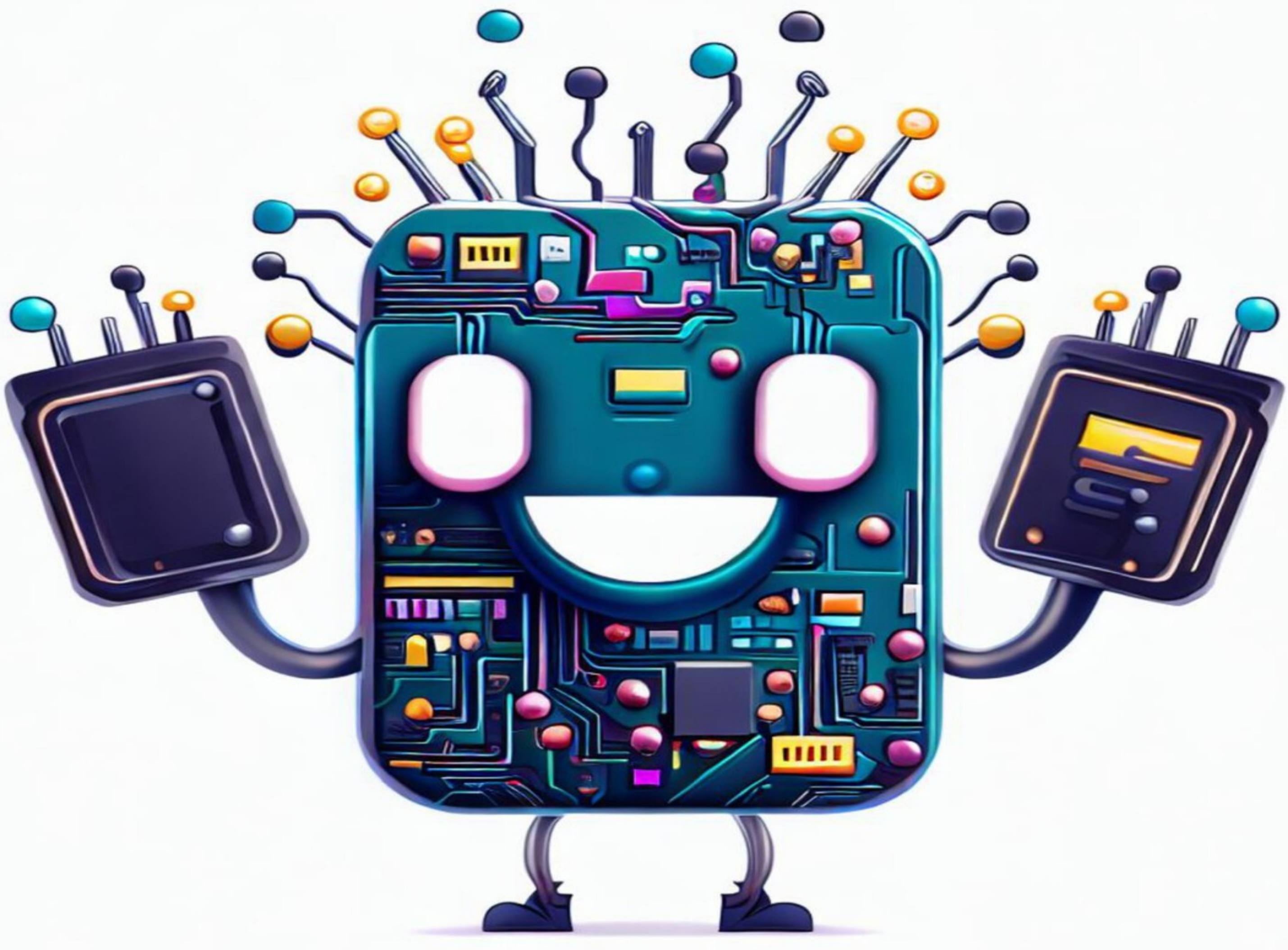
PCB-Design



101 Tips and Tricks

Jörg Rippel

PCB-DESIGN

101 TIPS AND TRICKS

Jörg Rippel

Disclaimer

All information in this book has been compiled by the author with the utmost care. However, errors cannot be excluded. The author is therefore obliged to point out that he accepts no guarantee, legal responsibility or liability whatsoever for any consequences arising from incorrect information. The author is grateful for any information regarding errors. Internet addresses and version numbers reflect the status of the information at the time of writing. Please note that no responsibility or liability can be accepted for the sources used. References to unavailable sources will be gratefully accepted and corrected in subsequent editions.

Most of the product names, company names and logos mentioned in this work are usually also registered trademarks and should be regarded as such. The use of product designations is based on the manufacturer's spelling.

2nd edition
2021 2023

Imprint:

Publisher: Joerg Rippel
P.O. Box 1310
64726 Bad Koenig
Germany
http://www.rippel.info/

(v20230924)

Table of Content

Foreword

In this book I would like to show you how to design professionally made printed circuit boards.

As a maker or hobbyist, you have probably already built your own circuits on breadboards. And you now have the desire to have a PCB professionally made. But how is such a PCB design done? What steps are necessary and what needs to be considered? What preliminary considerations are necessary and what tips and tricks help me with the design?

With 101 simple steps, this book guides you through the development of your own PCB. All basic questions are answered and various best practice methods ensure that your PCB design will be as professional as possible.

1. What is this Book about?

Successfully designing your own PCB requires a combination of electronics fundamentals, as well as a dose of specialist PCB knowledge. With this book, you will gain basic knowledge about designing and creating PCBs.

The aim is to enable you to design your own PCBs and have them manufactured as quickly as possible, with as little effort as necessary. For this reason, this book is structured in such a way that all information can be easily looked up.

It covers all the basics and the theoretical background. By using the right technical terms and basic knowledge about PCBs, you will be able to expand your knowledge and develop your skills either by talking to yourself or by studying other books.

It also shows you concrete examples and tips for PCB design. This is not a tutorial on how to use a PCB design program, but shows you what is important in designing traces, how to create the floor plan, what basic principles must be observed with regard to waste heat, how to reduce susceptibility to interference, how to design for high and low differentials, and that it can be counterproductive to plaster all unused parts of a PCB with ground planes.

And it shows you the most important standards, how to provide the design files for a PCB production and what things to consider when choosing a PCB manufacturer.

Not everything can be covered in this book. It is intended as an introduction to PCB design and provides all the necessary basics for designing "normal" PCBs. A chapter on high-speed design offers a general introduction to what most important. In this way, you can get a taste of high-speed design with the help of generally applicable rules.

Many technical terms and synonyms are used in this book. Sometimes the term printed circuit board, PCB or circuit board is used. This often results from the context, but can contribute to confusion at the beginning. There is therefore an appendix to explain the most important terms. It will be necessary to refer to it occasionally. This is solely for the reason that I also want to familiarise you with the jargon right away, which you will need for further self-study.

2. History and Origins of a Circuit Board

Before the introduction of the printed circuit board, components were freely wired. Components such as switches, variable capacitors, resistors and tubes, with their soldering eyes and wire connections, were also the mechanical attachment point for each other. You can still see this in old radios from the 1920s. Sometimes a soldering strip was also used as the central attachment point.

Although invented earlier, the use of printed circuit boards only began in the 1950s with single-layer copper-coated circuit boards. A few years later, two-layer coated PCBs were already being used. In the 1960s, these were plated through for the first time, i.e. the two layers were connected to each other by a wire bridge. The so-called multilayer boards appeared at the end of the 1960s. In the 1980s, SMD technology was added and since then the technology has been developing faster and faster.

The variety of components has increased greatly in recent decades. Just think of LED technology, for example. Until the 1990s, LEDs were only available in the colours red, green and yellow. Today, they are available in a wide variety of sizes, power classes, colours and designs.

This development has taken place in all areas of electronics. As a result, printed circuit boards are also evolving. The ever-increasing miniaturisation of the structures on the surface places ever greater demands on the manufacturing technology. Software for designing these boards must always reflect the latest possibilities. And the machines used in production must also become better and better.

The result of all this is that the production costs for a "normal" PCB continue to fall. Production lines that no longer meet the most modern production standards can thus continue to be used for low-cost mass production of simpler PCBs. This is often seen in the construction of devices: The motherboard is created in the most modern production, often a multilayer board with very fine tracks and small SMD components. The power supply board, on the other hand, is often a very simple, single-sided board with wired components. The weighting of the manufacturing costs could thus be placed by the manufacturer more on the main functionality and less on the power supply.

The hobby and amateur sector also benefits from this, because even multilayer boards with two or four layers have become affordable for private individuals. For the first time, hobbyists now have access to professionally manufactured circuit boards. After all, not every device needs PCBs, as they represent the state of the art in the latest mobile phones and computers. And so the previous generation of PCB technology still has its place and can continue to run.

3. The substrate material of a printed circuit board

Put simply, the printed circuit board, often referred to as the PCB, is a substrate of insulating material onto which metal is deposited and then structured. Structured means that the conductive pattern is carved out of a full-surface layer of metal. This is done by etching away parts of the metal layer.

A rigid PCB consists of an insulating material that acts as a support for the copper. This insulating material consists of a reinforcement and a binder (resin). The material for the reinforcement is paper or glass fibre, and the binder is an epoxy resin. The carrier material thus combines two functions: the strength of the PCB and the electrical insulation of the copper layers. The electrically conductive layer of a PCB is copper. This is applied as a foil to the insulating substrate. If only one layer of copper is applied to the substrate, it is called a single-sided board.

If a copper layer is applied to both sides of the carrier material ("core"), this is called a double-sided printed circuit board ("double-layer").

A double-sided PCB is not called a "multilayer PCB", even if it has two layers of copper. Only a PCB with more than two layers of copper is called a multilayer PCB.

There is a wide range of materials used to insulate printed circuit boards. The best known is the material known as "FR4", which consists of glass fibre and epoxy resin. There are many different material compositions under the name

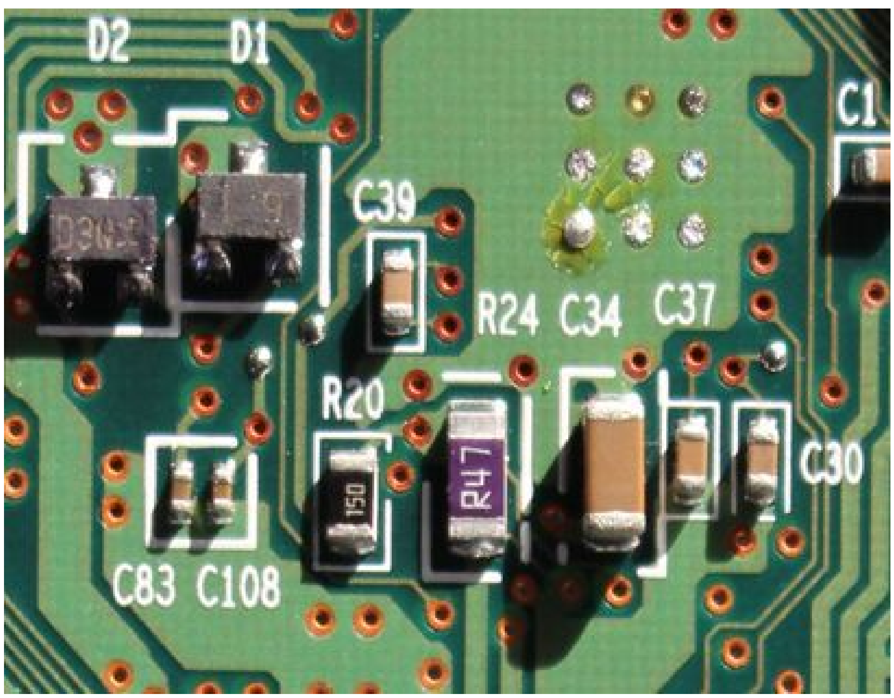
"FR4" due to the many different types of resin and fabric. However, it is only in exceptional cases that this can have an effect on the layout of the circuit and must be taken into account at the design stage. Normally, a PCB will not have a noticeable effect on the behaviour of the traces and does not need to be taken into account. (This is not the case for high-speed design!)

4. The Surface of a Circuit Board

The so-called soldermask is applied to the surfaces of a circuit board. Applied as a foil or varnish, it is intended to ensure the insulation of the components from the conductor paths. Since the solder pads are left free by the soldermask, they can accept solder and the components can be soldered after assembly. A pad can be a plated hole for a wired component or a surface pad for an SMD component.

The soldermask also increases the dielectric strength between the tracks, so that no flashover can occur at high voltages.

Nowadays, soldermask is available in different colours. In the past it was always green, but today it is available in other colours such as white, black, blue and red.



The silkscreen is applied on top of the soldermask. This marks the position of the components with their identifier and outline. Other texts, logos and the marking of prohibited zones are also applied with the help of the silkscreen.

To protect the remaining exposed copper on the PCB surface, an additional surface protection layer is applied to the board. This is necessary because the copper oxidises in the air and further processing of the PCB becomes impossible. No component can be soldered to oxidised solder joints. This can extend the shelf life of PCBs from a few weeks to over 6 to 12 months before further processing is required. If you need to store PCBs for longer than 12 to 18 months, contact your PCB manufacturer to see if this is possible for the PCBs they produce. The range of surface protection coatings, and their prices, is highly dependent on the PCB manufacturer.

The following surface protective coatings are commonly used:

Surface Coatings	
HAL / HASL	*„Hot Air (Surface) Leveling"* Copper contacts are coated with tin and smoothed with a stream of air.
Chemical tin	Tin Whiskers. Hardly storable. Critical processing.
ENIG ENEPIG	*"Electroless Nickel-Immersion Gold"* *"Electroless Nickel Electroless Palladium Immersion Gold"* Level contact surfaces. Popular.
Chemical Silver	Level contact surfaces. Common in the USA.
OSP	Organic Surface Protection. A dip that seals the copper. Leveler than HAL and cheaper than ENIG
Leaded HAL	Not ROHS compliant

A PCB with HAL surface is mostly used for wired components. For SMD PCBs, a more even surface is desired than is possible with the HAL surface.

HAL is also no longer suitable for PCBs with very fine structures ("fine pitch"); ENIG is preferred for this. The tin contracts and forms small hills during processing of HAL PCBs. When positioning components with extremely small pitch, the component slips down and the pins come to rest between the contacts.

Here's a tip: In the case of plug-in contacts on the board, such as computer plug-in cards ("gold fingers"), make sure that these contacts are made of hard gold. The soft gold normally used wears off too quickly during the plugging process.

5. Structure of a Multilayer PCB

Multilayer PCBs are used for complex circuits where the space on a single or double-sided PCB is insufficient. These are assembled like a sandwich from several double-sided PCBs with an insulating layer in between. This extra layer of insulation is called prepreg. This term is short for "pre-impregnated fibres". These fibres, pre-impregnated with reactive resins, are cured under pressure and heat to form this layer.

For a multilayer panel, all these layers are aligned, glued together and baked under heat in the pressed state. The melted resin penetrates between the layers and after cooling the whole stack is a solid bond..

The arrangement and sequence of the core material and prepreg can vary. The structure shown in Figure 4 is also known as "book construction". For multilayer boards, however, it is both this arrangement and the properties and thickness of the material that determine the impedance of the traces. In high-speed design, where the PCB itself has a significant effect on signal integrity, this must be taken into account. In this case, PCB manufacturers often use a different construction where "core" and "prepreg" are used in a different order.

With the current state of the art, multilayer boards with 60 or more copper layers are possible. PCBs with up to six copper layers can be found among the low-cost suppliers and can usually also be manufactured by any smaller PCB manufacturer. For the production of PCBs with around 20 layers it is also quite easy to find manufacturers. Anything beyond that is often custom-made for the military or aerospace sector.

Multilayer boards are usually produced symmetrically, i.e. with an even number of layers. The even number and symmetrical arrangement of layers in a multilayer board is also intended to prevent deformation in the course of further manufacturing and processing.

Deviations from this can be produced by some PCB manufacturers on customer request, but do not correspond to the normal production process. As a rule, such a special design is also only produced for large quantities and a corresponding customer. The effort involved is too high for small quantities. It is cheaper to use one more layer in the board than to insist on an odd number of layers. If necessary, it is better to use one more ground layer to benefit your layout.

Here's a tip: Two other special types of multilayer PCBs are the so-called AML PCB and multilayer PCBs with integrated heat dissipation.

The AML (Active Multi Layer) PCB is a multilayer PCB where the inner layers are filled with active or passive components.

Another variation is the 'metal core' PCB, which has a solid layer of aluminium in the middle of the multilayer structure to dissipate heat. (Not to be confused with a copper layer in a normal multilayer board).

These two special shapes can offer you a way out when all the classic approaches to solving your project have failed. But they are expensive.

6. Groud Planes

Please note that whenever someone refers to a "ground layer" or "ground plane", they are referring to a solid copper layer. This layer is sometimes referred to as a "reference layer" or "reference plane".

For PCBs that have two different supply layers or "power planes" (e.g. 3.3V and 5V), the common ground, or ground plane, is usually referred to as the reference layer. If only one supply voltage is used, the term "ground layer" or "ground plane" is more commonly used. Exactly which term to use depends on the circumstances of the design.

7. PCB Panelization

The PCB manufacturer uses a standardised format for PCBs. This so-called panel passes through the processing steps of its production line.

In the past, the customer commissioned the PCB manufacturer to produce a PCB, which was then manufactured in the format of the panel, with its edge length of about 60 cm/24 in. If the board was smaller, there was a large empty space around it, i.e. unused space. If the circuit board could fit more than once on a panel, it could be duplicated to fill the space - it was panelized.

For prototypes and small series, however, the costs of this production method were high. A few years ago, therefore, the so-called "pooling" was invented. The circuit board designs of different customers are collected and puzzled together in such a way that a panel is utilised as optimally as possible. In the past, there was quite a long waiting time until a panel was filled, but today there is no longer any noticeable delay with the PCB manufacturers who specialise in this.

Nowadays, PCBs for hobbyists and small series for prototypes are almost without exception produced in pooling. As a result, PCBs have become affordable in small quantities, or even as individual pieces.

What this means for you as a PCB designer is that you no longer need to arrange and multiply your designed PCB in panel format. So, unless you need so many PCBs that you are going to use the production line alone to manufacture your PCB for a large order, you no longer need to create a panel.

Here's a tip: On a tour of a PCB manufacturer, I had a conversation with the production manager about this very point. He told me quite clearly that no customer could know exactly how to define the panel for their production line and machines. In the CAM post-processing, they would define the panels themselves and at the same time fix errors in the EDA design files. As a customer, you should just send the design files of your PCB and specify how often you want it produced. In essence, the customer should concentrate on what they do best: designing their PCB. The PCB manufacturers would do what they do best: Produce the PCB. That is what they are paid for.

8. PCB Outline

When defining the outline of the circuit board, the dimensions of the PCB should be such that it can be mounted in the enclosure without mechanical stress. It should not tilt or bend during assembly. If you have used towering components such as heat sinks and capacitors, you should ensure that the PCB can be tilted during assembly if necessary.

If a metal enclosure is used, its coefficient of expansion should be taken into account to avoid stress on the PCB. The circuit board cannot follow the expansion and contraction of a metal enclosure and the mounting points should not block movement in this direction. Otherwise there is a risk of cracking or breakage of the PCB.

There should also be sufficient access to the PCB or other replaceable component, such as a fuse, for repair and maintenance.

In enclosures where the PCB is supported laterally by a rail, there must be sufficient clearance in the front panel for controls and connectors. However, the board must not slide back and forth in the holder when the controls are used. This could cause mechanical stress, e.g. on connecting leads, which should be avoided.

If the board is mounted on a rail, the top and bottom edges of the board should be free of conductive tracks to avoid short circuits. Even if the tracks are covered with solder resist, wear and tear (friction, vibration, assembly, transport) can expose the copper layer and cause unwanted short circuits.

9. Mounting Holes

Fixing points in the corners of the board will cause deformation if the board is sagged in the middle of a large PCB, especially if heavy components are placed in the middle of the board. It is possible that over time this mechanical stress could lead to a defect or even a point of contact with a component underneath.

Consider the main board of your computer. This board is about the size of an sheet of paper. If it is mounted vertically, a heavy processor fan can deform the mainboard after just a few years. For this reason, there is often an additional mounting hole, or a special mounting bracket, in the middle of the PCB to give the board and the fan a better hold. Boards can also vibrate and break during transport. Mounting holes should therefore be evenly distributed.

Screw heads tend to be wider than the hole. Make sure there is a protective zone around the mounting hole. Marking this on the PCB will make it clear when designing the board and prevent problems. Are insulating washers or spring washers used in the mounting? Is there a ground connection to be made? Does the cable lug, which may be used for a ground connection, require a protective zone? Think about this early in your design to avoid collisions with components.

