Multilayer PCB, 4.2 mm thick



Multilayer PCB with PCI slot beveling

The assembly workshop located in Moscow houses state-of-art precision surface-mounted assembly lines, manual assembly area, cleaning system, Automatic Optical Inspection (AOI) and an X-ray control unit.

New precision surface-mounted assembly line features two assembly units with a total capacity of up to 115,000 of components per hour, a state-of-art ERSA 3/20 reflow oven and an in-line AOI unit for assembly quality control.



Assembly

We offer quick and high-quality assembly service for complex PCBs.

The assembly workshop is equipped with modern equipment from the top manufacturers. The workshop includes two automated assembly lines, manual assembly area, reflow ovens, vapor-phase oven, two automatic optical inspection stations, an X-ray control unit, and other necessary equipment.

This allows us to assemble technologically sophisticated PCB for pilot, small and medium-scale production. We will be glad to see you among our customers.

Lead time

Small quantities	3 weeks
Big quantities	4–5 weeks
Express prototypes	2–2,5 weeks
Super-express	1 week

Components procurement

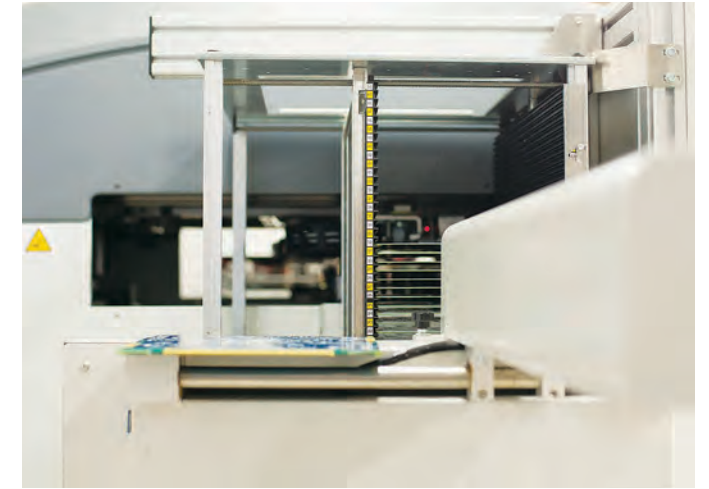
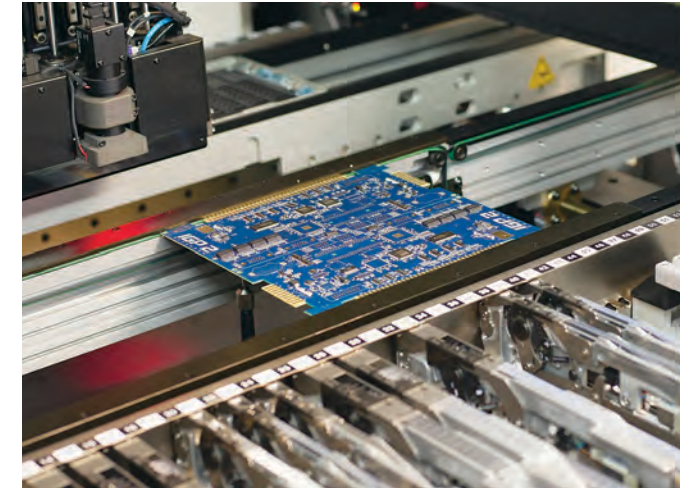
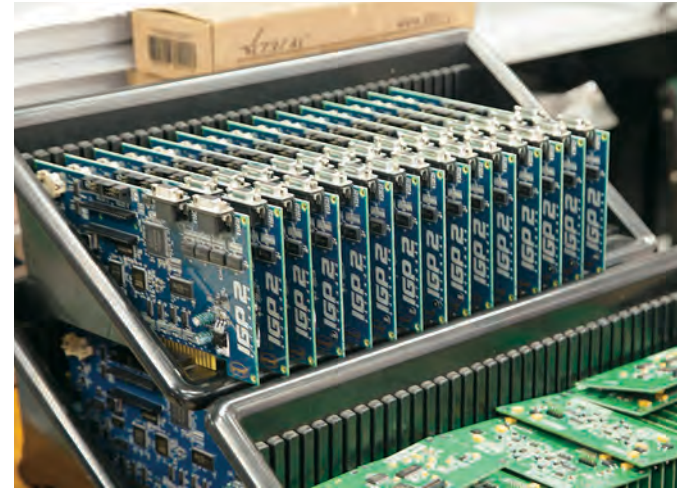
Take advantage of our components procurement for your PCBs. In this case, we provide discounts for all stages of contract manufacturing and guarantee the correctness of the assembly preparation for mounting. We purchase electronic components only from authorized distributors and guarantee high quality at competitive prices.

Requirements for PCBs

- Material – FR4 High Tg recommended, FR4 allowed;
- the solder mask must be applied over the copper;
- HASL coating, lead-free HASL or immersion gold;
- vias must not be located on SMT pads or close to them.

For assembly on automated units, it is desirable to combine boards in panels with holding tabs.

- the best size of a panel 200x250 mm (for panels with thickness of 1,5 mm);
- there should be three reference marks (fiducials) at the corners of the panel for automated assembly;
- on the both sides of the board, you need margins of at least 10mm, or a distance of at least 5mm from the edge, free of SMT components – for secure fixing to the conveyor



Documentation Requirements

1. Bill of Materials (BOM) and list of replacements: Excel, Word format.
2. Packing list for components (or transfer statement): Excel, Word format.
3. PCB file: Gerber, P-CAD, ACCEL EDA, OrCAD, Cadence Allegro, Altium, ODB ++ formats.
4. Stencil file: Gerber format.
5. Pick & Place file (coordinate program for the SMT robot): TXT, Excel, Word formats.
The file contains the coordinates of the centers of the XY components, rotation angle, side, and Part Number.
6. Assembly drawing: AutoCAD or PDF format.

BOM is the basic document for assembly, which is carried out in accordance with it. Supporting documents are: transfer list, replacement list, program for the SMT machine, assembly drawing file, and other documentation, if available. In case of discrepancies, the BOM has the advantage.

Requirements for the preparation of the set of materials

- Components for assembly should be delivered in factory packing (tapes, pallets, tubes) with the same key orientation;
- BGA components must be delivered in a vacuum package;
- For components in tapes, a 3% margin and a free insertion tail of 50 mm are needed;
- For other components, a 1% margin is needed, but not less than 1pc. - for components 0201 and less - a margin of 5% and a free insertion tail of 100 mm are needed.

Each electronic component part number must be packaged in a separate box, bag, or reel with a sealed end, with a clearly legible name and the indicated quantity correctly reflected in the transfer statement. Substitutions and their permissibility should be reflected in the transferred documentation.

The use of tapes without free tails is agreed upon and paid by the customer additionally.

Capabilities of Assembly Factory

Service Description	Parameters
Mounting large boards	up to 610x450x6.0 mm
Minimum Panel Dimensions	50x50 mm
BGA assembly	Pitch down to 0.4 mm
X-ray control BGA	Nanofocus with viewing at any angle
Mounting small passive components	0402, 0201 and 01005
Press-Fit connectors	Yes. It is possible to manufacture the tooling for non-standard connectors
Assembly of CCGA, COB, CSP, LGA non-lead packages	Yes, with the possibility of X-ray control
Mixed mounting BGA (lead and lead free)	Yes, for lead solder paste according to the average thermal profile
BGA repair and reballing	Yes, the pitch of the balls is from 1 to 0.3 mm
Double sided mirror mounting BGA	Yes
Mounting "package on package"	Yes
PCB washing	Yes, subject to customer instructions. It is required to indicate in advance the type of soldering – non-wash or water-wash
Mounting Flip chip, MCM, MEM, uBGA	Yes
Lead-free mounting	Yes
Lead forming CQFP	Yes, such as "gull wing", according to customer drawings
Varnishing	By Subcontractor
Manufacturing of mechanical parts	By Subcontractor
Assembly and packaging	Yes



Design of printed circuit boards as per the customer's circuit diagram

The holding company includes the design center for the routing of printed circuit boards based on electrical diagrams and technical specification. In 2009, the design center became the laureate of the Development of the Year nomination, established by the Electronic Components magazine, for its achievements in the field of high-quality and quick design of customized complex printed circuit boards.

The professional level of design bureau's employees allows optimal interaction with the customers, understanding all the nuances of the project, and even suggesting the best options for technical solutions when designing complex printed circuit boards.

Design center engineers possess knowledge in the field of digital, analog and high-frequency circuitry, concepts of signal integrity and electromagnetic compatibility. When developing the topology of the printed circuit board, they are based on the requirements of the IPC standards for providing technology in the manufacture and assembly of printed circuit boards (DFM and DFA).