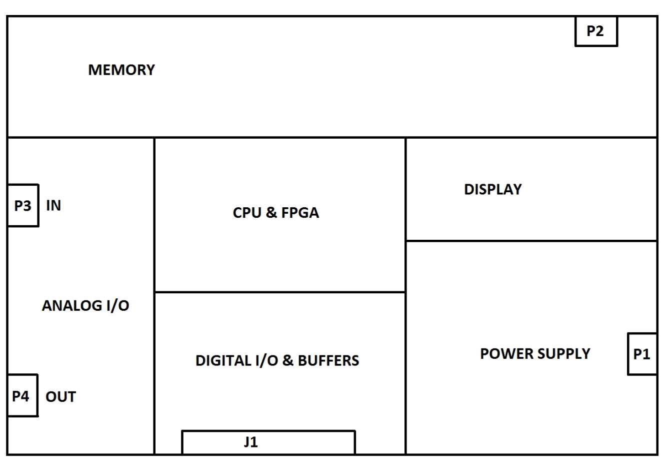
Here's a tip: Plated through-holes, also known as vias, under screws are fragile. The sleeve of the plated hole could break under pressure. Do not use vias close to mounting holes and keep them well away from screws.

10. Floorplan

A floor plan is created before the components are positioned on the board. This determines whether, for example, connectors need to be kept in a certain position if the enclosure has an access opening at that point. The position of the display on the board, or its connection cables, depends on the cut-out in the case.

Other considerations are also necessary, taking into account the characteristics and effects of the circuit itself. Heat generated by circuit components may need to be dissipated by airflow or through ventilation holes. Natural convection may need to be used in some locations, to avoid a noisy fan. Which way should warm air go? Attention must also be paid to the general position of the enclosure. Is the unit exposed to direct sunlight? Is it used in an enclosed space? Hot air rises (except in a vacuum), so cooling must work in all intended mounting positions, i.e. vertical or horizontal.

There are circuits that are purely analog or digital. There are also mixed circuits. In this case it is important to consider the digital and analog parts of the circuit in the layout. In general, parts of a circuit that cause - or are susceptible to causing interference - should be placed on the PCB with appropriate care to minimize interference to other parts of the circuit.

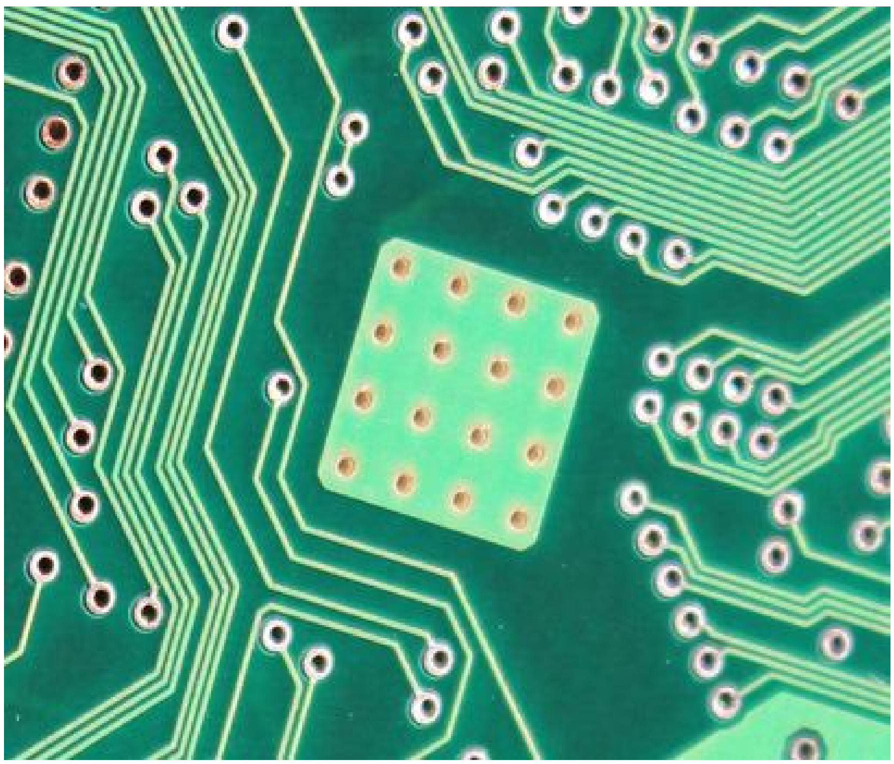


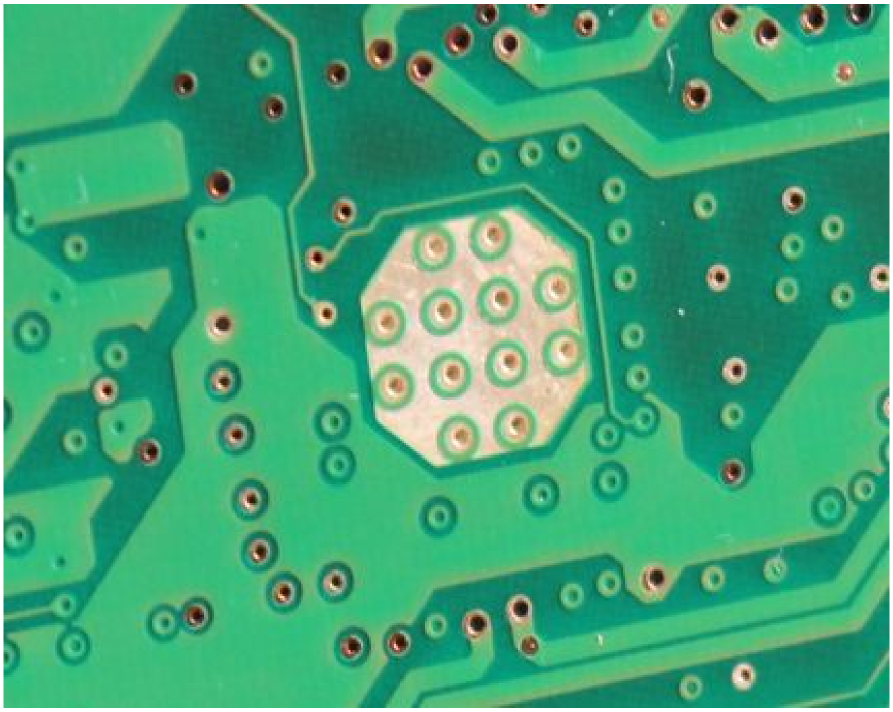
Do not underestimate the importance of the floorplan. The expert who designed the circuit may have thought very carefully about the positioning of components in the layout, and about parts of the circuit that need to be implemented in a special way to ensure that the circuit works properly. At best, you will find notes on this in the schematic.

In this example of a floorplan you can see that the analog part of the circuit is far away from the power supply. The components with the most pins, CPU and FPGA, are in the middle. This makes it easier to route the traces.

11. Heat Dissipation

There are a number of ways to remove heat from components. You will be familiar with the use of heat sinks and fans. These are easy to apply to wired components. Other wired components, such as diodes and resistors, cannot be heatsinked if they are of normal design. In this case, such components are soldered away from the board and positioned "on stilts" in the airflow of a fan.





This is not always possible with SMD components due to their design. It is possible to attach heatsinks to larger SMD ICs, but small components cannot support a heatsink as large as would be required to dissipate the heat. The component density on modern PCBs often does not allow enough space. But you can use the PCB itself to dissipate the heat.

The heat is then transferred to the PCB via "thermal pads". These thermal pads are vias that are placed under the component to carry the heat away from the component. These vias should be no larger than 10 mils in diameter to avoid clogging with solder. If there is sufficient space, the opposite side of the PCB can be provided with an enlarged copper surface area for heat dissipation, connected via the aforementioned thermal vias. The datasheet for such components will give you the exact design requirements for effective heat dissipation.

A special type of circuit board is the "metal core" PCB. This has a solid layer of metal in the middle of the multilayer structure to help dissipate heat. This can be a solution if your circuit has a problem with massive heat generation. However, your EDA software would need to support this and your PCB manufacturer would need to be able to produce such a board. This should be considered very early in the design process.

Here's a tip: Also worth mentioning is the ability to simulate heat distribution on a PCB, for example using HyperLynx software. In this way, hot spots on the PCB can be identified during the design phase and the layout can be modified accordingly.

12. Pre-Flight Check for your Design

What is the underlying design goal? Should the PCB:
- Be durable?
- Be easy to repair?
- Be cheap to produce?
- Be manufactured abroad?
- Have resilience?
- Be only a prototype or a small series?
- Be created under time pressure?
- Be made for publication and exhibition?
- Be made with how many layers?
- Be exposed to strong vibrations?
- Be subject to high thermal stress?
- How long should it be produced, how long must the components be available?
- ... ?

All of these criteria can have a direct impact on PCB design. Let's look at some other things that play a role:

For example, the colours of the LEDs or the display can be important if the users are colour blind. Or a deaf person may not be able to hear a warning buzzer. A design is about more than the arrangement of components on a board.

Before you start designing, make a list of the criteria that have a direct impact on the design and that are essential.

13. Component Selection

When selecting components, long-term availability in the correct package form must be chosen. Many wired components are being phased out as the industry moves to SMD technology wherever possible. As a result, production lines for wired components are becoming unprofitable and are being shut down.

This can mean that when components are discontinued, you are forced to rework the PCB. This may mean having to repeat all the acceptance tests. If you are a freelancer: Check the small print in your contract with your customer to see who will pay in this case.

Look for the words "not for new design" in datasheets of components. This is usually used by manufacturers to indicate components that will soon be discontinued.

If in doubt, ask the manufacturer!

Where possible, standardise the components you use. LEDs can be standardised in diameter, resistors can be standardised in E12 or E24, and so on.

Check the quality of components such as electrolytic capacitors and the temperature range in which they can be used.

Also check availability with several major distributors to get an idea of whether there may be any supply problems.

This topic is addressed again in the chapter BOM / Bill Of Material.

14. Availability / EoL

When considering the availability of components, it is not only the long term perspective that needs to be considered, but also the short term availability and quantity.

For example, if you want to produce a prototype or a small run and the power transistor is only available in minimum quantities of 5000, it may be easier and cheaper to change the design altogether.

For PCBs that have been in production for a long time, discontinuing components can have serious consequences. Lifecycle management should keep track of the availability of components used in your products.

This topic is addressed again in the chapter BOM / Bill Of Material.

15. The mechanical consideration of drillholes

Drilling holes in printed circuit boards may seem like a simple task, but there are many things you can do wrong. Let's take the example of a wired component. The leads have to go through holes in the PCB that are plated with copper. After drilling, this copper is deposited as a thin layer on the wall of the hole by a chemical process. This reduces the diameter of the hole. The thickness of the leads on a component is known, as is the thickness of the copper plating. So the PCB manufacturer tries to make the holes large enough to allow the solder to flow into the solder eye and firmly connect the lead of the component to the PCB.

So when drilling, a distinction is made between a plated-through hole and a non-plated-through hole.

In the table below you can see the rounding values for component holes as used for wired components.

Hole Size in mm			
Start Size	Stop Size	Rounded	Toolsize
0,51	0,55	0,55	0,70
0,56	0,60	0,60	0,75
0,61	0,65	0,65	0,80
0,66	0,70	0,70	0,85
0,71	0,75	0,75	0,90
0,76	0,80	0,80	0,95
0,81	0,85	0,85	1,00
0,86	0,90	0,90	1,05
0,91	0,95	0,95	1,10
0,96	1,00	1,00	1,15
1,01	1,05	1,05	1,20
1,06	1,10	1,10	1,25
1,11	1,15	1,15	1,30
1,16	1,20	1,20	1,35

1,21	1,25	1,25	1,40
1,26	1,30	1,30	1,45
1,31	1,35	1,35	1,50
1,36	1,40	1,40	1,55
1,41	1,45	1,45	1,60
1,46	1,50	1,50	1,65
1,51	1,55	1,55	1,70
1,56	1,60	1,60	1,75
1,61	1,65	1,65	1,80
1,66	1,70	1,70	1,85
1,71	1,75	1,75	1,90
1,76	1,80	1,80	1,95
1,81	1,85	1,85	2,00
1,86	1,90	1,90	2,05
1,91	1,95	1,95	2,10
1,96	2,00	2,00	2,15

The residual ring for component holes must be rounded to a minimum of 0.2mm all round.

As already mentioned: This rounding is done because the holes must be drilled larger than the nominal diameter to compensate for the application of material in subsequent processes.

16. Vias

Vias connect tracks or copper planes of the different layers with each other. They are electrical plated holes, so-called plated through holes "PTH".

The following dimensions are specified for the necessary holes:

Via in mm			
Start Size	Stop Size	Rounded	Toolsize
0,20	0,25	0,25	0,25
0,26	0,35	0,30	0,30
0,36	0,50	0,50	0,40

Vias must have at least 0.15mm of residual ring around the hole. The restring is the copper collar around the via that connects the metallised inner wall of the via to the copper layer of the board.

As you can see there is a gap between the via holes of 0.4mm maximum and the component holes of 0.7mm minimum. If you are using a special component that requires a 0.5mm hole, be sure to tell your PCB manufacturer. Otherwise you may end up with 0.7mm holes. A check by your EDA software, using the design rules set ("DRC" - Design Rule Check), will not show you this problem.

Holes larger than 2mm or 3mm are milled rather than drilled by most PCB manufacturers.

17. Oblong Holes

Slotted holes cannot be drilled cleanly and with a stable shape. These oblong holes must be milled. Milling cutters with a diameter of 1 mm or 2 mm are usually used. A 1 mm wide slot should be at least 2.5 mm long. For a 2 mm wide slot, the length should be at least 5 mm.

18. Milling

When separating PCBs from a panel, V-shaped notches are milled. Separating the PCBs is critical for assembled PCBs, as bending forces occur during breaking that can damage the PCB.

It is more common to separate the PCBs with a router, but this still leave small ridges. These ridges are then drilled – perforated with holes - to allow the PCBs to be separated from the panel without mechanical stress and risk.

19. Outline and Case

The outline of the circuit board should be such that it can be fitted accurately into an enclosure. Adequate air circulation should be provided if the circuit produces waste heat. Passive cooling is preferable to active cooling with a fan. If a fan is used, its vibrations should be decoupled from the case by rubber mounts. A temperature controlled fan with reduced speed ("silent") is optimal.

The case should have sufficient space for the cabling of controls and connectors, which often protrude well into the case.

For metal enclosures, it may be necessary to provide a ground connection.

For sensitive circuits, a hole in the enclosure should be provided for final adjustment. A plastic screwdriver with a slightly thicker shaft than a conventional screwdriver is often used for this purpose and should fit.

Some plugs have a plastic tab that locks into place. To remove the plug, you have to press it. Leave enough space on the side of the plug to reach the locking tab.

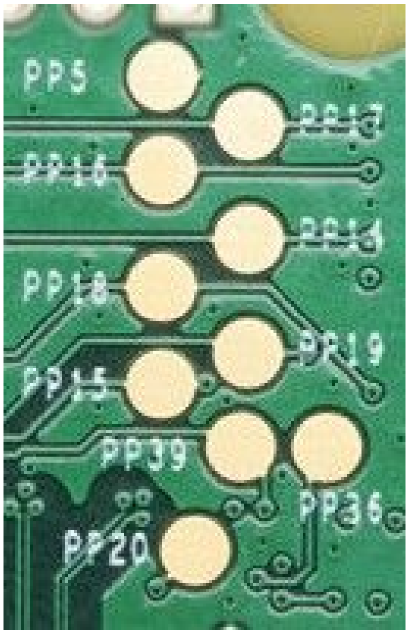
20. Maintenance & Repair

In PCB design, after placing the components, a 3D model of the board can be printed to check that the board fits. A 3D printer can be used to print a dummy of the board to investigate various mechanical aspects:

- Fitting accuracy of the case
- Collision testing for assembly and repair

If problems are identified at this stage, the PCB can be modified before the traces are laid out.

Plan for quick and easy disassembly and reassembly in case of repair. Make sure that the screws are the same (thread, length, type) and that all the connectors are easily accessible. Try to reduce the number of bolt variants. This will save time during maintenance and assembly and simplify component management.



Include test points in the layout to allow for troubleshooting and alignment of the circuit.

If possible, the test points should be easily accessible (e.g. on the side of the PCB) when troubleshooting with the unit powered on, without having to disassemble the PCB.

Plan test points for the power supply and, if possible and practical, for the signals in your layout.

Do not forget a central ground pin for connecting an oscilloscope or Logic-Analyzer.

Mark these test points and record their identifiers in the service documentation. Document these test points in detail, including expected values and signals you should see using your debugging tool.

21. Environment

A device affects the environment, but the environment also affects the device. So there are a few things to keep in mind.

If you are using LEDs to indicate operation and status, make sure they are not too bright in a dark room. A super-bright blue LED next to the TV in a dark living room or bedroom can blind everyone in the room with its strobe-like flashes.

When using LCD screens, it is important to set the correct viewing angle and viewing plane. The viewing angle is set when the LCD panel is manufactured. It cannot be changed.

The contrast of the display is changed by the orientation of the viewing plane. This is set by the "bias contrast voltage" or "bias angle". See the datasheet for your display.

A transparent cover is often placed over the display. This is usually polarised and can make it impossible to read the display in sunlight. This is often overlooked, especially on marine units.

As OLED displays are generally easier to read in sunshine, you should consider using them for outdoor equipment.

Again, consider the ambient temperature range, which should be taken into account in the design of the device.

22. Colour Blind

When designing a device, you can take other considerations into account, including colour blindness and colour vision impairment.

On average, about one in fourteen people who come into contact with your equipment will be affected by some form of colour vision disorder. This can include not only purchasers, but also service or repair technicians.

About 5-8% of the population is affected by colour vision deficiency, most commonly red-green deficiency. It is very rare for people to have a blue/yellow deficiency. A further 0.001% of the population is colour blind, i.e. only able to distinguish shades of grey.

A display with a combination of red and green LEDs would not be clearly visible. Connecting wires in colours that do not allow twisted insertion and clear identification also cause errors during installation, repair and maintenance. A choice of wires with wire markings is more sensible.

If you plan and consider colouring for the colour-blind and clear marking at the beginning of the design, there is no extra cost or effort involved.

Unlike professional users and repair technicians, people in the hobby sector and other parts of the world are not tested to see if they are colour blind.

23. Storage and Shelf-Life

As mentioned in the chapter on PCB surfaces, PCBs have a limited life when stored. It is best to check with your PCB manufacturer on the storage period for PCBs.

However, measures such as storage in a protective atmosphere can be used to extend the shelf life. If you are forced to store PCBs for longer than the normal shelf life because of equipment repair or contractual requirements, you must discuss the measures to be taken with an appropriate service provider.

Just as components may need to be stored in a protective atmosphere to prevent tin whisker formation, highly specialised PCBs may also need to be stored in a protective atmosphere. This prevents oxidation of the PCB from making it impossible to solder components.

24. Manufacturing & Assembly

Allow sufficient time for PCB production. The finer the structures and the more complex the PCB, the more time the PCB manufacturer will need. Time must also be allowed for electrical testing and shipping. For an overseas manufacturer, this can be a significant proportion.

If your PCB manufacturer also does the assembly, it may not be possible to provide them with components that are subject to export restrictions. This is particularly true in the aerospace and military sectors. This applies not only to the design, but also to individual components.

Even companies that specialise in PCB design will not get work from this sector if they are based in a country on the export restriction list, such as China.

Some countries also impose an import duty on components sent to the assembler. If possible, the assembler should source the required components locally to avoid this. On the other hand, there may be problems with the quality of the components.

Clarify in advance the file format of the design data required for the exchange and whether your EDA programme can handle it.

A local PCB manufacturer that you can talk to on the phone is easier than one overseas that you can only reach online. Accordingly, more or less must go into the layout and documentation to avoid misinterpretation. Although much of this over-cautious documentation may seem superfluous, it can make or break the return of working boards. On site, in the same time zone, this can all be made easier if the PCB manufacturer simply picks up the phone and calls when questions arise.

Try checking your design with an external tool. For a Gerber problem you can do this with the free tools "GerbView" or "gerbv".

The manufacturing specifications also include the type and colour of solder resist and the application of a placement print.

An electrical test, known as an 'e-test', can also be carried out. This tests the board for short circuits and open circuits before assembly.

25. Measurement

One of the first questions is always whether to use imperial or metric units.

Imperial inch units are common because components are manufactured with inch pin spacing. You may have noticed the 2.54mm pitch. All component spacings are multiples or parts of multiples of this, and are therefore referred to as "thou" and "mil".

The term "thou" refers to one thousandth of an inch, which is equivalent to 25.4 µm.

The term "mil" is a measurement used in printed circuit board design to indicate the thickness of copper plating and is equivalent to one "thou".

However, in English there is also a "mil" for a millimetre. To avoid confusion, it is preferable to use the term "thou" when designing printed circuit boards.

100 mil = 2,54 mm
1 thou = 1 mil = 1/1000 Zoll = 25,4 µm
100 thou = 0,1 inch = 2,54 mm
1 inch = 1 zoll = 25,4 mm = 2,54 cm

However, track widths are also often specified in "mil" and represent a globally distinctive size specification in this context.

Width in mil	Width in mm
8 mil	0,203mm
10 mil	0,254mm
12 mil	0,305mm

The basic rule is that in the context of PCB design, the terms "thou" and "mil" refer to the imperial system of measurement, and everything in the mechanical context, such as holes, outline and package size, uses the metric system of measurement with "mm".

Some SMD components may also have a metric pitch. Be aware of the possibility of confusion and, to be on the safe side, question every datasheet specification and double-check the dimensional information.

26. Snap Grid

Set the snap grid of your layout software to 50 thou (1.27 mm). A snap grid set to this value will allow you to centre the traces between the pins of the components. A finer grid of 25 thou (0.635 mm) is also commonly used, and is often the default value in PCB design software.