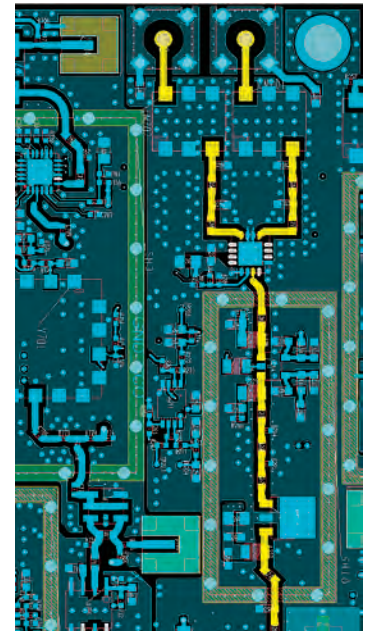
Design features

- Multilayer printed circuit boards (up to 20 layers) with BGA-cases (pitch down to 0.4 mm)
- Blind and buried vias, HDI (High Density Boards)
- Backplanes
- Flexible and flex-rigid printed circuit boards
- Microwave boards, analog and analog-digital boards

Design stages

The design process of a printed circuit board in our design bureau is logically divided into stages with the obligatory verification of the results at the end of the key stages and with the provision of intermediate results to the customer.

1. Obtaining from the customer the full information on mechanics, holes, legend and components.
2. Entering a mechanical drawing, verification by the customer, adjustments.
3. Creation of the components (preparation of the library).
4. Upload and verification of the netlist.
5. Component placement, customer inspection, adjustments.
6. Description of the project, setting routing rules.
7. Checking the placement, Net-list and rules.
8. Fan-out (preparation for routing).
9. Wiring of differential pairs, clocks, fast lanes, control signals, customer check.
10. Wiring the rest of the circuit and power planes.
11. Trace verification, adjustments according to customer requirements.
12. Filling empty spaces (copper balance).
13. Alignment of the labels of the silk-screen legend and assembly drawing.
14. Panelizing for group assembly.
15. Filling the title block in the drawing.
16. Successful completion of the project, signing of acceptance certificates.



An example of a microwave circuit board, containing receiving and transmitting paths for digital TV and microcontroller. A fragment of microwave trace

Terms and cost

The project may take from one to several working weeks, depending on its complexity, the quality of the input data provided and the speed of the customer's answers to clarifying questions on the Technical Specification.

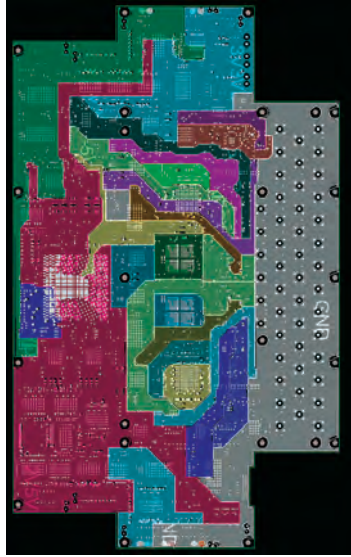
Estimated timelines for the design of printed circuit boards (excluding approvals)

Number of pins	Design time
Up to 3000	1,5–2 weeks
From 3000 to 6000	2–4 weeks
From 6000 to 10 000	4–6 weeks

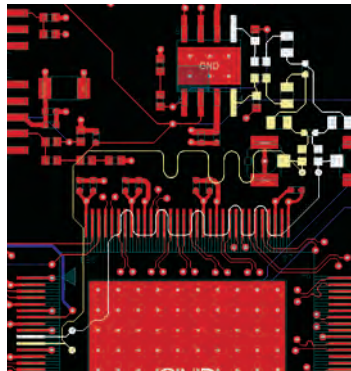
The cost of work is estimated individually based on the estimated labor capacity. Placing an order for the design of a printed circuit board in the design bureau turns out to be significantly more profitable than doing the design of the topology on your own or involving the homeworkers.

This is mainly due to the quality and accuracy of the work performed by our company, as a result leading to working samples earlier.

Why is it profitable for your company to order PCB design services in our design bureau?



An example of an on-board computer board with precision ADCs and DACs developed by our design bureau in 2014. An example of one of the power layers. Cutting polygons for different PWR nets



Routing a high-speed analog differential signal at the ADC input

Each company has its own development peculiarities. But sooner or later, they may need to route a printed circuit board, which cannot be done on time and with acceptable quality on their own: either all employees are busy, or there is no specialist with the necessary qualifications, or there is no CAD for the required level of printed circuit boards, or deadlines are burning ... It is necessary to transfer work to a subcontractor. Of course, this raises concerns: will the involved freelancer or design center do this work with the proper quality and at the right time and will there be any problems and inconveniences?