27. Components Library

The component library or database of your EDA software should contain all the components needed for PCB design.
For the components already in the library, it is important to check their specifications against the datasheet before using a component for the first time. In particular, you should check the pinout, hole diameters and package dimensions. If you can mark each component you have used, you can build up a selection of "trusted" components over time.

Such a library of components should be constantly updated. Components that are no longer available need to be removed, and newly available components need to be added. Some EDA software has a wizard that guides you through entering the required information from the component's datasheet.

This is a factor to consider when selecting your EDA software. It should be easy to include new components with their specifications and package design, and details of the protection zone around a component are also important.

There are component libraries that contain the part numbers for the component suppliers. This is very handy because you will get the component with a known quality. Calculation of the bill of materials can then be done easily and automatically.

There is EDA software packages have only a generic part library. In this case, you will not find an NE555 timer device, but only an abstractly defined IC in the 8-pin DIL design. EDA software that only has such a library can only be used for board layout. It is not possible to create a circuit diagram with such a selection of components, and it is not possible to simulate a circuit with such a selection, because the programme does not know anything about the electrical properties of the components. For simple circuits, where the board layout is developed without a schematic, such libraries can be used - but they are no longer state of the art for more sophisticated and complex designs.

If a design is to be simulated, a great deal of specific information needs to be available for each component. The advantage is obvious: a more accurate circuit simulation. Not only can you check functionality, but you can also simulate heat distribution and verify signal integrity in complex high-speed designs, taking into account the PCB layout. All this long before the first PCB is manufactured. This kind of circuit simulation has been available to everyone for a few years now.

If the component library is maintained by a different person, maybe the EDA programme must be able to access a central library. This allows component maintenance to take place simultaneously with PCB development, even on multiple workstations. In the high end of EDA software, this is standard; in the lower end, a central SQL database or shared folder can often be used. Components stored in a central location can then be shared by all workstations accessing them. This allows an effective division of labour. One person is responsible for component management, one for circuit design and one for layout. But they all use the same components from a central source, and consistency problems can be avoided.

If you can tag the components so that only those you have created are displayed, you can build up a reliable library of tried and tested components. Even in a small company, this selection of already used and therefore tested components is useful.

Whichever EDA software you choose, component management is a very important issue and should fit your workflow and requirements. Even for a hobbyist, an easy-to-use component library is important. If the database is too tedious to maintain, it will often be omitted.

28. Datasheets and Design-Guides

To design a circuit and layout a PCB, you need to know exactly what components you are going to use. For the simpler components, the datasheets will give you the necessary characteristics and often sample circuits. You will also find information on temperature range, power supply and much more. In this way, you will know how the manufacturer expects the component to be used. If you follow the general conditions given in the datasheet, you can assume that the circuit will work correctly.

More complex components, such as microcontrollers, have even more extensive documentation. In addition to the datasheets, there is a "Hardware Design Guide" and a "PCB Design Checklist" or "PCB Layout Checklist". If you follow these guides and checklists carefully, you can be pretty sure that your newly designed PCB will work right from the first version. Many design guides give examples of how to layout the tracks. But there are also practical tips, such as memory mirroring for DDR memory devices, the use of vias, or the impedances to use for differential-pair traces. For more complex circuits, these design guides are essential. Some manufacturers even provide layout files for the most popular EDA programs. The components can then be copied into your design along with the appropriate fan-out or escape routing. Don't hesitate to ask, it will not only save you a lot of time but also prevent many mistakes. The more complex your circuits and PCB layouts become, the more time you will spend working through the documentation that accompanies your components.

You will need the specifications you extract from the design guides at several points. For example, when you are planning how to build the board ("layer stack-up"), or when you are defining the design rules in your EDA programme. This means that DRC checks are not just driven by what the PCB manufacturer can produce in terms of structure width.

For complex designs, the reverse is true. The design rule specifications determine the design rule checks to be used. The PCB manufacturer is chosen based on their ability to meet and adhere to the specifications when manufacturing the PCB.

Take as much time as you need to review the datasheets and any other associated documentation. Overlooking information or specifications at this early stage in the design process can undo weeks or even months of work.

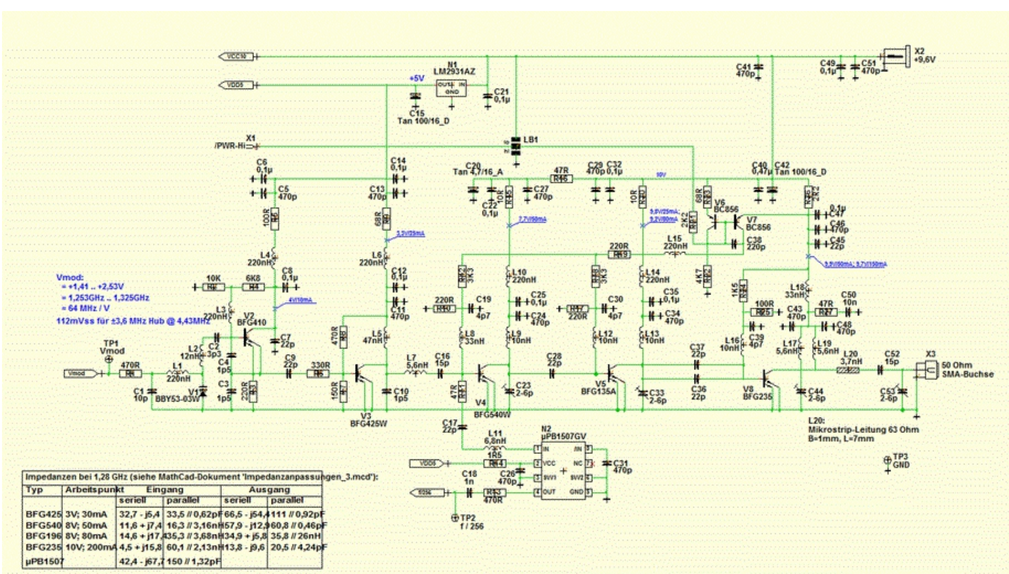
29. Schematic

A schematic shows how a circuit works. A layout only shows how a circuit is built. So you should draw a schematic for each circuit, not just a board layout.

The schematic is defined before the board layout. The components needed for the circuit are defined and selected. Ideally, the component will already be available in the component library, with a graphic symbol and a 3D package shape. This is where the advantage of a well-maintained component library, as mentioned in the chapter “Components Library”, becomes apparent.

If maintaining the component library is not a separate step in your workflow, each component used must now be cross-checked against the manufacturer's datasheet from which the component was sourced when drawing the schematic.

In particular, for wired components, the dimensions of the holes for the leads and the dimensions of the package must be checked. A twisted pin assignment can render an entire board unusable. Libraries supplied with the EDA software cannot automatically be assumed to be error-free.



The schematic therefore ensures the correct selection of components prior to board layout and prevents errors in the board layout.

The circuit can then be tested with a circuit simulator using the associated schematic to check basic functionality.

30. Circuit Simulation

Many EDA software packages have circuit simulation built in, making it easy to test a circuit. Simulating the circuit saves you from making careless mistakes. If the simulation software shows that the circuit works, then you have at least connected everything correctly.

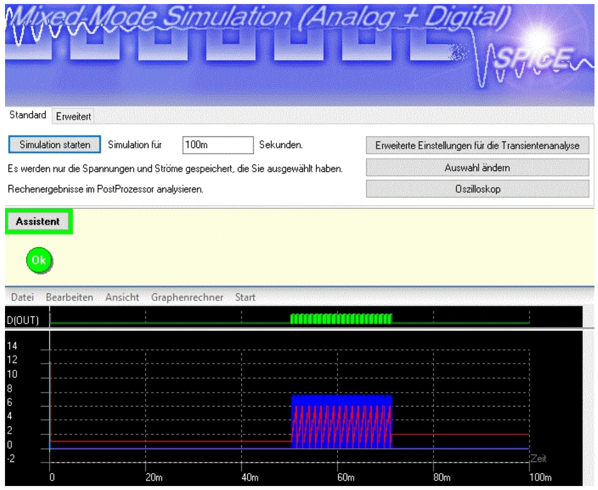
A widely used software and quasi standard for circuit simulation is SPICE. It can be used to simulate analogue, digital and mixed circuits. But there are a number of applications with a graphical user interface, more intuitive to use. They can be found under the names PSpice, LTspice, TINA, Multisim, ngSpice. The simulation software should be able to simulate mixed circuits (analogue and digital).

Running a simulation and evaluating the results requires some training. If the simulation is integrated into your EDA software, the associated documentation will cover how a simulation is carried out. If you are using SPICE or other separate software for simulation, you will want to consult their own documentation on how to enter the many specific parameters of a component for simulation.

Note that the simulation works with "ideal components". In reality, component values fluctuate, transistors of the same type never behave exactly the same, and component ageing is not taken into account. Inaccuracies in a circuit do not add up. But with a simulation you can at least check whether a circuit works in principle.

You can use the simulation to check which components play a critical key role and how changes to that component affect the overall behaviour of the circuit. In this way, you can influence the resilience and durability of your circuit in advance.

In the higher-priced segment of EDA software, more than just circuit simulation can be performed. Heat generation and signal integrity can also be simulated. Mechanical effects such as vibration and high acceleration can also be simulated to assess their impact. As a hobbyist, you should run a simulation, especially if it is built into your EDA software. Simulation has a firm place in PCB design. Familiarising yourself with them will help you and prevent mistakes.



31. Circuit Board Layout

Now you have a working circuit and a schematic, but how do you turn it into a finished circuit board? The schematic has shown us how a circuit works. The layout of the board is what makes production possible. It is important to understand that many additional decisions have to be made in the layout that were not considered in the schematic.

For example, the heat generated by the components and the size of the heat sinks required. Or the shape of the case in which the board will be installed. All this has to be taken into account during the design and layout phase of the PCB. As this is always a multi-stage process, you should not put yourself under pressure. Improving the layout several times is necessary and simply takes a lot of time.

Before you can start with the layout, you should already have defined a large number of preliminary considerations, or define them during the development:

- Dimensions of the board
- Fixing points
- Recesses
- Number of layers (This may change during design.)
- Fineness of structure (trace width, spacing)
- Selection of components
- Arrangement of components
- Arrangement of the circuit components in relation to each other
- Thermal measures
- Mechanical measures

These and many other points have been covered in the 'Pre Flight Check for your Design' chapter.

Create a general checklist, appropriate to your working environment, to help you remember important points in the future. Each new project will be different, but some items on the checklist will always be relevant.

Developments among PCB manufacturers in recent years have led to finer and finer structures in production. This is necessary not only for track width, but also for the use of very fine pitch semiconductor package designs.

Ask your potential PCB manufacturer about their design rules and limits. You can then set these in your layout software and use the Design Rule Check ("DRC") to check your layout. This will ensure that your board can be manufactured.

Think carefully about how you use the specifications and limits. If the manufacturer specifies a minimum distance between tracks, should you use the minimum possible value? Because then you are at the limit of what is possible with that manufacturer. But is it necessary for the circuit to work? It could also lead to a high scrap rate, which is reflected in the price. Or do you use coarser structures if this is possible for the intended layout? If the PCB is to be manufactured by another PCB manufacturer, then the lowest common denominator must be supported. Do not push the limits of what is possible. Find a reasonable compromise that allows you to use a different PCB manufacturer.

Here's a tip: When designing the layout, you should first consider the safety requirements. This could be, for example, high voltage clearances. Only then should you consider all other layout requirements, such as track spacing and width.

32. Virtual view of a printed circuit board

When designing a PCB in your EDA software, you always look at it from above, as if it were transparent. This means you look through the different layers of a PCB. This is a common feature of all EDA systems and means that, for example, a label is mirrored on the underside of the board.

The top copper layer is often called "TOP" and the bottom layer "BOT". The intermediate layers are numbered consecutively, e.g. with "IN1", "IN2" ..., or "1", "2" ....

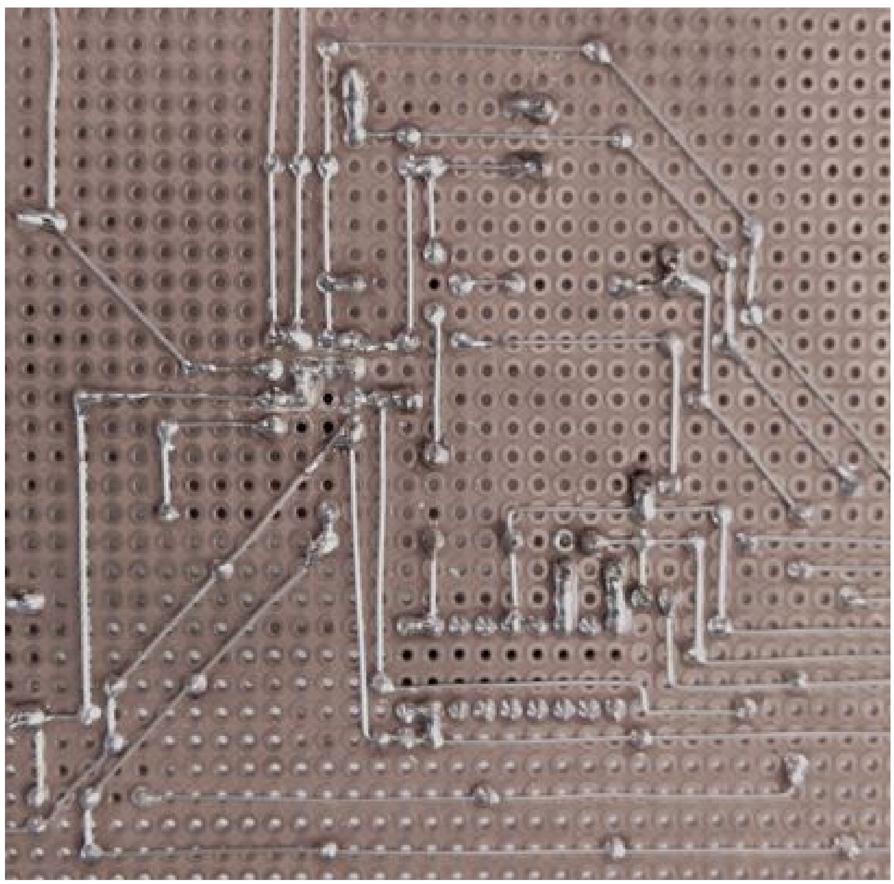
Only in the 3D view of a board do you see it in a natural perspective.

33. The Routing of the Tracks

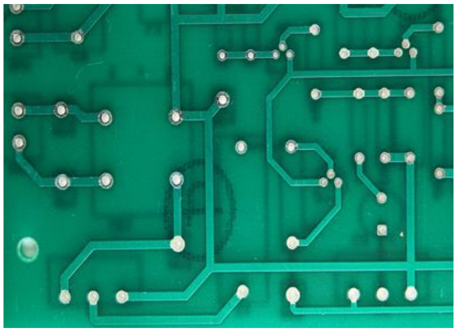
Tracks, along with components, are at the heart of a good PCB design. They connect the components and carry the power to the components. These tracks can be seen as 'just' doing their job or even as 'art'.

As a newcomer to EDA software, you may be wondering how to draw traces on a PCB. You are probably familiar with the process for a breadboard:

On a breadboard, the components are connected with silver wire.



A single-sided or double-sided PCB works in the same way. Whatever needs to be electrically connected is first virtually connected to conductive tracks in the computer using your EDA software. These virtual connections become copper tracks during the manufacturing process of the PCB.



After routing the tracks, the design would look like this:

When you route a trace, it should always take the shortest path between the components involved. The trace is then routed as directly as possible, but only horizontally, vertically or diagonally.

On a single-sided board, there is only one level, or layer, available for this purpose. As a result, a trace from component 'A' to component 'B' must also arc around the components in between. This increases the length of the trace considerably, but is unavoidable.

On a double-sided board, or a multilayer board, you can continue the trace with a via on another layer, and thus take the most direct route between component 'A' and component 'B'.

But which is better? The longer route or changing layers with vias?

The general recommendation is:

- The shorter route with Via is almost always preferable.

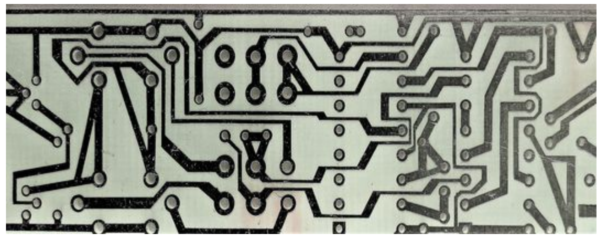
There are exceptions:

- When mass production of the board makes a large number of vias too expensive.

- The shorter path goes through a critical area, such as near a quartz oscillator.

- These are differential pairs. Then you also consider a longer path or move the components.

- You need a delay line, i.e. you want to affect the propagation delay.



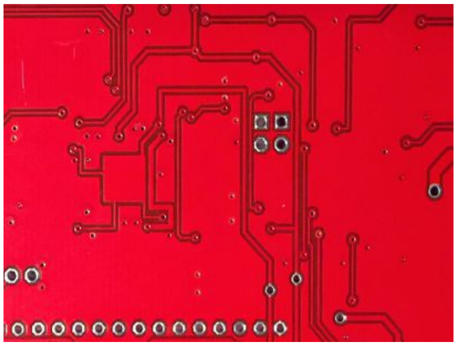
Pick up a couple of PCBs and look at the traces on the back. You will notice differences in the thickness of the tracks and how close together the components are. You will also notice how differently the PCB designers have worked in routing the traces. With a little practice and experience, you will see why and what is done and what is not.

On this hand-etched board, the traces are routed in a way that would not be considered "good" today. However, the design of this board worked excellently 30 years ago and it would still work perfectly well today. However, if the board were redesigned today, many things would be done differently.

Technical developments in PCB design change from year to year. PCBs designed today will not be "good" by the standards of the future; new methods will have proven themselves.

Therefore: If you have designed a PCB to the best of your ability today, it will work "well" today and in the future. Be proud of it, don't let your achievements be denigrated.

34. The Ground Plane



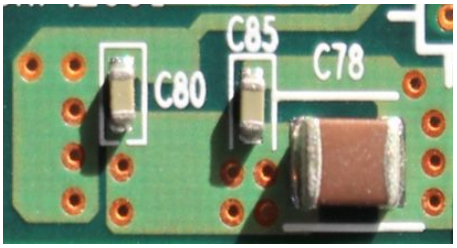
Nowadays, a technique is favoured for the PCB: everything that is not used for tracks becomes part of the mass and is used as a copper surface for a ground plane.

This is called "copper pouring". This copper now flows where there are no pads, tracks or vias and magically stays away from anything else that would cause a short circuit.

Everything that belongs to the ground potential is now connected to this ground plane.

So the remaining area on a single or double-sided board, everything that is not routed as a trace, is added to the ground plane and is supposed to improve the electrical properties of the board.

All opposing ground planes on a double-sided board, or in other layers of a multilayer board, are now connected by a vias.



Several vias are always used to make a good connection, a technique called "stitching" is used to place these vias to ensure a good electrical connection between these ground planes.

Here's a tip: A "copper pour" is not always an improvement. There are cases where it is better not to make the unused areas of a PCB a ground plane.

35. Track width

There is no general rule for trace width. The electrical requirements, space available and your own preferences will mainly determine the trace width you choose.

The finest trace width available will of course be the one your PCB manufacturer can produce. Generally you can go down to 8 thou (0.2mm), but from there you should check with your PCB manufacturer what is possible. Traces as wide as 3 thou (0.076 mm) are possible. However, the more complex the manufacture, the more expensive it will be. Try to use the widest tracks possible, especially those that carry power. Only use thinner tracks if space is at a premium. It is much easier to reduce the trace width in an existing layout than to make it wider.

Start with 50 thou wide traces for power and GND, 25 thou for normal traces and 10-15 thou to get between the pins of an IC. Keep at least 15-20 thou (0.3-0.5mm) between traces. For more densely packed SMD boards it is common to use 10 thou (0.254 mm) wide traces.

For many PCB manufacturers, the minimum trace width for standard PCBs is 0.2mm (8 thou). PCBs with finer structures are only produced on special request.

36. Track spacing

The spacing between traces also depends on the voltage applied to the trace and varies depending on whether the trace is on an inner layer or one of the upper layers. Refer to the IPC-2221 "Clearance for Electrical Conductors" guideline or the FED-22-02 design guideline.

These guidelines differ, sometimes significantly, in the minimum clearances.

A guideline in the IPC-2221 standard reads:
$0.6 + V_{peak} \times 0.005$ = spacing in mm

Practically applied:
0.6 + 20V x 0.005 = 0.7 mm = ~28 thou

In the newer IPC-9592B guideline it says:
V < 15V = 0.13 mm (5 thou) track spacing
"V > 15V and V < 30V = 0.25 mm (10 thou) track spacing
V > 30V und V < 100V = $0.1 + V_{peak} \times 0.01$

Practically applied to traces with a peak Voltage of 40 V:
0.1 + 40 V x 0.01 = 0.5 mm = ~20 thou

As you can see, depending on the formula used, the conductor spacing at 40 V can be less than at 20 V.

But remember: Minimum distances are always meant!