Let's describe the possible fears of customers regarding the design of the topology of printed circuit boards (PCBs) for outsourcing and find out which solutions are offered to customers by the design bureau.

Typical customer's objections and concerns

"Designing boards is one of our core competencies, and we must be able to do this internally"

The fact that every company should be able to do all the work "inside" is a serious misconception. This is especially true for the design of the layout of modern boards. Designing circuit boards, due to complexity and increased requirements for electronic products, is now "belongs" only to professionals in the field of printed circuit board layout. In order to do this within each R&D company, significant material and human resources are needed, which, due to the specifics of project work in the field of electronics, cannot be achieved 100% throughout the year. So, keeping "your own" PCB design engineers becomes unprofitable.

By the way, in many western and Asian countries, it is common practice to outsource the PCB design to professional design centers.

"I have a dozen of underloaded employees"

Experience shows that the qualifications of such "own" employees won't let you make a modern project quickly and at a high level, using all the capabilities of advanced eCAD systems. Not all companies have modern eCAD systems for printed circuit boards, and if they do, then the efficiency of such work is very low. So after all, it's easier to either retrain employees, or generally close the PCB layout development sector by outsourcing this work.

"Outsourcing design is unprofitable for our company"

Estimate the time and costs of the project done by you (license price, engineer's salary, taxes, overhead, vacation pay, sick leave and holidays, training, maintaining the qualifications of specialists, etc.), and you will see that the performance of work by a third-party professionals is at least 30% more profitable. We repeatedly asked our new customers to make this kind of calculation, and they were convinced. We suggest that you do this as well - let's compare the time and material costs of the project with the efforts of our design bureau and your employees, and you will see how profitable and convenient it is for your company to contact us.

"Design should be carried out within the company, as the developer likes to sit next to the designer and show where to put tracks and how to place components"

A modern systems engineer should clearly know the concept of the device, its functionality and implementation requirements in advance, not when half the board is already routed. Qualified PCB designer must complete this task, and the system technician only checks the design stages with the help of modern communication tools and free software for viewing the topology on the PC, freeing up valuable time for other tasks such as checks and calculations, working in the laboratory, study of datasheets and new technologies from component manufacturers, etc.

We have a technology that allows our customers to monitor the work of our engineer online without leaving their office or home.

"We will not be able to formalize the specifications, so that it would be possible to design a board without asking questions"

In this case you may contact us - we will help to correctly and clearly formulate the technical specification document for the customer, and the customer will only need to provide the necessary information on a particular stage of design. We may provide you with several templates of such a specification, with requirements already set for certain interfaces, power buses and routing critical signals.

"My circuit designers prefer to route the board themselves, as they are fully responsible for their work"

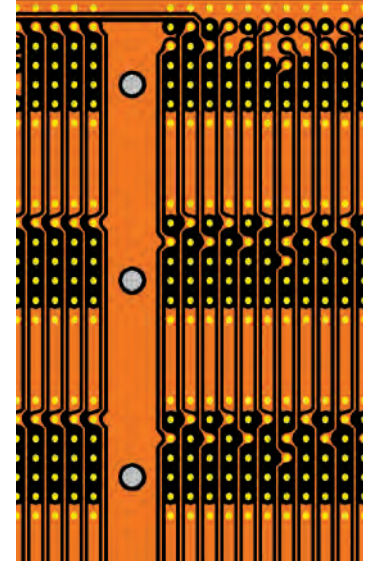
And it turns out that on a high level of CAD software they don't know (since they are routing for no more than 20–25% of their worktime) the DFM / DFA requirements (design for production), which means that it takes more time for the topology. Then the technologist rejects the board and requests for corrections be made due to non-compliance with the design rules for the production and assembly, i.e. the total design time increases and the quality suffers.

"We have very complex boards, a lot of serious interfaces, you can't design such a board"

To make sure that we are able of the most serious tasks, it is enough to talk with the head or employees of our design bureau and study the examples of work performed by employees. And the proximity of our own contract manufacturing allows us to deeply understand the tasks and make the boards of any complexity.