The following table is out of date and should only be used as a rough guide:

Track Spacing (simplyfied)			
Voltage (DC oder Peak AC)	Inner Layer	Outer Layer (<3050m)	Outer Layer (>3050m)
0-15V	0.05mm	0.1mm	0.1mm
16-30V	0.05mm	0.1mm	0.1mm
31-50V	0.1mm	0.6mm	0.6mm
51-100V	0.1mm	0.6mm	1.5mm
101-150V	0.2mm	0.6mm	3.2mm
151-170V	0.2mm	1.25mm	3.2mm
171-250V	0.2mm	1.25mm	6.4mm
251-300V	0.2mm	1.25mm	12.5mm
301-500V	0.25mm	2.5mm	12.5mm

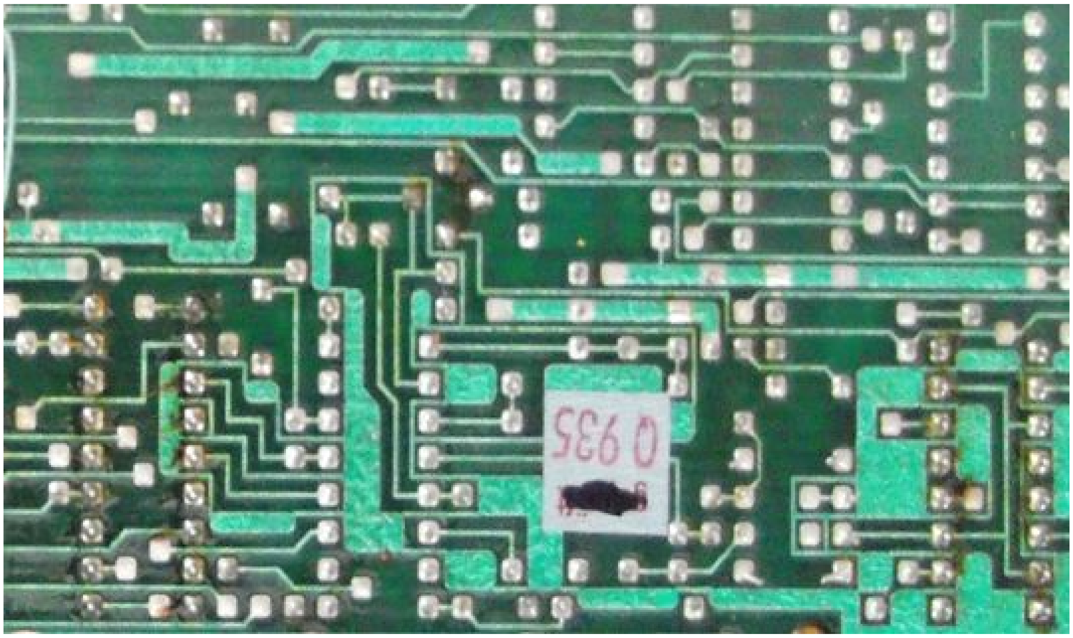
Nevertheless, this table is still used by many designers, so you should be familiar with it.

As you can see, the definition of minimum trace spacing is far from uniform. The more guidelines there are, the more opinions there are on the correct spacing.

Note that for many PCB manufacturers the minimum spacing between traces is 0.2mm. However, this is often dependent on the thickness of the copper used.

Here's a tip: Each layer of copper is etched in a bath and the etching time is different for thick and thin traces. If there is a mixture of very wide and very narrow traces, the manufacturer will have to work with average values, but this will usually not give optimum results. You should be cautious with a board that has such large differences in trace structures.

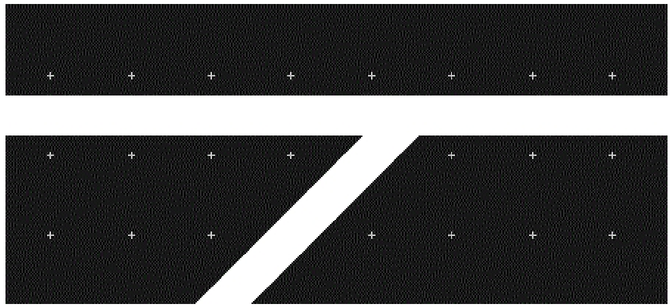
37. Tracks with 90° Angle



The fact that tracks were not usually laid at a 90° angle is also due to the etch bath. In the past, residues from the etching bath would collect at this angle and cause the track to become detached. Solder resist could also collect there. Although the switch to light-activated etching solutions has eliminated this problem, the 90° bend is said to "radiate" and should be avoided for EMC reasons. Various studies have supported or refuted this claim in recent years. To be on the safe side, it should probably be avoided. However, in some places a 90° bend is unavoidable and you can use it without hesitation.

In any case, avoid arguing with someone about it. If the person you are talking to is of the opinion that a 90° bend will cause the circuit to fail immediately and that only beginners do it... just agree with them and lay the tracks at 45° angles wherever possible.

38. Tracks with an Acute Angle



Do not connect tracks at an acute angle. As with the 90° angle, joining two tracks at this angle is said to have bad properties. In this case, however, everyone follows the rule. You will rarely find tracks on professionally designed boards that have been connected in this way.

39. Connection of SMD Solder Pads

Extensive rules have been defined for the approach of traces to SMD pads, at what point and at what angle the traces are allowed to meet the pad.

However, if you look at PCBs with SMD components from the last 15 years, you will see that these rules do not apply for everyone.

And the boards I looked at were all mass produced and worked.

So pay more attention to good routing and less to artificial rules.

40. Heat Generation from Tracks

The width of the track is determined by the current and the maximum allowable temperature rise of the track. A conductor track is made of copper with a specific resistance. When current flows through the trace, heat is generated. If the trace becomes too hot, it will separate from the PCB and will be destroyed.

To calculate the heat generated, you need to know the thickness and width of the trace. This will allow you to calculate in a calculation tool the maximum current that a trace specified in this way can withstand for a maximum temperature rise of 10 degrees.

Your EDA software may provide such a calculation tool. If not, you can download the free Saturn PCB Design Toolkit to perform this and many other calculations.

The IPC standard on which this table is based gives an indication of the trace widths to be used.

Trace width reference for 10°C temperature rise			
Ampere	Width with 1oz copper in thou	Width with 2oz opper in thou	miliOhm/inch
1	10	5	52
2	30	15	17.2
3	50	25	10.3
4	80	40	6.4
5	110	55	4.7
6	150	75	3.4
7	180	90	2.9
8	220	110	2.3

9	260	130	2.0
10	300	150	1.7

Do not use this table without cross-checking your actual conditions with a calculation tool.

41. Main Voltage

The spacing between tracks when they are not carrying mains power varies between layers. Tracks on the inner layers of the board will be closer together than those on an outer layer. The amount of voltage, and whether it is DC or AC, affects the spacing.

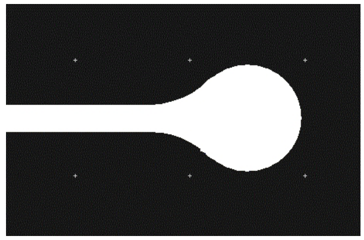
This is also affected by atmospheric pressure, i.e. altitude above sea level. Again, use the calculation tool in your EDA software or an external program to determine the optimum distance.

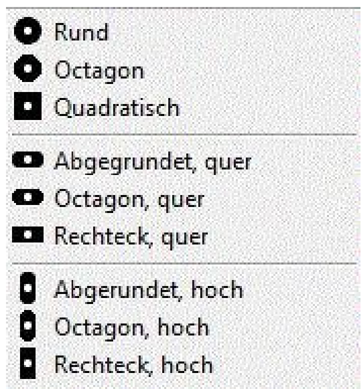
For mains voltage conductors, the required clearance can be found in the standards. DIN EN 60664-1 and IEC 60950 provide further information on this specific topic.

Here's a tip: As a rule of thumb, 1 mm of clearance is required per 100 V. The peak voltage at 230 V AC is 325 V, giving a clearance of 3.25 mm. For PC power supplies, a spacing of 5-6 mm is often used. Just to be on the safe side.

42. Pads

To solder the components on a PCB, you need solder lugs or solder pads. Solder pads are used for wired components and have a drill hole with which the component is soldered.





An SMD component needs a soldering point ("pad"). This is a small bare copper area on the surface of the PCB and the SMD component is soldered onto it.

The shape of the pad depends on the manufacturing process of the PCB and the component you are using. There are different shapes (teardrop, snowman, oval, square, round...) of what a pad can look like.

43. Teardrop-Pads

Teardrop pads are also a topic of conversation, as are traces bent at 90°.

Allegedly, liquid from the etching bath would collect in the corner between the trace and the pad. To prevent this, the pad is transformed to flow into the track in the form of a drop.

One advantage lies in the manufacturing process. If the layers are not perfectly aligned when bonding a multilayer board, the hole may not be exactly in the centre of the pad. With a teardrop pad you have a little more tolerance in one direction. This can determine whether the hole has separated the pad from the PCB trace, or whether some copper has been left next to the hole.

The size of the pads should always take into account the alignment tolerances when making the PCB, especially when drilling the holes. As the pads are drilled, the alignment of the PCB must be very accurate to ensure that the copper ring is drilled through the centre.

The ratio of pad size to hole width should be at least 1.8 times or at least 0.5mm larger.

Pads for wired components such as capacitors and resistors should be at least 70 thou.

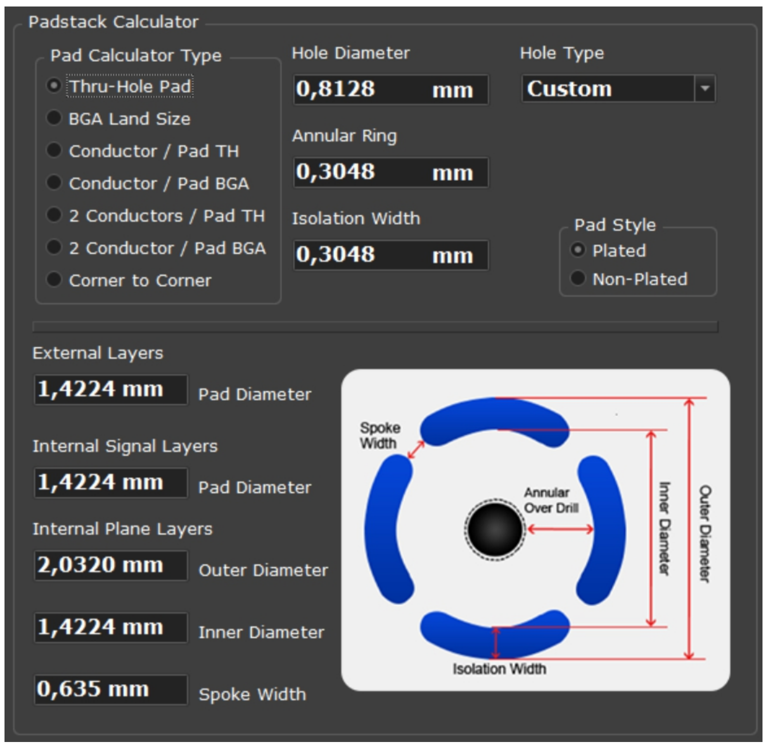
Components such as ICs or pin headers should have oval pads that are approximately 60 thou long and 90-100 thou wide. The pad for the first pin of an IC is used for orientation and alignment of the IC and should be rectangular, not oval.

Use this method for all components with pin numbering.

44. Padstack

A Padstack in a multilayer PCB define every aspect how a components pin will be fastened to a PCB and how tracks are connected to them.

It also defines everythin it surrounds: the sleeve around the hole, the clearance for the soldermaks, the connection to the planes on the outer layer and the layers inside the layer-stack, the areas for copper pads, thermal reliefs and clearance areas, and the solderpaste.



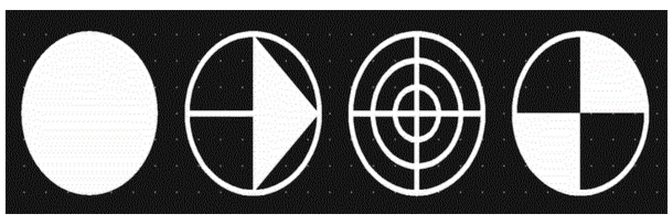
From a PCB designers perspective the drill hole may also be a part of the padstack definition.

45. Fiducials

Fiducials ("registration marks") are used to visually align the panel, or PCB, to allow accurate placement of components. Use at least two fiducials on opposite corners of your PCB. They do not have to be directly in the corner, but if they are diagonally opposite each other, it is optimal.

Your PCB manufacturer will also be able to tell you the best position and optimum number of fiducials for your PCB design, depending on their machines in production. Often these are placed on the connecting bars of the panel. For large PCBs, they are also positioned directly inside the layout to simplify assembly and minimize distances during the placement of components.

Fiducials are also needed for positioning SMD components with a fine pitch (FPGA, BGA, etc.). IPC guideline 7351 explains their use.



If you forget the fiducials, your PCB manufacturer may incorporate them in the necessary places in the CAM post-processing of the layout file.

Here's a tip: Fiducials sometimes differ depending on the PCB manufacturer. Check your board manufacturer's guidelines to see what they specify regarding the use and appearance of fiducials. For the general assembly of a board, the quartered circle is often used. For exact positioning of critical components, a fiducial that looks like a filled circle is used directly next to the component.

46. Clearance Zones

As already mentioned, the assembly of printed circuit boards is ideally fully automated. Placement machines work at very high speeds to place a component on the board every 0.09 seconds. To do this, the space where the component is to be placed must be free. Just as you need space for your fingers to grip a component, the placement machine needs space to position the components. Also, components cannot be placed underneath other components. All of this is achieved through the use of clearance zones.

In general, when placing components, it is important to ensure that there is enough space to mount the PCB. The dimensions of the PCB must be defined in advance so that it will fit into the enclosure that is usually already present. The dimensions should be known not only in two dimensions (length and width) but also in the available height. Otherwise, when creating the circuit diagram, a component could be selected that exceeds the maximum usable height of the installation location.

To prevent components from colliding with each other, the protection zone defined around the component must be taken into account. Components must not be placed too close to each other on the board, otherwise they will overlap, and violating this rule should only be taken as a deliberate risk in exceptional cases. The machine always relies on the protection zones being respected.

When placing connections, care must be taken to ensure that there is sufficient space for cable routing and that connectors can be accessed for repair. For example, connector contacts should not be closer together than human fingers can grip.

47. The Best Layer Stack-Up

Setting up the layers of a multilayer board is the first step in the layout process. Most of the time, as a maker or hobbyist, you will only use single or double layer boards. But for more complex circuits it may be necessary to use a multilayer board. The question of how best to use the layers, and the differences between them, quickly arises.

On boards with more than two layers, you should route the signals on the outer layers. This makes it easier to troubleshoot and use test points in the layout.

The inner layers should be as contiguous as possible, especially the ground layer. You may always need to define the layer structure differently, but placing the power and ground layers on the inner layers is a best practice.

As you start to develop your first multilayer boards, you will be wondering how to use the layers. The following table shows an example of the layer structure of multilayer boards as it is generally recommended and used.

Multilayer Stack-Up										
4 Layer	S	G	V	S						
5 Layer	S	V	G	V	S					
6 Layer	S	G	V	V	G	S				
6 Layer	S	V	S	S	G	S				
6 Layer	S	G	V	G	S	S				
7 Layer	S	G	V	G	V	G	S			
8 Layer	S	G	S	V	V	S	G	S		
8 Layer	S	G	V	S	G	S	G	S		
10 Layer	S	G	V	G	V	G	S	S	G	S
10 Layer	S	G	V	G	S	G	S	S	G	S

<u>Explanation of symbols:</u>

S = Signal-Layer
V = Vcc, Power Plane, Supply-Layer
G = GND, Ground-Layer, Reference-Plane, Ground-Plane

Rules of thumb:

- Inner signal layers are used for high speed signals such as data and address buses.
- Place data and address buses on different signal layers.
- For example, multiple supply layers are used for different supply voltages. (e.g. 3.3V and 5V).
- When constructing layers, follow the manufacturer's component design specifications or your customer's guidelines.
- Inherent interference is reduced if the signal layers are shielded from the power layers by intermediate ground layers.

Here's a tip: "Keep it Balanced” - Distribute the signal and power layers evenly. The copper content should be the same to prevent the board from warping. If the copper layers have different thicknesses, they should be distributed symmetrically around the core. The copper content of each layer should also be symmetrical. Some EDA programs have a "Copperbalance" function for this.

48. EMV and Layer

For EMC considerations, it is sometimes advised not to use a whole layer for power supply, but to route the power to the individual connection pins with a track. With this design, you can use another ground layer and achieve better reverse current behaviour. Unique ground layer return current paths for signal layers improve signal integrity. If possible, each ground layer should be assigned to only one signal layer.

A PCB with 4 layers can never be optimal in this respect and always represents a compromise. Only a PCB with 6 layers represents an optimal solution with the listed layer stackup (S,G,V,V,G,S) with regard to decoupling, EMC, signal integrity and reduced intrinsic interference. However, this leads to a limitation to only two signal layers. All other variants for a stack-up with 6 layers lead to a compromise in one direction or the other.

In order to achieve further improvements in a multilayer board with regard to signal integrity and crosstalk, not only the arrangement of the copper layers in relation to each other is important, but also the targeted use of core and prepreg material between the individual copper layers. Their different material properties can have a useful influence on the properties of the PCB.

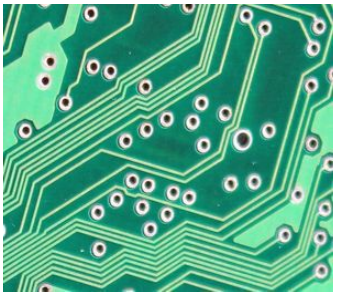
As you will have noticed, the ground layer is never on the outside of the PCB. Ideally, the ground layer should be a ground plane with as few interruptions as possible. For this reason, it is not placed on the top layer of the board, the component side. The components placed there would fragment the ground plane too much. This arrangement of layers also requires a higher number of vias.

On many boards from the hobby sector you can see that the remaining free areas on the top side of the board have been filled with ground planes. This is usually not necessary.

But when it is necessary, special design rules must be followed.

Although created with good intentions, these heavily interrupted ground planes cause sometimes more problems than they solve. It is better not to use them until you have acquired enough knowledge to understand the specifics of such a design. Not everything that has a positive effect on the PCB under certain conditions can serve as a general recommendation and should become a habit. The principle "much helps much" is not applicable in PCB design.

49. Types of Vias

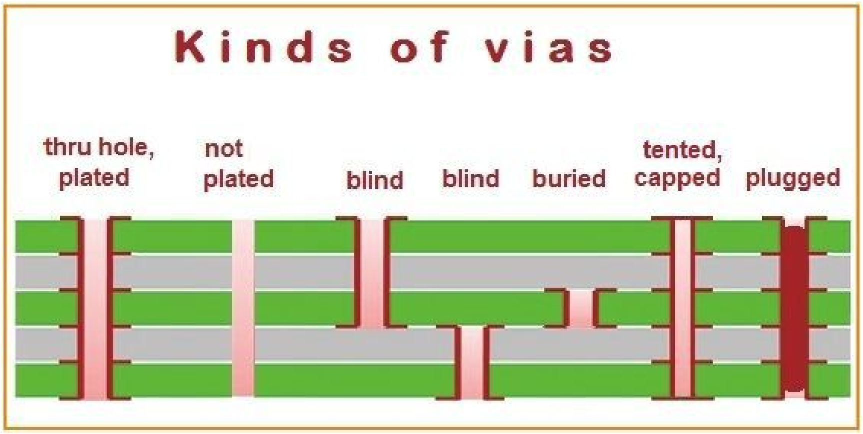


An electrical connection on the PCB must also be able to function across the copper layers. For this purpose, holes are galvanised, i.e. copper is deposited on the inside of the hole, which connects the layers of the PCB with each other. This through-hole plating, called "via", thus connects the top and bottom of a PCB, or different layers, and has a residual ring at its end that serves as a connection.

In the case of wired components, you can use their connecting legs as ducts at the same time. Do not use via directly under components unless they are for thermal pads.

There are other types of via, the most important ones being:

- A "blind via" connects an outer layer to an inner layer in a multilayer board. It is therefore also called a "blind via".
- A "buried via" connects one or more inner layers in a multilayer board and is not visible from the outside. It is also called a "buried hole".