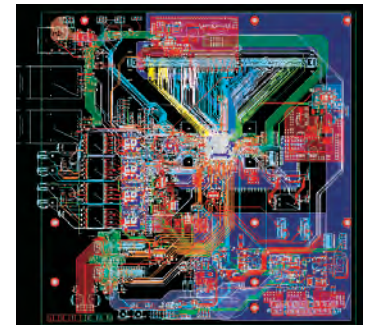
"We use another CAD system (Altium, Mentor, P-CAD), and we are asked to archive the board files only in certain format"

Generally, it doesn't matter what CAD to use for production and assembly of boards. We are accepting any formats of the electric circuits, even on paper. We are a company with a history (on the market since 1997), we are not going to disappear anywhere, and we can make small changes and additions to the project quickly and free of charge. And if you really need it, it is possible to convert projects into CAD systems that you require. After all, we can do projects both in Altium and in Mentor, it just takes a little longer, because these CAD systems are less convenient for us. We hope that you will not force us to do complex projects in P-CAD, which is hopelessly outdated. However, we also have a solution for P-CAD that allows to transfer the project from P-CAD to Cadence Allegro for routing and save the finished routed file back in P-CAD.



Compact-PCI cross-Board with customer's extensions for 16 6U slots.

Example of one of the internal layers



Example of a printed circuit Board with DDR3 interface of the memory bars, with tuning of data and address

PCB mountable EMI shield cans



Forming – means stamping according to a specified program from a pre-cut "flat pattern" on the sheet material, performed on a special machine

Pulling – it is the production of an entire screen made of sheet material with the use of the special stamp



Developer's kit

We supply high-quality high-frequency and microwave shield cans for mounting on printed circuit boards. The purpose is shielding of electromagnetic fields, protection from interference in high-frequency circuits.

Configuration design. The shield can consists usually of a frame mounted on a board with the help of solder or solder paste, as well as a spring cover. "Pins" can be made under the frame to fix it on the printed circuit board. On top of the frame, a bar can be attached that allows shields to be mounted from pallets to printed circuit boards automatically, by vacuum nozzle. Both the frame and the cover can be provided with holes of different shapes.

Method of manufacture. There are two methods of making shield cans – by forming from a pre-cut "flat pattern" or by "stretching from a flat sheet".

We supply standard size shield cans (see the table on our website), that cost less, or you can order a custom design shield by sending us drawings or a three-dimensional model in the format of AutoCAD, Solid, etc. In addition, we can help with the design of the exact 3D model based on your sketch according to the exact location on your printed circuit board.

Materials

Thickness – from 0.1 to 0.4 mm, standard – 0.3 mm.

1. Based on copper:

- 1.1. Brass (pre-coated with electroplating / post-galvanic coated / uncoated).
- 1.2. Alloy C770 – 18% Nickel and 27% zinc, and the rest is copper.

2. Based on steel:

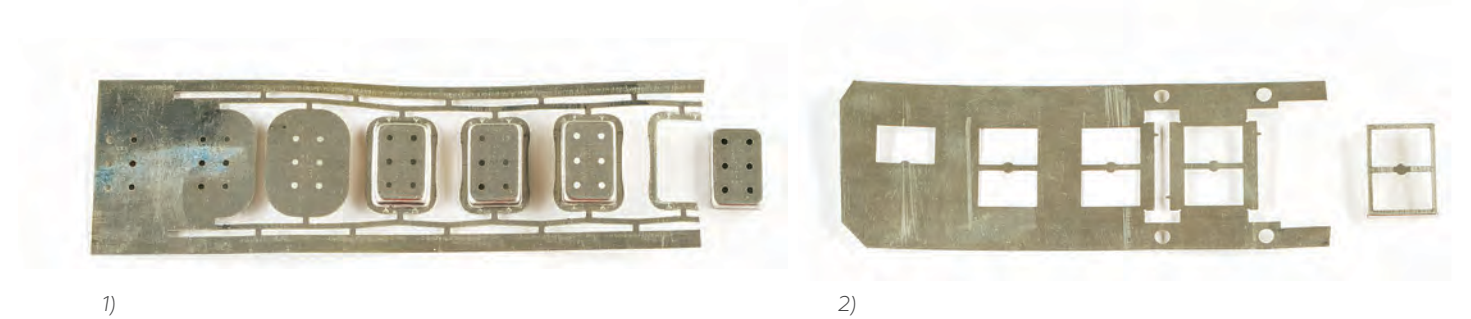
- 2.1. Carbon steel (pre-electroplated / post-electroplated tinned).
- 2.2. Galvanized steel.
- 2.3. Stainless steel.