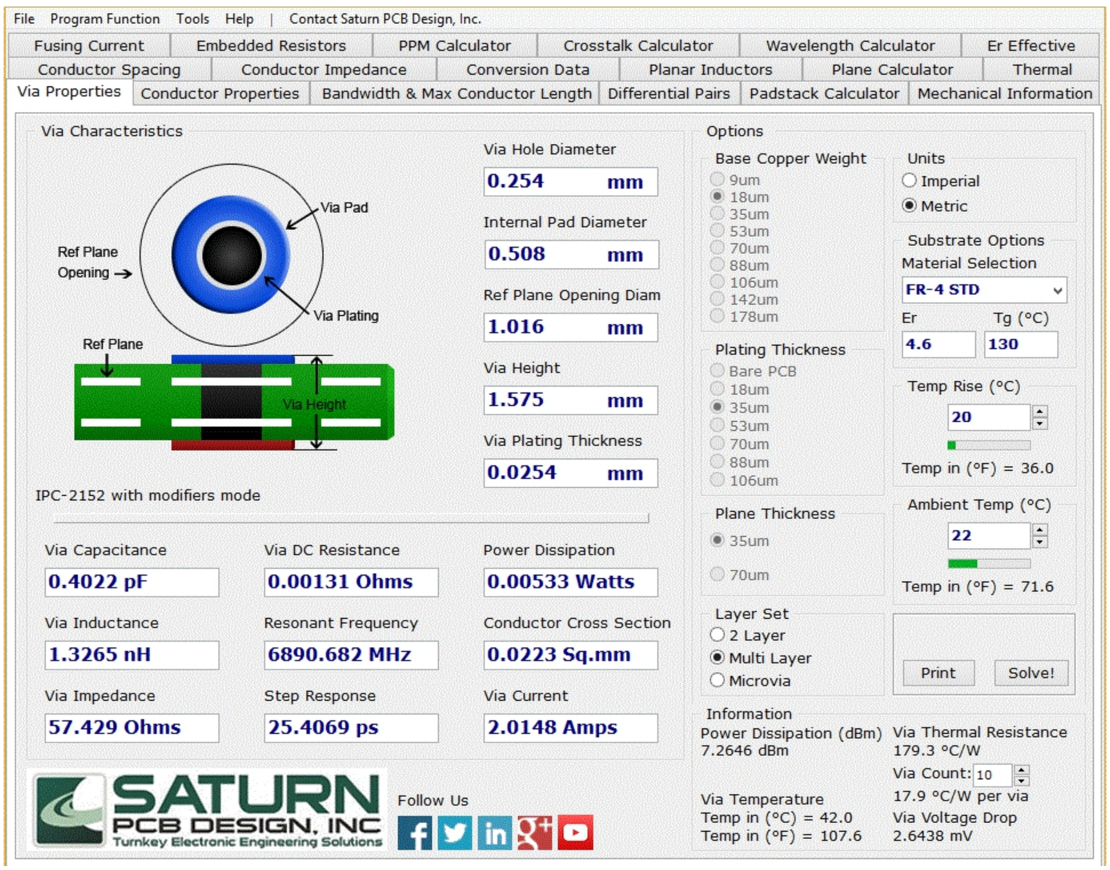


- - A "micro-via" is used on very dense boards and high-speed designs.

There are now a large number of via types with a wide variety of designations. No EDA software supports them all and no PCB manufacturer can implement all variants. If you have unusual requirements, you must first check whether they can be accommodated in the design.

50. Via Current Capacity

The current carrying capacity of vias depends on the diameter and thickness of the copper surface. A conservative rule of thumb is 1 Amp. However, you should recalculate the real value with your values in a suitable tool. Your EDA software should have this integrated, otherwise you can use the free Saturn PCB Toolkit.



Here's a tip: The diameter ("via hole diameter") is decisive for the current carrying capacity.

51. Juggling all the Balls

A PCB layout is not something that can be done in one go. The more complex the circuit and the more detailed the PCB specifications, the more tedious the layout can become. A layout is almost always a compromise. No layout is perfect. Some specifications limit the possibilities, so not everything can be done. The more you can think about placement, spacing, heat generation, mounting, space requirements, repairability, EMC, EMI and all the other details, the better the design will be.

But even if you had all the time in the world, you would never finish the design because there is always something to improve.

That's why it's important to set limits when thinking about the design. Not so tight limits that the task cannot be done, but not so much time that you get bogged down.

52. High-Speed / Low-Speed

The whole subject of high-speed design provides enough material for several books. However, the book you are holding in your hands is primarily intended to be a general introduction to PCB design. Therefore, I would like to give you a few basic rules about high-speed design to get you started.

There is no "one true and correct answer" to any of the following points, which are often illustrated by abstractions and simplifications. You will often meet people who have it all figured out. Do not be intimidated. If you have a problem, consult several sources and look for the solution that might fit in the intersection. Try a few things before you commit.

High speed design does not necessarily refer only to high frequency signals, but also to signals with a steep rising or falling edge. But what frequency is considered "high speed" is a matter of debate. In general, above 50 MHz, the easy-to-implement rules should be followed. Above 200 MHz, it becomes more difficult to maintain signal integrity. Above 500 MHz, impedance control and striplines are required. Above 1 GHz, balanced signal transmission ("differential signalling") is required. It is imperative that you follow the specifications and layout examples in the critical component datasheets. Much of the work has already been done for you.

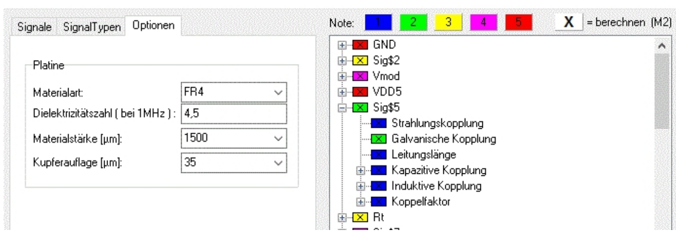
With the miniaturisation of electronics, increased clock frequencies, higher data rates in electronic circuits, combined with today's low power technology, circuits have become more susceptible to electromagnetic interference (EMI). This is a particular concern for signal integrity.

Electromagnetic compatibility (EMC) applies in both directions. The equipment should not be disturbed by electromagnetic effects (immission), nor should it emit its own interference radiation (emission).

Legislation (not only in the EU) stipulates that appropriate protection requirements must be met. These are laid down in the relevant standards; more on this subject can be found under the keywords "CE marking" and under Directive "2006/95/EC". A specialised test laboratory will be able to help you.

For high frequency signals or signals with steep switching edges, you need to consider the electromagnetic effects. The first thing that will cause you problems in your PCB design will be the rising and falling edges of the signals. The dielectric losses in the PCB material will cause the signals to attenuate significantly on large boards.

At high frequencies, a trace no longer behaves like a resistive connection between two points, but becomes a functional part of the circuit.



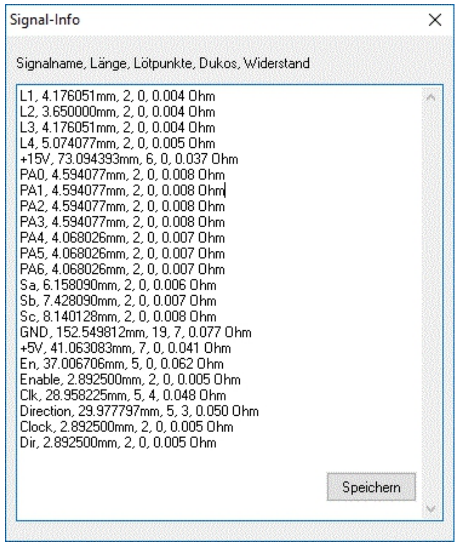
Your EDA software may offer an EMC analysis function to simulate and evaluate the behaviour of the PCB.

A digital signal, which in simple terms is a square wave of a certain frequency, consists of the clock frequency and its harmonics. These harmonics represent the frequency range up to the so-called "knee frequency" that the PCB must be able to carry. If the trace is too long for signals at this frequency, it takes on the characteristics of an antenna and acts as a transmitter. The interference caused by reflections from such a poorly matched 'makeshift antenna' will disrupt the signal integrity. This is why microstrip lines are used for high frequencies.

In low-speed signal transmission, the electrical signal is constantly present along the entire length of the conductor.

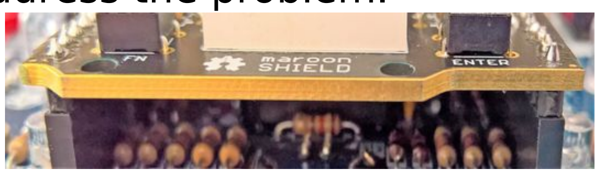
In high-speed signal transmission, the switching time of the component is only a fraction of the electrical propagation time of the signal along the conductor path. While the signal has already undergone a change at the front end of the conductor path, it has not yet arrived at the rear end of the conductor path. Wave packets are formed. Any disturbance in the path due to a change in impedance, e.g. due to a change in the width of the path, causes a partial reflection of the signal. This will travel back to the transmitter as a wave, superimposing on subsequent waves travelling forward. In the case of an antenna, this would be called a mismatch. As a result of this interference, correct operation of the circuit is no longer guaranteed.

Critical Trace Length		
Logic-Family	Switching-Time	Critical Länge
S-TTL	5,0 ns	36 cm
10KECL	2,5 ns	18 cm
AS-TTL	1,9 ns	14 cm
F-TTL	1,2 ns	9 cm
PCI Bus	0,7 ns	5 cm
100HECL	0,7 ns	5 cm
100KECL	0,5 ns	3,5 cm
GaAs	0,3 ns	2 cm



If you encounter such problems in your PCB design, first try to reduce the length of the affected traces by appropriate component placement.

Your EDA software may provide you with a tool to display the length of the trace. Together with the EMC analysis, you will be able to address the problem.



Another way to improve EMC and signal integrity is to metallise the edge of the PCB. When a board is in operation, generated but unneeded energy is radiated from the signal layers and supply layers over the edge area of the PCB. Such problems also occur when positioning high-speed connectors near the edge of the PCB because the reliable ground reference is missing there. Talk to your PCB manufacturer about thisYour EDA software may provide you with a tool to display the length of the trace. Together with the EMC analysis, you will be able to address the problem.

Here's a tip: These are just a few key words and rough summaries to make you aware of possible problems. If you feel you need to know more about high-speed design and want to learn more about the techniques mentioned here, consult the relevant books. It takes time, but it's better than messing up a PCB design through ignorance.

53. Multilayer - Low-Speed Design

The arrangement of the layers in a circuit board affects the behaviour of the circuit. In circuits with high frequencies more than in circuits with low frequencies. One can try to use the respective advantages for the different application purposes. Therefore, in low-speed design one has more freedom in layer stack-up than in high-speed design.

Power layer and ground layer do not have to be adjacent. In general, it is recommended to place these two layers directly next to each other in a multilayer board. In a low-speed design, however, it may make sense to place the signal layer between these two layers because the current takes the path of least resistance. In a high-speed design, the current takes the path with the lowest inductance. This is called the reverse current path.

54. Multilayer - High-Speed Design

A high-speed design dictates the layer stack-up due to its sensitivity. Deviation from this should only be done after careful consideration, as any deviation from best practice can negatively affect signal integrity and EMC performance.

The supply layer (power layer) and ground layer (ground layer) are placed directly next to each other.

The signals in a high-speed design always seek the path with the lowest inductance (return current path). Due to the capacitive coupling between the power layer and the ground plane, any interference that occurs is suppressed.

55. Thickness of Layers

The thickness of the PCB between the individual layers can be deliberately influenced. A thick insulation layer between signal layers can suppress crosstalk (coupling, crosstalk) and thus inherent interference. A thin insulation layer between the signal layer and the ground plane leads to a low impedance. It now depends on your PCB design and the structure of the layers which effects you can and want to achieve. Talk to your PCB manufacturer about what thicknesses of prepreg are available, whether they can produce PCBs with controlled impedance, and where the dielectric constant εr (Epsilon-R) of their composite material used as a PCB material lies. This is a very advanced technique used in high-speed design.

56. Why not two Signal-Layer Side by Side?

Do not place more than two signal layers side by side on a multilayer board. Supply layers or ground planes should follow above and below these two signal layers. If a third signal layer were placed between the other two signal layers, the return path of these signals would flow through the other signal layers and not through the supply layers or ground planes behind them, as the path with the lowest impedance would be sought. Avoid this layout at all costs. When designing, consider not only the direct signal connection to optimise the "outgoing current", but also the return paths (ground planes or ground tracks) to avoid "ground bounce".

57. Orthogonal Tracks

Two adjacent signal layers should always have a right-angled track layout. Route the tracks on one layer preferably horizontally and on the other layer preferably vertically. If you follow this rule as far as possible, you will prevent crosstalk of signals between traces that run parallel. This rule may not always be implemented, but the more critical the trace (e.g. clock signals) the more you should follow all the rules of the art.

58. Clock Signals

Traces with clock signals (CLOCK signal) should be laid between two layers with ground planes (ground layer). In general, the tracks to the clock generator should be kept as short as possible. If the trace of the clock signal is also on the signal layer, keep a larger distance to the neighbouring tracks. Also, do not place sensitive tracks near quartz oscillators or a ceramic resonator - keep a protective distance.

59. Guard Trace - Shunt Trace

A guard trace is a trace of 0V potential that acts as a fence around traces carrying clock signals or other system-critical signals. It prevents crosstalk and protects against leakage current flowing along the surface of the board.

A shunt trace is located just above or below a protected trace and follows it along its entire length. It should be three times the width of the trace to be protected.

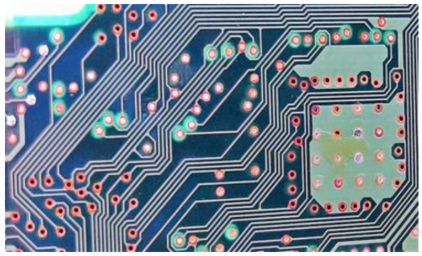
Guard or shunt traces do not improve signal integrity, they only protect it from external influences.

The use of guard traces is controversial in some applications. Simply increasing the distance between two traces to place a guard trace between them reduces crosstalk between the traces.

A guard trace is only used on single or double layer boards.

Differential pair traces do not require guard or shunt traces due to their functional principle.

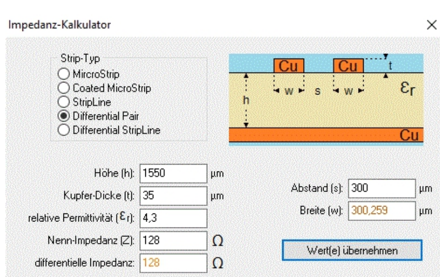
60. Differential Pairs



When routing differential pairs of tracks, we speak of at least two tracks that are routed across the PCB in dependence on each other. The same length is to be maintained for all traces involved. The tracks can also be meandered to maintain the length tolerance. If possible, perform length compensation between the two traces of a differential pair close to the signal source, i.e. at the beginning of the trace pair. However, length compensation ("delay tuning") between different differential pairs can be placed anywhere.

The distance and width of the two traces to each other and to the next ground plane influences the impedance (they should not be too close together), as well as the thickness of the traces. The length affects the signal propagation time and vias should be kept as small as possible.

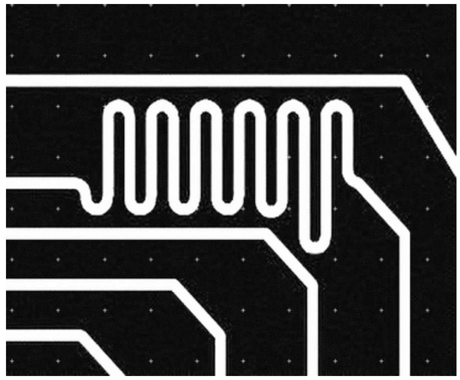
As you can see, there are a lot of rules to follow. Fortunately, your EDA software will take care of all these rules for you by allowing you to enter the relevant values. However, your PCB manufacturer may be able to give you more precise specifications, along with an appropriate layer stack-up.



For example, differential pairs are required for USB, PCIe and LVDS (Low Voltage Differential Signaling) interfaces. PCIe can have a maximum length difference of 0.1 mm at 2.5 Gbps. At 1 Gbps, it can be a maximum of 0.5 mm. For such critical PCB layouts, follow the specifications in the chip manufacturer's layout data sheet very closely.

Filling spaces between differential trace pairs, e.g. with ground planes, has an impedance changing effect and should be avoided. Better EDA programs will show you the disturbance of the spaces as "Uncoupled Length".

61. Propagation Delay



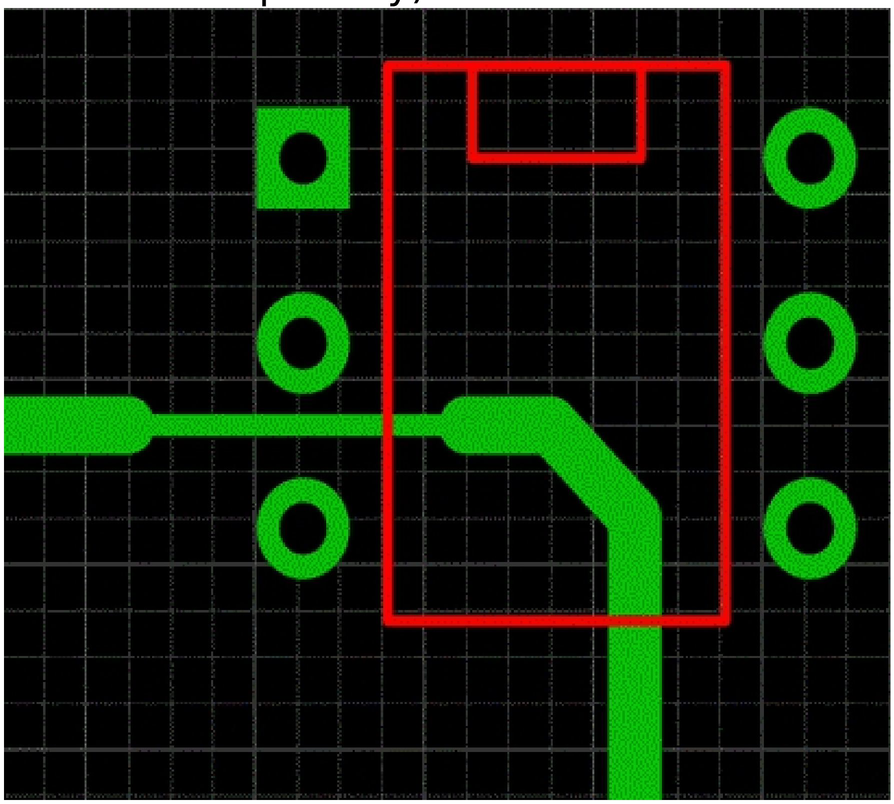
Traces can be meandered to delay a signal because you want to achieve a particular propagation delay for a signal. This 'propagation delay' can be calculated by a tool and the result of the calculation is used to extend a trace by the specified value. You may want to change the length of a trace, for example to match the propagation delay of the trace to other traces.

This is used, for example, in DDR SDRAM memory chips. In a data bus, the traces must be the same length. Depending on the type and arrangement of the memory chips on the board, the traces can be laid in a "T-branch" or "fly-by" topology and should follow the relevant specifications and rules very closely. The processor's design guide will provide layout examples of exactly how the routing should be done. In this case it is more complicated because the clock signal and the various buses (clock-to-address/clock-to-strobes/strobe-to-data) must also be properly timed. The manufacturer's datasheet also provides precise design guidelines for the tolerances to be maintained.
To minimise the impact on signal integrity, the tracks are meandered (delay tuning, meandering or serpentine routing).

Here's a tip: The manual routing of tracks for DDR memory chips can take weeks. There are a number of rules that must be followed meticulously. Some EDA programs can automatically route such traces based on predefined design rules. This can save a lot of time! The result need not be worse than the manual way. In the beginning, you should do this manually a few times to internalise the rules. Only then will you be able to judge whether the automatic tool has produced a useful result.

62. Necking

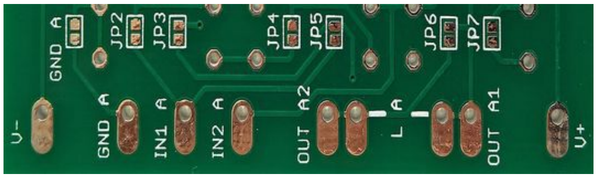
Reducing trace widths ("necking" or "necking down") to be able to route between the connecting legs of a component should not be used on critical traces (in terms of signal integrity and current quantity).



63. 1:1 Printout

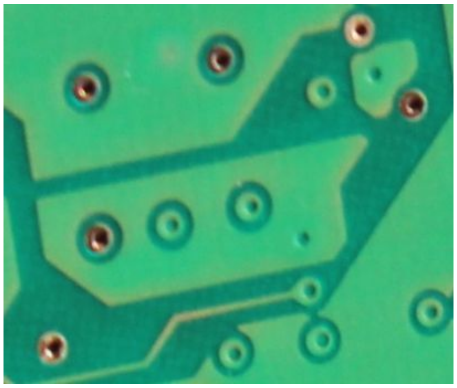
Print out the board in 1:1 format on paper and place the components on it. Errors will quickly become obvious this way.

64. Elongated Pads



The pads of SMD components are often made slightly larger if they are to be soldered by hand for the first prototypes.

65. Unconnected Ground Planes



Ground planes and ground layers should be as contiguous and undivided as possible. Also, do not form islands that are only connected to the rest of the ground plane by a narrow bar. Do not leave any free-standing and unconnected ground islands.

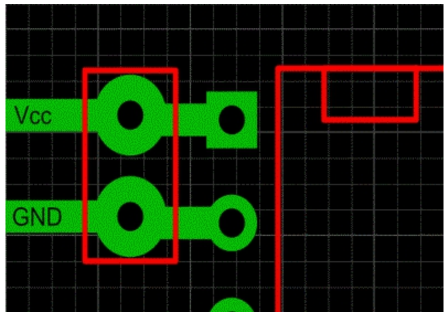
66. Bending Radius

In the case of axially wired components, such as electrolyte capacitor with welded connection wire or component with glass bodies, the bending radius must be taken into account, otherwise too much mechanical stress could be exerted on the component during assembly if the drill holes are too close together, thus causing damage in the connection.

67. Spring Tension

Heatsinks that are pressed onto a component to be cooled with spring force will bend the PCB. A metal support should absorb the force applied to the underside of the PCB. Alternatively, the PCB should be thick enough not to bend..

68. Blocking Capacitors



All ICs in your circuit should have capacitors as close as possible to the power supply pins. Here the capacitor is between the IC and the power supply! Such blocking capacitors prevent the supply voltage from collapsing when the ICs draw high current in pulses.

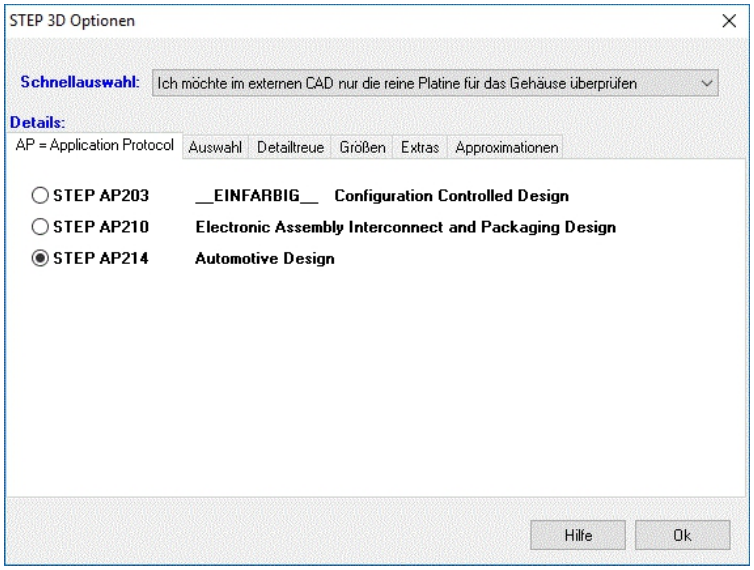
These so-called switching currents in digital circuits generate steep-edged pulses on the supply lines. Such a capacitor keeps the voltage in the rest of the circuit stable.

Use ceramic or flat film capacitors with a low ESR (equivalent series resistance). In the past, a value of 47 nF or 100 nF was used as a general rule for normal ICs. In today's high frequency digital technology, the impedance of the capacitor connected must be as low as possible. The data sheet for your specific IC will tell you what value the blocking capacitor must have and which type is suitable.

An impedance plot of the capacitors you use with the values 1 nF, 10 nF, 47nF and 100 nF will show you the most suitable value for your frequency.

69. Custom-fit Enclosure

Some PCBs have special dimensions and the case is also custom made. Or such an enclosure is to be printed in a 3D printer.



To ensure that the PCB still fits exactly into the special enclosure, you can export the PCB as a CAD file in STEP format. This is then loaded into the CAD programme, which is then used to design the enclosure for an exact fit. This data transfer ensures that the PCB fits exactly into the enclosure and that the drill holes are correct.

The STEP export settings usually allow you to select exactly what is to be exported. Either the PCB only, or the PCB with components, so that the height of the assembly can be taken into account when designing the enclosure.

70. Microstrip and Stripline Design

Microstrip traces are used in high frequency technology. They come in many different forms, with very different characteristics. This is the crux of the matter! Your EDA software must be able to distinguish and map these variants. Be absolutely precise in your definition and documentation so that there is no room for misinterpretation by the PCB manufacturer. Double and triple check!

71. Layer Numbers

Label each copper layer with a layer number. For a 4-layer multilayer board, put a "1" in the copper layer of the first layer, the top side. In the following inner layers a "2" and "3". The bottom copper layer gets a mirrored "4".

Each number is offset to each other so that when you view all the layers, the numbers can be seen next to each other. If you have only shown one copper layer because you want to work on this layer in a concentrated way, you can make sure where you are by looking at the number shown. This prevents you from accidentally working on the wrong layer.

72. Project Documentation

The project documentation, i.e. the documentation of all aspects of a design, the layout files and the drawings, should at no point force the user to interpret the content. The documents should also be prepared in such a way that there is no need to search for details and no information is overlooked. This is not an exaggerated perfectionism, but a way of avoiding unnecessary extra work and costs. Particularly in the case of projects that are to be updated and revised after a number of years, the assumptions that were common knowledge at the time of creation must also be documented. These assumptions change over the years due to new production methods and have an unwanted effect on the layout and circuitry.

73. Board Labeling

Label the board with project name, design name, version number, date, creator and company name.

Give all LEDs meaningful names, such as "PWR LED" or "Status LED". Connectors, fuses, jumpers could be labelled "Pwr Con", "Display Fuse" and "Cfg Jmp". The pins of a pin header or programming interface are easier to use for debugging work if they are clearly labelled and prevent confusion and incorrectly connected test devices.

There is enough space on almost every board to use the silkscreen for this and it costs no extra.

74. Test Points

Test points, or SMD pads for ISCP, should not be tinned. Spring contact pins ("pogo pins") will then not function reliably. Vias that have not been covered by solder resist ("tented"/"untented") can be used as test points if necessary.

75. Settings for Routing

Before you start routing, you need to set some basic parameters. How thick do you want the tracks to be? Should the power traces be thicker than the signal traces? What, if anything, was determined during the preliminary design considerations? Where are the forbidden areas, where no tracks should be routed? In which areas of the board are critical traces not allowed?

Use your board manufacturer's DRC specifications as a minimum, and revise the settings a second time with your own markups.

You can save the DRC settings as a profile in your EDA software so that they are always valid and reusable. However, check with each new project to see if new guidelines or specifications have come into effect and your settings need to be adjusted.

76. Autorouter

At some point in the past, it was thought that computers could do jobs described with rules and specifications better than humans. At that time, the autorouter was invented. Completely ignored was the fact that work based on experience, intuition and skill is also called craftsmanship. When work becomes art, the result is more than the sum of its parts. The computer today can often perform tasks very well nowadays, but what it can never have is that one crucial idea, the knack, the break with the rule that can solve a problem and be elegant at the same time. The autorouter, however, is still with us.

An autorouter is often desired as a feature. But its usefulness is limited. The more complicated a PCB design becomes, the more inexperienced PCB designers want an autorouter to do the complicated work for them. But the more complicated the routing of the tracks becomes, the more unsatisfactory the results of the autorouter become. Even with the autorouter, you don't come close to the ideal goal of "design with the push of a button".

Let's assume you want to design a single-sided PCB. You define the size of the board and position the desired components on it. Since you have previously drawn a circuit diagram, the connections between the components are drawn in as air lines (the so-called "rubber bands" or "airwires"). Now you want to have these connections laid as traces and you start the autorouter to do its magic. The result can is predictable: it will most likely fail. There will be air lines left over.

This has nothing to do with the quality of the autorouter you are using. It has to do with the fact that on a single-sided board all the traces have to be routed so that they do not cross each other. On a single-sided board, if the traces can only be placed on one plane, there is simply not enough room for them to avoid each other. The number of connections and the number of planes that can be used to route the tracks can always lead to a point where not all the tracks can be routed. In this case, there is no way to route the trace so that it finds enough space or so that it does not cross another trace.

With vias, the Autorouter can try to work around this problem by changing layers. However, this requires at least a two-sided board. It can happen that the traces the autorouter places at the end of the routing are criss-crossed through the board. But you have to check if this is good for the trace. Depending on the task of the trace (it could be a critical clock trace), it could have a negative effect on the function of the circuit. Or it might not.

If possible, try to make the routing so simple that you can do it without the autorouter by rotating and repositioning components.

This will seem difficult at first. Like an insoluble problem that only wastes time. But this is the kind of experience you need for successful and error-free PCB design. Eventually, you will develop the necessary intuition to understand how the components need to be placed for the routing to work.

If you reach a point where you can't make any progress, you can try the Autorouter. Save the current state and let the Autorouter try to route the tracks you can't. Sometimes you just miss an opportunity. Watch the Autorouter and see if you have already tried that route. With experience, the Autorouter won't be able to fool you, but sometimes it's worth a try.

An autorouter cannot tell what a track is used for. It does not know how to avoid interference or which components cause interference. So when routing tracks, you should connect the power supply first. Then follow the tracks that are susceptible to interference. Then the tracks that represent clocked data buses (memory components). Tracks carrying signals below 50 kHz can be routed without concern. However, special attention should be paid to certain areas of the board. In the vicinity of crystals and clocks, with the associated external circuit with capacitor, no trace should be routed too close, as it could be adversely affected by them.

As you can see, there are a handful of considerations to be made for each component, and then the best compromise must be found for the situation. It is precisely when there is no optimal answer and things get difficult that the Autorouter cannot find a simple solution.

Also, the star-shaped merging of ground lines, as is necessary in some circuits, cannot be done by an autorouter and must be done by hand.

Remember that an autorouter can only do the routing for a part of the tracks to be routed. In some cases, if there is enough space and the traces are not critical, you can let the autorouter do the hard work. You can always route other traces by hand afterwards. This mixed approach can save time.

High-end autorouters come with a few more settings and options that can do a lot at the touch of a button. These can produce impressive results. Especially when routing memory chips and similar high-speed components, an autorouter's routing support is often used with good results.

77. Interactive Routing

Another special feature is "Interactive Routing", "Follow-Me Routing", "Sketch Routing" and "Push and Shove Routing". Even if these features are controlled manually, an autorouter is active in the background to perform the required routing. Have a look at some videos on the internet uising these keywords. It is very exciting to see how these autorouters work. You just decide where you want the tracks to go and the autorouter takes care of the details.

78. Autoplacer

The autoplacer tries to place the unplaced components on the board in a sensible way.

When there were endless rows of TTL ICs on a board, an autoplacer could swap them back and forth until the trace lengths were minimal. This did not mean that the positioning of the tracks was optimal, but at least you were moving in the right direction.

However, with today's complex layouts, there are many specifications to consider. An autoplacer quickly reaches its limits and is of little use. It cannot know that one transistor needs a large heatsink and needs to be placed with a lot of clearance, but the other transistor with the same package shape is not affected.

Repair and maintenance considerations are also not taken into account.

There are many examples of heat dissipation, RF emissions and critical lead lengths that an autoplacer cannot avoid when positioning components.

Try the autoplacer of your EDA software and see what settings can be given to it and watch the results, but do not rely on it.

79. Design-Files Formats

A modern EDA software should support different output formats, at least the Gerber format should be supported. This output format is at least necessary for the data exchange between development and production. Gerber files are used to more than 90% for the output of layout data in PCB development.

Standard Gerber (RS274D)

The original Gerber X0 format was developed in 1980 and was the quasi-standard for controlling photo plotters. These were used to produce the foils for exposing the printed circuit boards. However, it contained a lot of room for misinterpretation, resulting in allocation and format errors. Standard Gerber is obsolete today.

Extended-Gerber (RS274X)

Extended-Gerber is also called Gerber X1. In 1998, the Gerber format was revised and the possibility of defining complex objects such as lettering, measuring marks, areas and pads with special shapes was added. X-Gerber represents today's standard for the exchange of PCB data and has a distribution of more than 90% for data transfer.

Gerber X2

Is a compatible extension to Gerber X1, released in 2013, and includes more attributes and definitions for layers, pads and vias. It includes bare board attributes and therefore full bare-board fabrication info.

Gerber X3

Compatible with ECAD-Software that can open Gerber X2 and X1 files. The final specification was released in 2020 and includes more assembly attributes to identify centroid, pin 1, component outline and more. It also adds new top and bottom component layers for bare-board and full assembly fabrication info.

ODB++ and ODB++(X)

ODB++ is a file format designed to avoid many errors and misinterpretations of data in PCB manufacturing. It was introduced in 2010. All data for production, assembly and test are in a single file.

With ODB++(X), the data is in an XML structure. It is considered a quasi-standard among PCB manufacturers and is being further developed by the IPC as standard IPC-2581.

IPC-2581

The IPC-2581 file format was developed by a consortium of companies in the electronics industry.

It provides (like ODB++) for a cross-company and cross-process flow. From PCB design to manufacturing, including details for tooling, fabrication, assembly and inspection, everything is embedded in a single data format.

It was first published as IPC-2581A in 2012 and underwent a revision in 2013 as IPC-2581B.

This consortium of companies has complex PCBs produced worldwide. The data output in this new format is intended to be more unambiguous and error-free.

80. A Gerber output file can contain some pitfalls

When reviewing your Gerber files, you may notice that large copper areas are displayed with a striped pattern. In this case, your EDA software generated the area by filling it with parallel lines. This method is quite common, but it is also a relic of the past and thus only represents the possibilities of standard Gerber.

With the extended Gerber (RS274X) format, this has supported the area filling of polygons since 1998. If a ground plane is created on a layer of the board, it is filled with an "area fill" in X-Gerber. This is the current standard and less error-prone for the board manufacturer.

In the X-Gerber output dialogue of your EDA software you may find an option how to fill these areas. You may find a selection for polygons with the option "fill with lines only" and "G36/G37 Area Fill".

If your EDA software does not display or support this, your board manufacturer will use their CAM postprocessor to make the necessary corrections for their manufacturing technique. Each board manufacturer will use their CAM software to translate your layout files to control their CNC machines. In the process, the output files of the common PCB design software products are also corrected.

Whether you use OrCad, Altium, DipTrace, Sprint-Layout, CircuitStudio, Eagle, or any other software, there will be peculiarities and errors in the output files. A PCB manufacturer knows these. Because his manufacturing process must run error-free, he has discovered and eradicated many of them over time. These corrections are then applied in CAM processing and the design files are cleaned of the errors. Nevertheless, this is error-prone and a residual risk remains. Misinterpretations can occur with a new programme on the market, updates with new features, or just bad luck, and a board with errors is produced.

Try to check your output files with an external Gerber viewer to avoid surprises.

81. NC Drill / Excellon Files

NC drill files, also known as Excellon files, are used in the manufacturing process of printed circuit boards (PCBs). These files contain information about the locations, sizes, and shapes of holes that need to be drilled into the PCB. The NC drill files are read by computer numerical control (CNC) machines that control the drilling process.

Here's why NC drill files accompany Gerber files in PCB manufacturing:

- Hole Information: Gerber files contain information about the copper traces, pads, and other elements on the PCB, but they don't include details about the holes. The NC drill files provide the necessary information for drilling holes accurately, such as the hole diameters, positions, and the types of holes (plated or non-plated).
- Drilling Automation: By using NC drill files, the drilling process can be automated. The CNC machines read the instructions from the NC drill files and position the drill bits precisely at the locations specified, ensuring consistent hole placement across multiple PCBs.
- Alignment and Registration: NC drill files assist in aligning the drill holes with the existing circuitry on the PCB. The CNC machines use the reference points from the Gerber files to align the drill holes accurately, ensuring proper registration between the drilled holes and the copper traces.
- Manufacturing Accuracy: Combining Gerber files with NC drill files allows for higher precision in PCB manufacturing. The accuracy of hole placement is critical, as it affects the functionality and reliability of the final electronic device.
- Different Manufacturing Steps: PCB manufacturing involves multiple steps, such as copper etching, solder mask application, and component placement. The NC drill files are used specifically for the drilling process, while the Gerber files are used throughout the entire manufacturing process.

Both files work together to ensure that the final PCB meets the design specifications.

In summary, NC drill files are essential in the manufacturing of PCBs as they provide instructions for drilling holes accurately. They accompany Gerber files because Gerber files focus on the copper traces and other elements on the PCB, while the NC drill files focus solely on the hole information required for drilling. Together, these files ensure proper registration, alignment, and manufacturing accuracy in PCB production.

82. EDA Format Design-Files Handover

Many PCB manufacturers also accept the PCB design data in the file format of the EDA software. This saves the user having to export the data as a Gerber file, and the saved conversion on the user's side prevents errors from creeping in.

The PCB manufacturers are in contact with the EDA software manufacturers and are provided with conversion software by them. This is integrated into the CAM process and the conversion, with the corrections required on the manufacturer side, is carried out directly using automatic methods.

Overall, this is less error-prone and the additional effort is manageable.

It should be noted here that every layout file, regardless of format, is interpreted by the PCB manufacturer, i.e. everything is translated into machining steps to be carried out for production. Just because you have defined something in a layout file in some way, it does not mean that it will be manufactured that way. Depending on the machines used and depending on the manufacturing process, small changes may be made to your design. With a bit of luck, the PCB manufacturer will also discuss these with you beforehand if they are necessary. It is also fair to say that a good PCB manufacturer will often know better than you what is needed in a layout for you.

For your first PCBs, use a local PCB manufacturer with whose customer service you can communicate easily and in your language. Discuss with them which file formats are accepted for the layout data. Simplification during handover leaves less room for misinterpretation and errors.

83. Components Designators

Switch symbols and designators were standardised as early as the 1940s. The following list is a compilation of common designators, which is a mixture of many different norms and standards.

Designator	Component (englisch)	Component (german)
B, BT, BY, BAT	battery	Batterie
C	capacitor	Kondensator
CB	circuit breaker	Trennschalter
CN	capacitor network	Kondensator Netzwerk
CON	connector	Steckverbinder
CP	connector, adapter, coupling	Stecker
CR	diode	Diode
D	diode	Diode
D	integrated circuit (digital)	Digital IC
DC	directional coupler	Richtkoppler
DL	delay line	Verzögerungsstrecke
DS	neon bulbs	Neonröhre
E	terminal	Klemme
F, FS	fuse	Sicherung
FD, FID	fiducial	Passermarke
FL	filter	Filter
G	generator	Generatoren
H	LED, light emitting diode	LED, Leuchtdiode
H	mounting hole	Montagebohrung
H	horn	Hupe
IC	integrated circuit, chip	IC

ISP	In Service Programming	ISP Anschluss
J	stationary connector	Anschlussleiste
J	connector, female	Anschlussbuchse
J, JP	jumper header	Anschlussleiste
K	relay	Relais
L	Inductor	Spule
LED	LED, light emitting diode	LED, Leuchtdiode
LS	loudspeaker, buzzer	Lautsprecher, Summer
M	meter	Messinstrument
M	motor	Elektromotor
MG	motor generator	Generator
MK	mikrophone	Mikrofon
MP	mechanical part	Mechanisches Bauteil
N	integrated circuit (analog)	Analog IC
P	movable connector	Flexible Steckvorrichtung
P, PL	connector, male	Stecker
PS	power supply	Stromversorgung
PT	thermocouple	Thermoelement
Q	transistor, thyristor	Transistor, Thyristor
R	resistor	Widerstand
REL, RLY	relay	Relais
RN	resistor network	Widerstandsnetzwerk
RT	thermistor	Thermistor
S	sensor	Sensor
S, SW	switch	Schalter
T, TR	transformer	Trafo
TB	terminal board, terminal strip	Klemmleiste
TC	thermocouple	Temperatursensor

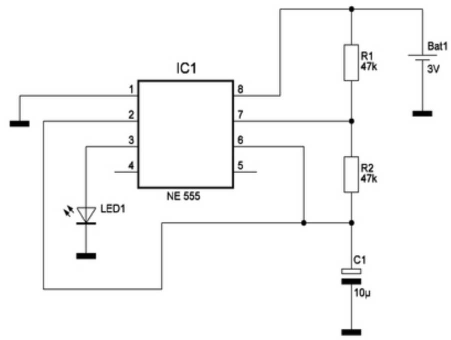
TP	test point	Testpunkt
TZ	transzorb	Suppressordiode
U	IC, integrated circuit	IC
U	photo couplers	Optokoppler
V	tube, valve	Röhre
V	diode, transistor	Diode, Transistor
VR	zener diode	Zener-Diode, Z-Diode
X	PTC, NTC, thermistor	PTC, NTC, Thermistor
X	crystal	Quarz, Resonator
X	connector	Anschlussklemme
XTAL, Y	crystal	Quarz, Quarzoszillator
Z	zener diode	Zener-Diode, Z-Diode

The identifiers are not always unique and some are used twice. This list is not exhaustive! You can use your own designators in your PCB layout if you think they make more sense.

Some customers may have their own specifications regarding the standards and norms to be followed. In some cases, this may also influence the naming of components.

In Germany, electrical circuit symbols are standardised by DIN EN 60617 "Graphical symbols for circuit diagrams" or IEC 60617. It replaced DIN 40700 and DIN 40900 in 1998.

84. Bill Of Material - BOM



The Bill of Materials (BOM) is a list of everything that is needed to make a product. Typically, this means every electronic component used on the printed circuit board. But this is often not enough to make a device. For example, there may be remote controls in the front panel that show up on the schematic but not on the board layout. There may also be ribbon cables or other electrical connections that are not listed anywhere in the schematic or board layout, but need to be listed in the bill of materials. If a device is to be manufactured by an external service provider, all these details are necessary.

Let's take an example of a display circuit using an NE555 timer. The corresponding bill of materials might look like this:

#	Component	Type
1	IC1	NE555
1	LED1	LED rot
2	R1,R2	47k
1	C1	10µ
1	Bat1	3V

Such a bill of materials, together with the schematic, is sufficient to reproduce a prototype. However, it is not sufficient to enable an external service provider to manufacture a product using this information, nor to produce a calculation of manufacturing costs.

For example, IC1 has been selected from the software's component library and is therefore an "ideal electronic component". However, the 555 timer component exists in many variants and designs. Among other things, they come with different supply voltages and temperature ranges. If an external service provider were to select the cheapest and most readily available 555 timer component, the "NE555" type would probably be chosen. However, what is needed in this circuit is the CMOS version "TLC555CP". This IC works from a supply voltage of 2V and is therefore well suited for battery operation.