Surface finish

We use a 100% pure tin coating on the shield cans, which allows us to use standard assembly process with tin and lead-tin solders. The soldering temperature and time both correspond to the soldering mode for standard electronic components.

Note. The material is covered with tin before forming and trimming.

Packaging. In pallets or tapes for automated assembly.



1)

2)

Configuration design options

Spring cover

This very popular and traditional technique ensures the presence of many spring-loaded contacts that fix the cover on the frame, and convenient removal and putting on of the shield cover if necessary. The methods of filling the spring-loaded contacts can be different.

Cover with recesses

The recess cover provides both a latch device and a point for electrical contact. This mounting method can be used for both high-profile and extremely low-profile shield cans.

Frame without rim

As a rule, a rim is made along the upper edge of the frame, which provides a significant strengthening of the shield can, which simplifies operations and its mounting. In cases where the components are too close to the shield can walls and do not allow you to make a rim, the wall may be left flat, without a reinforcing rim. Such shield cans can be either without rims at all, or with rims only in some places.

Multi-section shield cans

When shielding is required between a set of closely spaced multi-functional circuits, the most optimal solution is to create a single multi-section protection under a single cover. These shield cans take many different shapes and designs depending on the geometry of the circuits on the printed circuit board.

Full-length shield cans

At higher frequencies, the engineers give the preference more airtight structures of electromagnetic protection. Full-length shield cans provide a seamless protective enclosure. These shield cans could even be manufactured as one-piece frames with pulled grips on the cover. Unlike shield cans that are bent from the reamer, they must be pressed using special molding tools.

Cover shield cans

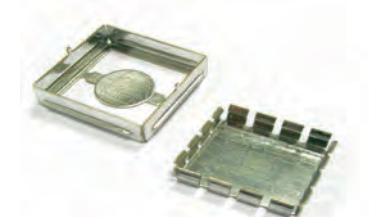
This is the most cost-effective form of shielding. Cover shield cans are used on proven and highly reliable circuits, since access to the circuit elements is not possible after the can is soldered.

Developer's kit

For developers, we provide an inexpensive kit of 25 standard shield cans of different sizes, which allows to quickly select and check the desired shield during development and then, if necessary, order any batch of such shields (see the photo "Developer's kit").

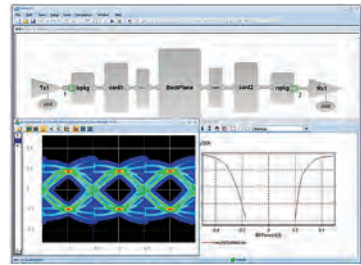
1) Stretching (stamping) from the tape, a sequence of steps

2) Stamping from a pre-cut "flat pattern", a sequence of steps

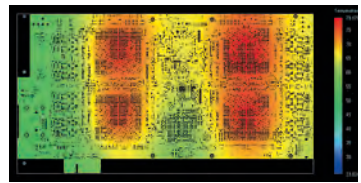


Shield cans with spring cover

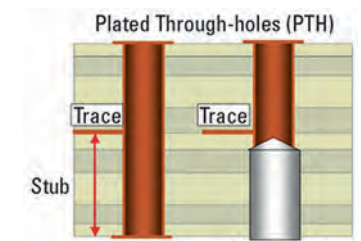
Delivery of Cadence Allegro eCAD



The library of standard configurations with available standard models allows you to model almost any high-speed data link



When using Cadence Sigrity, it takes about 30 minutes to complete temperature simulation of a 16-layer PCIe Board with 4 FPGAs consuming 100 watts each



The support of modern technologies, such as Back-Drilling, gives the developer a unique tool for effective and high-quality execution of any projects

Being a partner of Cadence Design Systems, we offer companies in Russia that use low-level PCB CAD systems (such as P-CAD or its software successor/follower) to supply and implement the most powerful software for PCB design – Cadence Allegro.

If your R&D department or your subsidiary in Russia uses outdated CAD systems, then the efficiency of developing complex modern printed circuit boards is likely to be low. The efficiency, speed, and correctness of circuit board development can be significantly improved with the CAD for higher-level printed circuit boards – Cadence Allegro.

Cadence Allegro is recognized as one of the most advanced and convenient packages in the world, aimed at designing complex multi-layer printed circuit boards.

If you purchase the license, we'll perform error-free conversion of the customer's libraries and projects from P-CAD, Altium and Mentor to Cadence Allegro, provide training for design engineers, build a front-to-end design flow, and provide technical support.