Type	V (min-max)	Temp (min-max)
NE555	4,5-16V	0-70°C
SE555	4,5-18V	-55-125°C
TLC555CP	2-15V	0-70°C
TLC555IP	3-15V	-40-85°C

What if you want to use a fixed power supply instead of a battery? Then the TLC555IP may be sufficient. Or does the temperature range have different requirements?

This is an example of how all relevant details for component selection should be included in the bill of materials, or at least in the circuit documentation.

85. THT or SMD?

What design will the product be manufactured in? Will the circuit be partly or wholly SMD? Then a different type of 555 timer component is required.

But there are other points to consider and details to define:

- Tolerance and wattage for the resistors
- Diameter, lumen, type (diffuse) for the LED
- Tolerance, type (electrolytic capacitor), manufacturer for the capacitor
- Type identification
- Minimum order quantity
- Supplier name
- Delivery time
- Purchase order number
- Manufacturer's guaranteed availability period

A good bill of materials contains more than just the most essential information:

Quantity	Designator	Type	Design	Manufaturer	Value	MOQ	Delivery Time
1	IC1	TLC555CP	DIP	TI	-	50	1 Week
1	LED1	LED rot	3mm	Kingbright	L-934LID, low current	10	3 Days
2	R1,R2	47k	Axial	Yaego	0207, 0,6W, MF0207FTE52-47K	1000	2 Weeks
1	C1	10µ	Radial	-	-	300	Ready for Shipping
1	Bat1	3V	2R10	Camelion	Zink-Kohle, 3V, 950mAh	1	Ready for Shipping
1	Platine	-	-	-	FR4, roter Lötstopplack	6	3 Weeks
1	Gehäuse	-	-	-	PE, Batteriehalter integriert	200	8 Weeks

A bill of materials should be so complete that every part and every other component of the product, now and in the future, can be clearly identified.

86. Availability and MOQ

The delivery time is important for project planning, the minimum order quantity (MOQ) for costing and the number of units. Especially for components that can only be purchased in large quantities and with a long delivery time, it is necessary to order early - perhaps even before the project starts. Obtaining this information in advance will help to ensure that the project is well planned.

The manufacturer's full type designation for a component is particularly important for product lifecycle management. If a component is discontinued, it must be checked against the bill of materials for products still in production or their spare parts inventory.

A detailed parts list of the components used, with all the important information about the manufacturer and type, makes it clear where action needs to be taken. In this case, sufficient stock or a redesign of the product can be carried out in good time.

With the old parts list and its lead times and minimum order quantities, you can then see how difficult it might be to get a replacement.

Whether the detailed component information is in the bill of materials or in the project documents is irrelevant. What is important is that the information is properly recorded and gets to where it is needed. To the supplier, to lifecycle management, to inventory management, to quality assurance and to purchasing. Adapt the documents so that all the information you need is right at your fingertips and you do not have to search for it.

87. Design Cost Optimization

Design cost optimization refers to the process of designing a printed circuit board (PCB) with the goal of minimizing the overall production cost while maintaining the required functionality and quality. The objective is to achieve an optimal balance between performance and cost, ensuring that the PCB meets the project's budget constraints.

To achieve design cost optimization, several factors need to be considered:

1. Component Selection: Choosing cost-effective components without compromising functionality or quality is essential. It involves researching and comparing prices, considering alternative suppliers, and evaluating the cost-performance trade-offs.

2. Design Complexity: Simplifying the PCB design by reducing the number of components, layers, and intricate features can lead to cost savings. Complex designs often require more manufacturing steps, specialized equipment, and additional testing, all of which increase production costs.

3. PCB Size: Minimizing the physical dimensions of the PCB can result in material savings and reduce manufacturing costs. Smaller PCBs require less raw material and can be more cost-effective for fabrication and assembly processes.

4. Layer Stack-Up: Optimizing the layer stack-up configuration is crucial for achieving the desired electrical performance while minimizing costs. Careful consideration of the number and arrangement of signal, power, and ground layers can help avoid unnecessary complexity and reduce production expenses.

5. Manufacturing Techniques: Understanding the capabilities and limitations of various manufacturing techniques can lead to cost savings. For example, choosing standard PCB thicknesses, tolerances, and panelization

techniques that are readily available can lower production costs.

6. Design for Testability: Implementing design features that facilitate testing and inspection processes can reduce manufacturing costs. Test points, built-in self-test (BIST) circuits, and test access points (TAP) can help streamline the testing phase, identify defects, and improve production efficiency.

7. Design for Manufacturability: Designing the PCB with manufacturability in mind can minimize production costs. Considerations such as proper component placement, avoiding complex routing patterns, and adhering to design rules provided by the manufacturer can improve yields and reduce the need for rework.

8. Design Iterations and Prototyping: Iterative design cycles and prototyping allow for identifying and resolving potential issues early in the process, reducing the risk of costly modifications during mass production.

9. Supplier Collaboration: Engaging in effective communication and collaboration with PCB manufacturers and component suppliers can result in cost savings. Working closely with suppliers to optimize procurement strategies, negotiate prices, and explore alternative options can help reduce overall costs.

10. Design Validation and Simulation: Utilizing simulation tools and techniques can identify potential design flaws and optimize performance before production. This helps avoid costly design changes and ensures that the PCB meets the required specifications.

By considering these aspects and adopting a systematic approach, designers can effectively optimize the cost of their PCB designs without compromising quality or performance, ultimately resulting in more economical and competitive products.

Design for Environmental Sustainability

Design for environmental sustainability, also known as eco-design or green design, involves integrating principles and practices that minimize the environmental impact of a printed circuit board (PCB) throughout its lifecycle. The objective is to create PCB designs that are environmentally friendly, resource-efficient, and contribute to a more sustainable future. A few of the points have already been addressed in more detail elsewhere in the book, but here I will summarise all the thoughts and key words on the subject:

1. Material Selection: Choosing environmentally friendly and recyclable materials for the PCB, such as lead-free solder and RoHS-compliant components, reduces the use of hazardous substances and facilitates end-of-life recycling.
2. Energy Efficiency: Designing the PCB to minimize power consumption can contribute to energy savings during the product's use phase. This includes optimizing power distribution networks, reducing parasitic power losses, and utilizing low-power components when possible.
3. Waste Reduction: Reducing waste generation during PCB manufacturing and assembly is crucial. Designing for panelization and efficient use of materials, minimizing scrap, and promoting responsible disposal of waste are essential considerations.
4. Design for Disassembly: Facilitating easy disassembly and separation of PCB components at the end of the product's life enables efficient recycling and reuse. Using modular designs and standardized interfaces allows for selective replacement and upgrade of components.
5. Life Cycle Assessment (LCA): Conducting a life cycle assessment helps evaluate the environmental impact of the PCB design from raw material extraction to disposal. It allows for identifying hotspots and implementing design strategies to minimize overall environmental burdens.

6. Environmental Compliance: Ensuring compliance with environmental regulations and standards, such as RoHS and REACH, is crucial. Designing with these standards in mind helps avoid legal and environmental liabilities.

7. End-of-Life Considerations: Planning for the end-of-life phase of the PCB involves considering recycling options, promoting circular economy principles, and designing for extended product lifespan through upgradability and repairability.

8. Supply Chain Management: Collaborating with suppliers who prioritize sustainability, adhere to environmental standards, and offer environmentally friendly products and services contributes to the overall sustainability of the PCB.

9. Energy Harvesting and Renewable Energy Integration: Exploring opportunities to incorporate energy harvesting techniques, such as solar or kinetic energy, into the PCB design can enhance its energy efficiency and reduce reliance on non-renewable energy sources.

10. Environmental Communication: Providing clear and accurate information about the environmental impact and sustainability features of the PCB design to stakeholders, including consumers, promotes transparency and awareness.

By incorporating these principles into PCB design, designers can minimize resource consumption, reduce waste generation, and contribute to a more sustainable and environmentally responsible electronic industry. Designing for environmental sustainability not only benefits the planet but also enhances the long-term viability and competitiveness of products in the market.

88. EMC and EMI

Design for Electromagnetic Compatibility (EMC) refers to the practice of designing a printed circuit board in a way that minimizes electromagnetic interference (EMI) and ensures that the circuit can operate reliably in the presence of electromagnetic fields. The individual points have already been addressed in more detail elsewhere in the book, but here I summarise them once again in key words:

1. Grounding and Shielding: Proper grounding techniques and the use of shielding materials can prevent the coupling of unwanted electromagnetic energy and minimize the emission and reception of electromagnetic radiation.
2. Signal Integrity: Maintaining good signal integrity is crucial to reducing EMI. Properly designed transmission lines, controlled impedance, and minimizing signal reflections help mitigate the effects of noise and interference.
3. Component Placement: Thoughtful component placement can minimize EMI by reducing the lengths of high-speed signal traces and minimizing the coupling of signals with sensitive components.
4. Power Integrity: Ensuring stable and clean power distribution on the PCB reduces the potential for voltage fluctuations and noise, which can lead to EMI. Proper decoupling and bypassing techniques are essential for maintaining power integrity.
5. Routing and Trace Separation: Careful routing practices, such as maintaining appropriate trace separation distances, can reduce crosstalk and electromagnetic coupling between signal traces.
6. Filtering and ESD Protection: Implementing filters and transient voltage suppressors (TVS) at key points can help mitigate electromagnetic interference and protect components from electrostatic discharge (ESD).

7. EMI Shielding: Incorporating EMI shielding measures, such as metal enclosures, conductive gaskets, or shielding cans, can prevent electromagnetic radiation from escaping the PCB or entering sensitive components.

8. Ground Plane Design: Proper design of ground planes helps minimize ground loops, improve signal return paths, and reduce EMI. Splitting ground planes and providing dedicated ground planes for sensitive components can be effective strategies.

9. PCB Stack-up: Optimizing the layer stack-up configuration can improve EMC performance. Placing power and ground planes strategically within the stack-up helps control electromagnetic fields and reduces EMI.

10. Compliance with Standards: Ensuring compliance with EMC standards and regulations specific to the target market or industry is critical. Compliance testing and certification validate the PCB's EMC performance and its ability to operate in electromagnetic environments.

11. EMC Simulation and Testing: Utilizing electromagnetic simulation tools and conducting EMC tests during the design process can identify potential issues early on and allow for necessary design adjustments to meet EMC requirements.

Designing for EMC is crucial to ensure reliable operation and minimize the interference of electronic devices with other devices or systems. By incorporating these considerations into PCB design, designers can reduce EMI, enhance electromagnetic compatibility, and improve the overall performance and reliability of the PCB in its intended operating environment. If you have a chance, take a look at these industry standards for a more in-depth look:

ECSS-E-ST-20-07C
Electromagnetic Compatibility

ECSS-E-ST-20C
Electrical and Electronic

IEEE Std 1100-2005
Recommended Practice for Powering and Grounding Electronic Equipment

IEEE Std 1138-1994
Guide on EMI Evaluation of Human Exposure to Magnetic Fields

89. ESD and TVS

ESD (Electrostatic Discharge) and TVS (Transient Voltage Suppressor) are measures used to protect electronic circuits from damage caused by electrostatic discharge events or transient voltage spikes. A few of the points have already been addressed in more detail elsewhere in the book, but here I will summarise all the thoughts and key words on the subject:

1. ESD is the sudden flow of static electricity between two objects, potentially damaging electronic components. It can occur during handling, manufacturing, or through environmental factors.
2. ESD Protection Components: ESD protection components, such as diodes or varistors, are designed to divert and dissipate electrostatic charges safely, preventing them from damaging sensitive circuitry.
3. ESD Protection Strategies: Implementing ESD protection involves adding ESD diodes to input/output lines, grounding the PCB, using ESD-safe handling procedures, and incorporating shielding techniques.
4. TVS: TVS devices provide protection against transient voltage spikes, such as those caused by power surges, lightning strikes, or electromagnetic interference. They absorb and redirect high voltage transients away from the protected circuit.
5. TVS Diodes: TVS diodes are commonly used TVS devices. They have a fast response time and can handle high transient currents, safeguarding the circuit by clamping excessive voltage levels.
6. TVS Selection: Choosing the appropriate TVS device involves considering factors such as voltage rating, clamping voltage, response time, and energy-handling capability to match the specific protection requirements of the circuit.

7. Placement: ESD and TVS protection components should be strategically placed near vulnerable circuit nodes to ensure effective protection. Placing them close to connectors, sensitive ICs, or inputs/outputs is essential.

8. PCB Layout Considerations: Proper PCB layout practices, such as minimizing trace lengths, ensuring proper grounding, and separating sensitive signals from noisy sources, aid in reducing the susceptibility to ESD and transient voltage spikes.

9. Compliance with Standards: Following industry standards and guidelines for ESD and TVS protection, such as IEC 61000-4-2 for ESD and IEC 61000-4-5 for surge protection, helps ensure adequate circuit protection.

10. Testing and Verification: ESD and TVS protection should be tested and verified using appropriate test equipment and methods to ensure their effectiveness under real-world conditions.

11. Education and Training: Educating personnel about ESD risks, proper handling procedures, and ESD control measures is crucial to minimize the occurrence of ESD-related failures.

By implementing ESD protection measures such as ESD diodes and proper grounding techniques, together with TVS devices to suppress transient voltage spikes, circuits can be protected from potential damage. The selection, placement and proper integration of ESD and TVS components, together with compliance with relevant standards, will contribute to reliable circuit protection against ESD and transient voltage events. The design details of how such protection needs to be implemented in the circuit requires an electrical engineer or PCB designer familiar with the specific circuit design. The datasheets for these components usually provide examples of how these parts must be incorporated into the design to provide effective protection.

If you have a chance, take a look at these industry standards for a more in-depth look:

ANSI/ESD S20.20

Guidelines for the development and implementation of an Electrostatic Discharge Control

IEC 61000-4-5

Surge immunity testing for electrical and electronic equipment against transient voltage spikes

JEDEC JESD625

Characterization and testing of TVS devices used for ESD protection

ESDA TR53

Guidance on the selection and application of TVS diodes for ESD protection

90. Reduce Design Complexity

Design complexity in the context of printed circuit board (PCB) design refers to the level of intricacy and sophistication involve The individual points have already been addressed in more detail elsewhere in the book, but here I summarise them once again in key words:

1. Component Density: Higher component density increases design complexity, as it requires careful placement and routing to avoid signal interference and ensure accessibility for assembly and testing.
2. Number of Components: The more components a PCB design contains, the greater the complexity. Managing a large number of components necessitates efficient organization, clear labeling, and optimized routing.
3. Signal Integrity: Designs with high-speed signals or complex signal paths introduce additional complexity. Maintaining signal integrity through controlled impedance, proper routing techniques, and minimizing noise sources is crucial.
4. Layer Count: Designs with multiple layers introduce complexity due to the need for accurate alignment, impedance control, and proper interconnectivity between layers.
5. Constraints and Rules: Incorporating design constraints and adhering to design rules add complexity but ensure that the PCB meets the required specifications and constraints.
6. Power Distribution: Complex power distribution networks require careful consideration to minimize voltage drops, optimize power delivery, and prevent noise coupling.
7. Manufacturing Challenges: More complex designs often pose manufacturing challenges, such as tighter tolerances, specialized materials, and intricate assembly requirements.
8. Testing and Debugging: Complex designs can be more challenging to test and debug. Accessibility to test points,

adequate visibility, and effective debugging techniques are essential considerations.

9. Design Iterations: Complex designs may require multiple iterations and revisions to achieve the desired performance and functionality, increasing the design timeline and effort.

10. Documentation: Comprehensive documentation becomes more crucial with increasing complexity. Clear schematics, detailed component placements, and thorough design notes aid in understanding and future modifications.

11. Design for Manufacturability: Complex designs must consider manufacturability aspects, such as panelization, assembly constraints, and proper documentation for fabricators.

12. Design Team Collaboration: Complex designs often involve collaboration among a team of engineers, further necessitating effective communication, version control, and coordination.

13. Project Management: Complex designs require efficient project management to track progress, manage resources, and meet project milestones within the specified timeframe.

14. Design Validation: Validating complex designs through simulation, prototyping, and testing helps identify and resolve potential issues early in the design process.

15. Cost Considerations: Complex designs often incur higher manufacturing and assembly costs due to additional materials, specialized processes, and increased complexity of testing and inspection.

Managing design complexity requires a systematic and methodical approach to ensure a successful PCB design. Balancing functionality, manufacturability, testing, and cost considerations while maintaining signal integrity is key to addressing the challenges associated with complex designs and achieving a robust and reliable PCB layout.

91. Resilience

Resilience in PCB design refers to the ability of a printed circuit board to withstand and recover from various stresses, disruptions, or failures while maintaining its intended functionality. Here are key aspects to consider regarding resilience in PCB design:

1. Component Selection: Choosing high-quality and reliable components is essential for building a resilient PCB. Components with proven track records, extended operating temperature ranges, and robust construction contribute to the overall resilience of the design.
2. Redundancy: Incorporating redundancy in critical components or subsystems can enhance resilience. For example, using redundant power supplies or redundant communication paths can ensure continued operation even if one component or path fails.
3. Robust Design Practices: Employing robust design techniques, such as proper decoupling, signal integrity considerations, and thermal management, helps improve the resilience of the PCB. By addressing potential issues and ensuring stable operation under varying conditions, the PCB becomes more resilient.
4. Environmental Considerations: Designing the PCB to withstand environmental factors, such as temperature variations, humidity, vibration, and shock, is crucial for resilience. Selecting appropriate materials, conformal coatings, and protective enclosures can enhance the board's resistance to environmental stresses.
5. Testing and Validation: Rigorous testing and validation processes are vital to ensure the resilience of the PCB. This includes functional testing, stress testing, environmental testing, and simulation to identify potential weak points and verify the board's performance under various conditions.
6. Fault Detection and Fault Tolerance: Implementing fault detection mechanisms, such as built-in test circuits or

diagnostic features, allows for the detection and identification of faults or failures. Additionally, incorporating fault-tolerant design techniques, like error correction codes or redundant circuits, enables the PCB to continue functioning even in the presence of faults.

7. Repairability and Serviceability: Designing the PCB with repairability and serviceability in mind enhances its resilience. This includes features such as test points, accessibility of components, and clear labeling, making it easier to identify and address issues during maintenance or repairs.

8. Documentation and Traceability: Maintaining comprehensive documentation, including schematics, BOMs, and design notes, aids in troubleshooting and resolving issues, thus improving the resilience of the PCB over its lifecycle.

9. Compliance with Standards: Adhering to relevant industry standards and regulations ensures that the PCB meets specific resilience requirements, such as those related to electromagnetic compatibility (EMC), safety, and reliability.

By considering these factors during the PCB design process, engineers can create more resilient boards that can withstand adverse conditions, recover from failures, and maintain reliable operation over time. Resilient PCB designs are crucial in applications where reliability, durability, and uninterrupted performance are critical. If you have a chance, take a look at the aerospace industry standards for a more in-depth look:

ECSS-Q-ST-70-38C

High-reliability soldering for surface-mount and mixed technology

ECSS-Q-ST-70-08C

Manual soldering of high-reliability electrical connections

ECSS-E-ST-20C
Electrical and Electronic
ESA PSS-01-609
Radiation Design Handbook

92. Standards and Guidelines

Printed circuit board manufacturing is complex and difficult. So is the development of printed circuit boards. The machines and operations must be extremely precise and perfectly coordinated. Components are now very small, the smallest form of an SMD component is only 0.2 mm wide and 0.4 mm long. An assembly machine can place such a small SMD on the board in 0.09 seconds. There can be hundreds of components on a board. Before a PCB is assembled, many processing steps are required to make the PCB. After the board is assembled, the components must be soldered and tested. All of this is done automatically and largely without human intervention.

The automation of such highly complex processes requires many standards and guidelines to ensure that a PCB designed in Europe can also be manufactured in China. Cultural and linguistic differences must not play a role. That is what these standards and guidelines are for. But as you will see, it is not all as simple as you might think.

The most important document concerning PCB design is IPC-2221, the "Generic Standard on Printed Board Design". This 1998 guideline replaced the old IPC-D-275 (also known as Military Standard 275), which was published in 1991.

IPC-7351, the "Basic Requirements for an SMD Surface Mount Design and Land Pattern Standard", was adopted in 2005 and provides information on package design.

Although local guidelines exist in various countries, the IPC standards are by and large the globally accepted industry standards.

Here's a tip: In many places in this book I refer to the contents of various standards. Sometimes they are European and sometimes they are American. Sometimes they contradict or differ from each other. I have not tried to reconcile them, nor have I tried to judge which guideline is correct. Personally, I always start with the most conservative standard and the one that promises more safety.

The more you get involved with electronics and PCB design, the more you will come across such discrepancies. You will often find confusing specifications and the answer to many questions is "it depends". Then ask how it was done before. Or ask your customer if they have a guideline they want to follow.

93. Select an EDA Software

Choosing your EDA software is a long-term decision. A very long-term decision!

Every software needs a long period of time until you have familiarised yourself with it and can also handle it efficiently. This also includes knowing the limits of the programme so that you can identify a design that cannot be realised with the programme in advance. Depending on the complexity of the PCBs you develop, you may need one or the other feature in your EDA software. While some tasks can still be done by external tools (e.g. calculating the necessary track width), differential pair routing cannot be outsourced. Other features such as pin swapping can be done manually for smaller components, even if it takes a little more time.