By delivering and implementing Cadence Allegro at leading Russian enterprises, PCB technology group ensures rapid development of CAD by engineers, transfer of component libraries, diagrams, and old PCB designs, as well as integration of Cadence Allegro with mechanical 2D and 3D CAD systems, enterprise resource and project management systems, and PLM/PDM systems.

An inexpensive, but powerful enough basic license can be extended with powerful options that cover the most high-tech needs of the PCB design departments and contain an electrical circuit modeling system, a microwave design module, and much more.

Basic license and Cadence Allegro options

The package is based on the basic CAD set of PCB Designer, which consists of three components - circuit editor, Allegro PCB editor, and the SPECCTRA autorouter. Despite the relatively low cost, the basic set gives the design engineer very powerful tools for designing the circuit and printed circuit board.

And, if some departments of the enterprise require other functions, they can be easily purchased as separate "floating" options and uploaded to the enterprise's server for use by different departments, if necessary.

The main advantage of this option is almost complete coverage of all the functions required for designing complex and advanced printed circuit boards.

Customer support

In order to provide support to official Cadence Allegro users in Russia, as well as to familiarize potential users with the capabilities of this advanced CAD system, PCB technology group provides trainings for PCB design engineers on the design methodology for circuit boards.

Our service

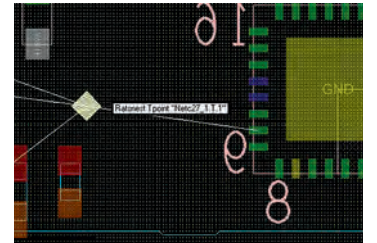
1. Selection of the optimal configuration of workplaces.
2. Supply of computer equipment and network equipment for the organization of work places of design engineers.
3. Software installation and license maintenance.
4. Trainings on the methodology of designing multilayer boards for design engineers.
5. CAD development assistance and online support for engineers.
6. Translation of component libraries from P-CAD or Altium to Cadence Allegro.

Advantages of Cadence Allegro

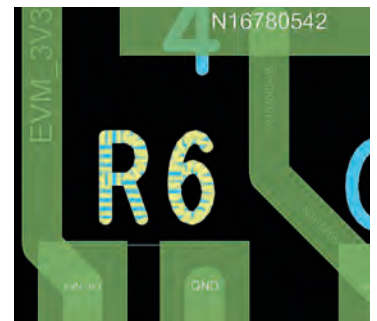
When compared with simpler packages (the most common ones in Russia are P-CAD, Protel, Altium, Orcad Layout), the following main advantages of the Cadence Allegro package can be noted:

- 1) Deep two-way communication of the circuit with the PCB layout;
- 2) Convenient and fast routing of high-speed circuits;
- 3) Routing of differential pairs;
- 4) Alignment of the chain lengths taking into account delays inside the chip and vias;
- 5) Online monitoring of the connection length in signal groups with length control;
- 6) Full set of rules and restrictions with online control;
- 7) Ability to set the sequence of connections in the chain;
- 8) Signal integrity simulation system;
- 9) Free viewer for the customer;
- 10) Personal settings of the user interface;
- 11) Macros for quick execution of groups of operations;
- 12) Customizable user routines.

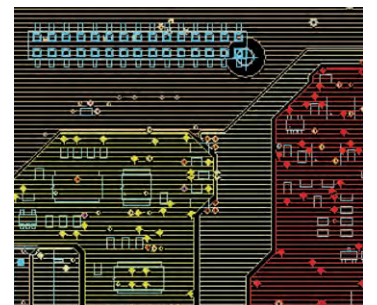
In addition, many Russian PCB designers are familiar with the interface of a very convenient and powerful circuit editor OrCAD, which is built into Cadence Allegro as a whole. Also, many people, including P-CAD users, are familiar with the most convenient autorouter SPECCTRA, which is a part of the Allegro package.



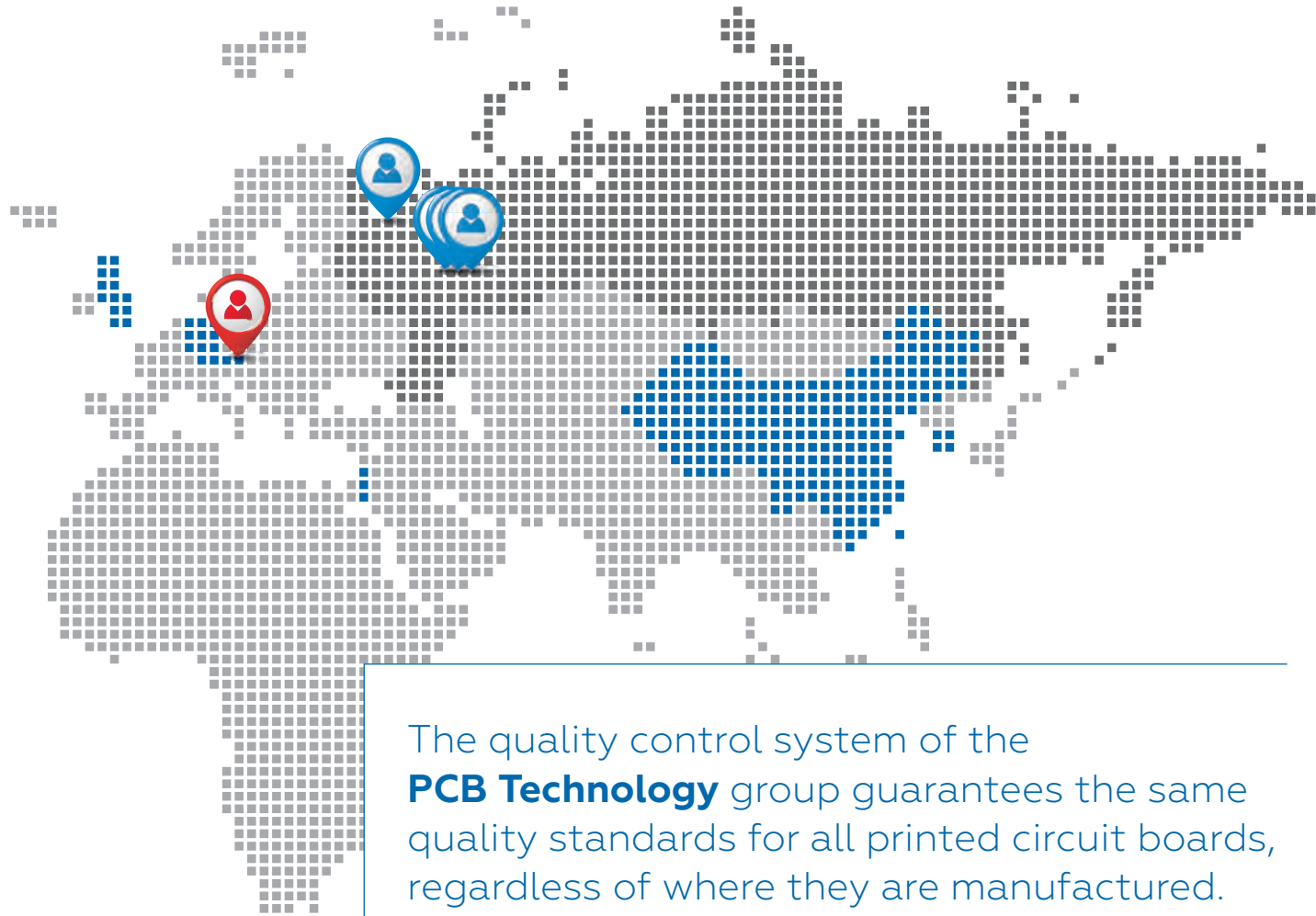
Ability to set the order of pin tracing in the chain and align the "shoulders" of the chain allows to trace high-speed interfaces



Indication of the chain name on routes, outputs, and polygons increases the speed of operation



Unique convenience of working with polygons



The quality control system of the **PCB Technology** group guarantees the same quality standards for all printed circuit boards, regardless of where they are manufactured.



Manufacturing plants location:

- China
- Great Britain
- Germany
- Austria
- Israel
- South Korea



Partner companies:

- ELLWEST (Austria)



PCB technology group:

- "PCB technologies" LLC
- "PCB technologies" LLC, Saint Petersburg
- "KB Schematics" LLC
- "PCB SOFT" LLC



If you agree that the quality is in the first place, we will be glad to see you among our customers.



Moscow

Tel. / Fax multi-channel: +7-499-558-02-54, +7-495-545-17-08,
pcb@pcbtech.ru www.pcbtech.ru

Address: 26 V Bolshaya Pochtovaya str.,
Moscow, 105082,
building 2, office 406

Saint-Petersburg

Tel. / Fax: 8-812-779-12-23
Phone mob.: 8-911-927-01-81
E-mail: spb@pcbtech.ru

Address: 197022, Saint Petersburg,
Vsevolod Vishnevsky str., 12, lit. A, office 302