On the one hand, EDA software can try to control too many parameters in your design, which can be annoying if you are only creating a very simple layout. On the other hand, it can give too little guidance for complex designs, which could result in a faulty layout.

Finding the right balance between the capabilities of your EDA software and the complexity of the PCBs you are designing will help you work efficiently and quickly. But features are not everything. Usability is much more important!

You will get used to "your" software over the years and it will be all the more difficult to change.

Ideally, if you are not a hobbyist or amateur, you should familiarise yourself with two EDA software products. This way you can compensate for sudden difficulties in one software and work around problems. PCBs of varying complexity are then developed using the more suitable EDA software. Over time, as your board requirements increase, you can also change your EDA software.

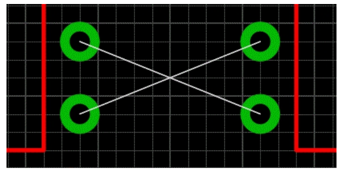
94. Future Proof Software

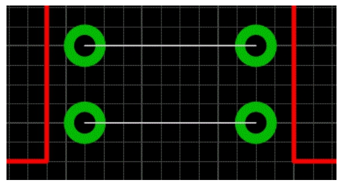
The EDA software you use must be future-proof. If you have developed products that need to be revised in five or ten years' time, for example because a component has been discontinued, you need to be able to edit the old files. And you need to be able to transfer the updated design to the PCB manufacturer, assuming they still accept the file format. Sometimes you are contractually bound by your customers to provide support for such a long period. It is then necessary to archive the data together with the EDA software in the appropriate version.

Think back ten years and compare the software you had then with what you have now. Now think about the EDA software you are considering purchasing. Would licensing and activation still work ten years from now? How does activation work? Is the software locked to a computer? What happens if the vendor or product is no longer on the market? Do you have a copy of your licence that can be reactivated without external dependencies? Will you always be able to reinstall the software in the future when you buy a new workstation? How quickly can the software be transferred to the new workstation and activated? Will it all work on a Friday afternoon when the hotline is unavailable?

A good test is to reinstall the EDA software on a freshly installed computer without an internet connection. Does it work, or is the software dependent on some licensing servers? Will these servers still exist in the future?

95. Pin Swapping



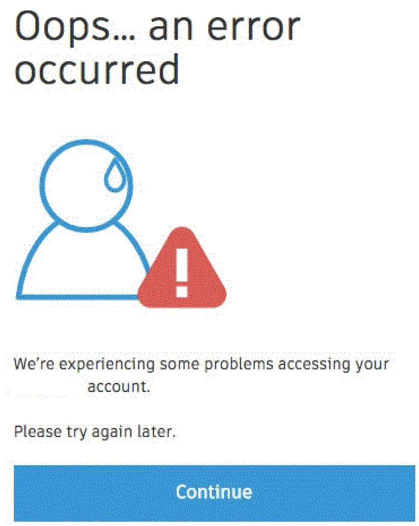


Such unwanted crossovers occur when the schematic is transferred to the board layout. In the schematic you only see an AND gate, its position in the IC is not important. However, when the components are placed on the board, the position of the gate in the IC has a major influence on the routing. For example, if the traces leading to the two inputs of an AND gate cross, pin swapping can be used to swap the connections.

For devices with several hundred pins, such as an FPGA, automatic determination of the best-placed pins greatly simplifies routing and saves time.

96. Activation and Licensing Problems

A real-life example:



Imagine you need to make a correction to a PCB layout just before a deadline. So you sit down at your computer and launch your EDA software. The next thing you know, you see an error message with a sad little man and the programme won't start. Such a direct dependency, tied to internet access every time the programme starts, can cause you problems in some circumstances.

Sometimes you are contractually bound to secrecy by your client. In this case, you need to be particularly careful that the circuit and design data you develop for your customer does not fall into the hands of third parties. This means that cloud-based EDA software is out of the question unless you can ensure that the data is stored locally. Check the EULA of the cloud you are using.

In some environments with heightened security regulations, only offline operation, i.e. without internet access, may be possible. In some countries, such as China, it is impossible to access the EDA manufacturer's servers. So if you need to be on-site for a development job, you may encounter fundamental problems and be unable to use your EDA software.

So choose your EDA software wisely! Be aware of how dependent you will become on this software.

97. Hardware Configuration

The PCB designer's work is carried out on a PC of the appropriate performance class, as specified by the software manufacturer as the minimum requirement. In general, no excessive demands are made on the PC, but by paying attention to a few points, you can make your work more pleasant.

If possible, use a CAD graphics card. This has a similar hardware architecture to powerful gaming cards. However, they use different drivers. Suitable CAD graphics cards include the NVIDIA Quadro, AMD FirePro or AMD Radeon Pro series. This is not absolutely necessary, and often a powerful graphics card is all that is required for current games. This can have a positive effect on complex layouts. Check whether the EDA software you are using uses hardware acceleration ("OpenGL") and which graphics card is suitable for it. Often the manufacturer will give recommendations for the graphics card. EDA software that does not flicker when zooming and panning is easy on the eyes.

A large screen is advantageous for PCB layout. It allows you to place the schematic and the PCB view side by side, which supports a fluid workflow. A monitor with a resolution of 2560x1440 or 2560x1600 pixels and a size of 27" to 30" is best. The trend is towards so-called 4K monitors. Their resolution and size are very suitable. But make sure that your EDA software supports the so-called "high resolution" display.

It is generally recommended to use two monitors for PCB design. This way you can view the schematic and the PCB in parallel and compare them directly ("cross-probe").

Here's a tip: If your EDA software is a 32-bit program, it can only use a maximum of 2 GB of memory per process. If you are using 64-bit Windows, the LAA (Large Address

Aware) flag can be set, allowing the 32-bit program to use more than 2 GB (maximum 4 GB) of available memory. So even if you have 8GB or more of RAM installed in your computer, it will not be used by the 32-bit EDA software. Of course, the rest of the system can still benefit.

98. Acquisition Costs of EDA Software

You can spend a lot of money when you buy EDA software. Once you start making money from your service, the PCB layout software you use becomes a working tool. Perhaps even your main tool. The time you spend creating a circuit and PCB layout will be a significant part of your overall effort. Cheaper is not always better. If an EDA software package has integrated tools that help you avoid errors, or if you can develop twice as fast as with a free (or cheaper) product, then the higher price will quickly pay for itself.

Test the speed and handling of the software you have shortlisted before committing to it.

Some software comes with a one-off purchase price, while others require an annual subscription fee for continued use. Make sure you understand the ongoing costs and whether you can continue to use the software after the ongoing support and licensing costs have expired.

99. EDA - Number of Features

The range of features required in EDA software, the "feature list", is based solely on what you need in PCB design. Start small and try a handful of products to get a feel for it. It is very unlikely that you will find the right software with the first one you try. Take your time and run the same project with different EDA packages. You will notice differences and end up with a better idea of what each offers. You may even find your favourite.

Too many features and you can't use the software because you don't know what to specify in the many dialogues.

Too few features and you will make very slow progress with your design because you will be dependent on external tools and will have to manually solve some problems and tricky requirements when routing the tracks.

If the software is very confusing and difficult to use, you may never get to grips with it and just end up cursing and in a bad mood as you listlessly click a design together.

So what do you need? It depends on your experience and your needs. Are you someone who spends eight hours a day designing PCBs in EDA software? Or do you do it once or twice a month? Or once or twice a year? How complex are the PCBs you need to design?

You are also likely to evolve and your software requirements will change. This can happen especially in the early days. Then look at whether your EDA software can be extended through different levels of expansion.

100. Feature Scope of an EDA Software

Hobby use:

EDA software for the maker and hobbyist should first and foremost be intuitive to use. This user group only designs a few boards per year and should be able to implement their ideas easily. Overly complex software would only result in wasted time trying to remember the implications, possibilities and intricacies of the settings dialogues. The complexity of the boards being developed is low to moderate.

There are now so-called maker editions and "non-profit" versions of many EDA software products. These are less expensive than the commercial versions. You can save a lot of money with a comparatively large feature set, but you may have to live with the restriction that you are not allowed to develop commercial products with it.

If the autorouter in such a version costs extra money, just leave it out. Its usefulness is too limited to contribute anything at that level. Better practise your routing skills.

Semi-professional use:

Semi-professional EDA software, for ambitious users who do a lot of PCB design, should be good value for money and future-proof. The complexity of PCBs can be demanding. All the features should be included in the software and allow for continuous work. If the full-featured version is still too expensive, the software should be able to grow with your needs by only upgrading to the next level when needed.

However, there are a number of features that are very interesting for this application area:

- Forward and backward annotation, so that if you change the connections in the PCB layout, this is automatically reflected in the schematic - and vice versa.
- Project management that includes a change log, a to-do list and the ability to archive version statuses.

- Good and fast support, ideally speaking the same language and offering training where necessary.
- The software should include circuit simulation, a usable autorouter, differential pair routing, delay lines (meandering) and pin/pad swapping.

Professional use:

The 'big players' in EDA design environments for large companies with demanding PCB design are OrCAD, Allegro, Altium-Designer, Pulsonix, PADS, Mentor Graphics and Cadstar. These programmes have ongoing costs if you want to receive software updates and support. Getting to grips with and using such extensive software can only be done effectively by those who make it their day job and spend many hours working with it every day. When such software is the basis of the business, and even the most complex designs have to be handled, it is important not to save money at the wrong end. To really master design software of this magnitude, you need to have spent several hundred hours on it and completed several projects with it. Without training, you can only click on the surface and not exploit the possibilities.

101. Finally

This is now a collection of PCB layout tips and tricks. There is much more to the subject, you never stop learning, but I hope you can find some good points to improve your PCB design with the content listed here.

To think outside the box a bit, some aspects have been covered that are not necessary for drawing the traces, but can be part of a good design.

As you will have noticed, not all guidelines and specifications are set in stone and rarely are they unambiguous. Some old guidelines, although already obsolete, are still actively used. Many standards are also frequently revised to keep up with the latest technological developments. So try to keep up with developments in your field of work and check regularly to see if the standards have been revised.

Good luck!

102. Terms and Abbreviations

EDA	Electronic Design Automation
CAD	Computer Aided Design
ECAD	EDA
CAE	Computer Aided Engineering
CAM	Computer Aided Manufacturing
CIM	Computer Integrated Manufacturing
BOM	Bill Of Materials
THT	Trough Hole Technology
SMD	Surface Mount Device
PDM	Product Data Management
Layout-Software	CAD programme (EDA) for creating printed circuit boards
PCB-Design	Refers to the placement of components and the creation of tracks and related aspects, e.g. the display and operating elements.
PCB-Layout	Refers to the placement of components and the application of conductive tracks.
Circuit Board	PCB without Components
Panel	Used in production. Consists of several, not separated, PCBs and has an edge length of approx. 60 cm.
Breadboard	Circuit board for hobby purposes and prototypes
High-Speed-Design	Printed circuit boards with a circuit whose clock is in the GHz range.
EMV-Design	The design of a high-speed PCB with attention to electromagnetic compatibility.
Floorplan	Arrangement of the components on the PCB according to a special grouping.
Autorouter	Is supposed to lay the tracks correctly at the push of a button.

DFT	Design for Test
DFM	Design for Manufacturing
AWG	American Wire Gauge
BGA	Ball Grid Array
CLCC	Ceramic Leaded Chip Carrier
CNC	Computer Numeric Controlled
DIN	German Institute for Standardisation. The national standards organisation.
EDA	Electronics Design Automation
EIA	Electronics Industries Automation
EN	European Standard
ESD	Electrostatic Discharge
IC	Integrated Circuit
IEC	International Engineering Consortium.
ISO	International Standardisation Organization.
PGA	IC-Package
PLCC	IC-Package
ppm	Parts per million
SO-	IC-Package
SOJ-	IC-Package
VDE	Electrical Engineering Association. A technical-scientific association.
EMC	Electromagnetic Compatibility
EMI	Electromagnetic Interference
EMV	Elektromagnetische Compatibility
Crosstalk Coupling	Crosstalk of signals to one or more other tracks.
MOQ	Minimum Odered Quantity
IPC	International trade association for the printed-board and electronics assembly industries
forward/back	Changes in the circuit diagram or in the

-annotation	board view are updated in the entire project.
Tin Whiskers	Dendrites made of tin. Leads to short circuits.
Pitch	Centre-to-centre distance of the component connections. In other words, the grid dimension of the components.

103. EDA-Software Links

Free/Open source

CircuitMaker (Altium) - http://www.circuitmaker.com/
DesignSpark (RS comp) - http://www.designspark.com/
EasyEDA - http://easyEDA.com/editor (web based)
FreePCB - http://www.freepcb.com/
Fritzing - http://fritzing.org/
gEDA - http://www.geda-project.org/
KiCAD - http://kicad-pcb.org/
Minimal Board Editor - http://www.suigyodo.com/online/e/index.htm (English, Japanese)
Open Circuit Design - http://opencircuitdesign.com/
Orcad DOS (old) - http://tech.groups.yahoo.com/group/OldDosOrcad/
PCB Elegance - http://www.pcbelegance.com/
PCBWeb Designer - http://www.pcbweb.com/ (Windows only)
Protel (Altium) AutoTrax/EasyTrax - http://www.altium.com/community/downloads/en/downloads_home.cfm
TCI - http://b.urbani.free.fr/
ZenitPCB - http://www.zenitpcb.com/Index.html

Limited free version

CADint - http://www.cadint.se/p_free.asp
Cadstar Express (Zuken) - http://www.zuken.com/products/cadstar/downloads/express.aspx
CometCAD - http://www.cometcad.com/
DipTrace - http://www.diptrace.com/
Eagle (Autodesk) - http://www.autodesk.com/
Layo1 PCB - http://www.baas.nl/layo1pcb/uk/downloads.htm
McCAD - http://www.mccad.com/index.html

OrCad (Cadence) - http://www.cadence.com/products/orcad/pages/downloads.aspx#demo

Osmond PCB - http://www.osmondpcb.com/index.html (Only PCB)

TopoR - http://eda.eremex.com/

WinQcad - http://www.winqcad.com/

Commercial

Allegro (Cadence) - http://www.cadence.com/products/pcb/Pages/default.aspx

Altium - http://www.altium.com/

Ariadne - http://www.cad-ul.de/ariadne-eda.html (German)

AutoTRAX DEX 2020 - https://kov.com/

Bartels AutoEngineer - http://www.bartels.de/bae/bae_en.htm (English, German)

BoardMaker3 - http://www.tsien.info/index.php#

Board Station (Mentor) - http://www.mentor.com/products/pcb-system-design/design-flows/boardstation/

CADint - http://www.cadint.se/products.asp

Cadstar (Zuken) - http://www.zuken.com/products/cadstar

CIRCAD - http://www.holophase.com/index.html

CircuitCREATOR - http://www.circuitcreator.com/creator/index.htm

Circuit Studio (Altium) - http://www.circuitstudio.com/

Circuit Wizard, PCB wizard - http://www.new-wave-concepts.com/products.html

CometCAD - http://www.cometcad.com/

CSiEDA - http://www.csitek.co.kr/ (Korean)

CSiEDA - http://www.csieda.co.jp/en/csieda/ (Japanese, English)

DipTrace - http://www.diptrace.com/

Douglas CAD/CAM - http://www.douglas.com/software/pro/prolayout.html

Dreamcad - http://www13.ocn.ne.jp/~dreamnet/ (Japanese)
Eagle (Autodesk) - http://www.autodesk.com/products/eagle/overview
Easy-PC - http://www.numberone.com/easypc.asp
EDWinXP - http://www.visionics.a.se/index.html
Expedition Enterprise (Mentor) - http://www.mentor.com/
ICADPCB (Fujitsu) - http://www.fujitsu.com/my/services/software/business/icad_thirdpage.html
Layo1 PCB - http://www.baas.nl/layo1pcb/uk/index.html
McCAD - http://www.mccad.com/index.html
Orcad (Cadence) - http://www.orcad.com/
Osmond PCB - http://www.osmondpcb.com/index.html (Only PCB)
Pads (Mentor) - http://www.mentor.com/products/pcb-system-design/design-flows/pads/
Pantheon PCB software (Intercept) - http://www.intercept.com/products/pantheon-pcb-design
Proteus PCB Design - http://www.labcenter.com/products/pcb_overview.cfm
Pulsonix - http://pulsonix.com/index.asp
Quadcept - https://www.quadcept.com/en/index.html
Rimu Schematics/PCB - http://www.hutson.co.nz/
Scooter PCB - http://www.scooter-pcb.de/scooter-pcb/index.html (German)
Sprint-Layout - http://www.abacom-online.de/uk/html/sprint-layout.html
SuperCAD/SuperPCB - http://www.mentala.com/products.htm
Target 3001 - http://www.ibfriedrich.com/ (English, German, French)
TINA (DesignSoft) - http://www.tina.com
TopoR - http://eda.eremex.com/
Ultiboard (National Instruments) - http://www.ni.com/ultiboard/

Upverter - https://upverter.com/ (free for open source projects)
Vutrax - http://www.vutrax.co.uk/index.htm
WinCircuit - http://alain.michel13.free.fr/Anglais.html
WinQcad - http://www.winqcad.com/
XL designer (Seetrax) - http://www.xldesigner.com/
Zuken - http://www.zuken.com/en

Free SW for PCB order service (no gerber files export)

ECAD Pro - http://www.pcbdesignandfab.com/ (gerber files $25/board)
Expresspcb - http://www.expresspcb.com/
Pad 2 Pad - http://www.pad2pad.com
PCB123 - http://www.sunstone.com/PCB123-CAD-Software.aspx (gerber files $100/board)
PCB Artist - http://www.4pcb.com/free-pcb-layout-software/

Schematic Capture

sPlan - http://www.abacom-online.de/uk/html/splan.html
TinyCAD - http://sourceforge.net/apps/mediawiki/tinycad/index.php?title=TinyCAD
XCircuit - http://opencircuitdesign.com/xcircuit/

Gerber tools

3D Gerber Viewer (online) - http://mayhewlabs.com/3dpcb
Circuit people online Gerber viewer - http://circuitpeople.com/
DFM Now! - http://www.numericalinnovations.com/pages/dfm-now-free-gerber-viewer
GCPrevue (Graphicode) - http://www.graphicode.com/Download_GC-Prevue_and_Free_Trials

GerberLogix - http://www.easylogix.de/products_detail.php?prog_id=1
gerbv - http://gerbv.sourceforge.net/
McCAD - http://www.mccad.com/FREE_GerberViewer.html
Online Gerber-Viewer - http://www.gerber-viewer.com/
Tracespace - http://viewer.tracespace.io/ (online viewer)
ViewMate - http://www.pentalogix.com/viewmate.php (Free & Commercial versions)
Viewplot - http://www.viewplot.com/
ZofzPCB 3D Gerber Viewer - http://www.zofzpcb.com/

PCB panelizing
CAM 350 - http://www.downstreamtech.com/CAM350XL.html
FAB 3000 - http://www.numericalinnovations.com/fab3000.html

Stripboard/Veroboard layout tools
DIY Layout Creator - http://diy-fever.com/software/
LochMaster - http://www.abacom-online.de/uk/html/lochmaster.html
Stripboard Magic - http://www.marlwifi.org.nz/other/stripboard-magic
Stripes - http://sites.google.com/site/libby8dev/stripes
VeeCAD - http://veecad.com/
VeroCAD - http://xtronic.org/download/verocad-3-veroboard/
VeroDes - http://www.heyrick.co.uk/software/verodes/

Simulation - Free
5Spice - http://www.5spice.com/ (not free for commercial use)
Cider - http://embedded.eecs.berkeley.edu/pubs/downloads/cider/index.htm
CircuitLab.com - https://www.circuitlab.com/ (Online only)
Logisim - http://ozark.hendrix.edu/~burch/logisim/

LTpsiceIV (Linear Technology) - http://www.linear.com/designtools/software/
Ngspice - http://ngspice.sourceforge.net/screens.html
Qucs - http://qucs.sourceforge.net/
SimOne - http://eda.eremex.com/ (limited)
TINA-TI (Texas Instruments version) - http://www.ti.com/tool/tina-ti
XSPICE - http://users.ece.gatech.edu/~mrichard/Xspice/

Simulation - Commercial

Advanced Design System, ADS (Keysight) - http://www.keysight.com/
AIM Spice - http://www.aimspice.com/
B2 Spice (Beige Bag) - http://www.beigebag.com/
Circuit Wizard - http://www.new-wave-concepts.com/products.html
ICAP/4, etc. - http://www.intusoft.com/default.htm (Limited Free version available)
Micro Cap - http://www.spectrum-soft.com/index.shtm (Limited Free version available)
NI Multisim (National Instruments) - http://www.ni.com/multisim/
NL5 Circuit Simulator - http://nl5.sidelinesoft.com/
Plecs - http://www.plexim.com/
PSpice (Cadence) - http://www.cadence.com/ (Limited Free version available)
SIMetrix - http://www.simetrix.co.uk/ (demo available)
Super Spice (Anasoft) - http://www.anasoft.co.uk/ (Limited Free version available)
TINA - http://www.tina.com/English/tina/start.php
TopSpice - http://penzar.com/topspice/topspice.htm (Limited Free version available)
Via Designer - http://www.viadesigner.com/
WinSpice - http://www.winspice.com/