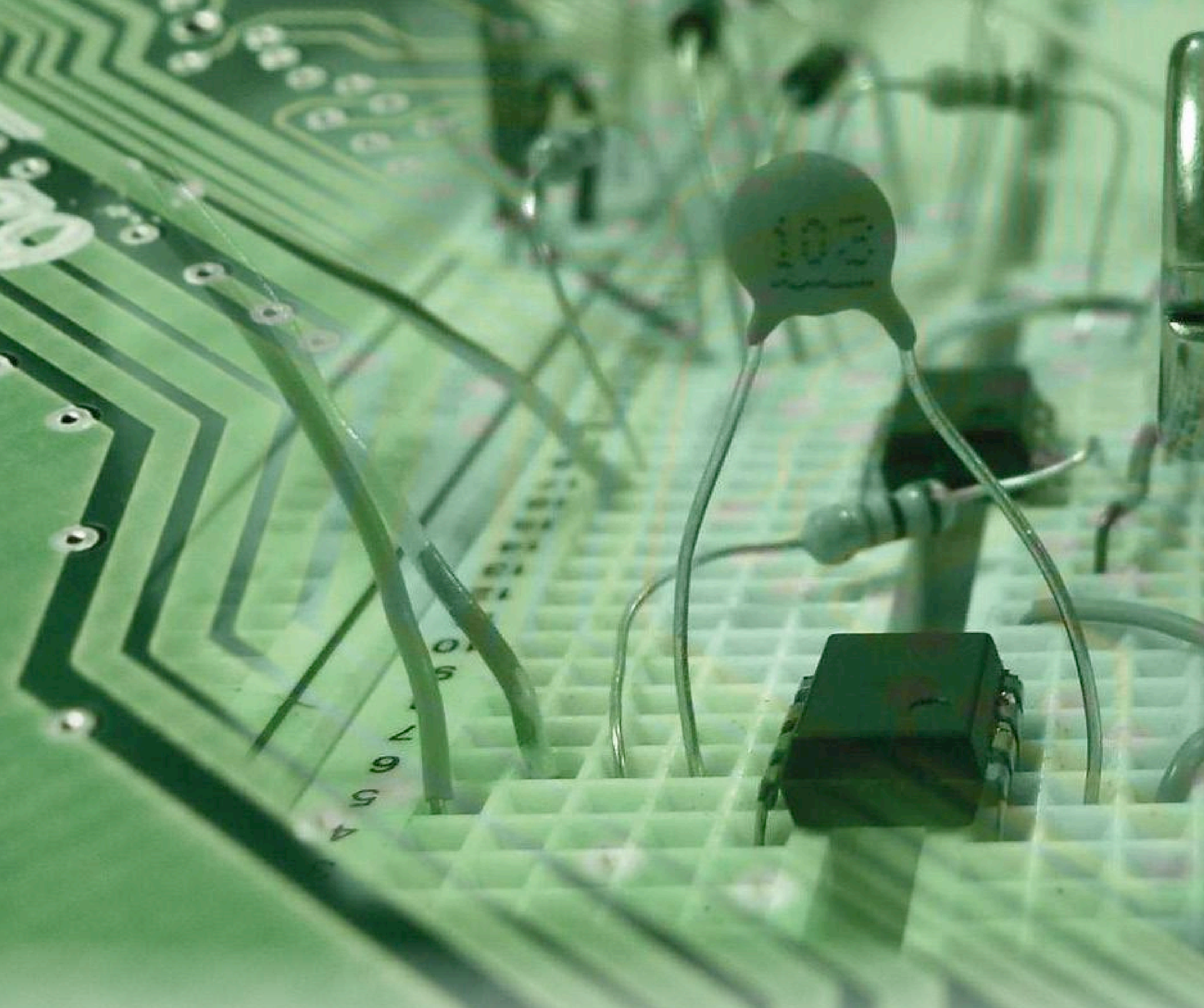


PCB Design is Not Easy

The art of laying traces

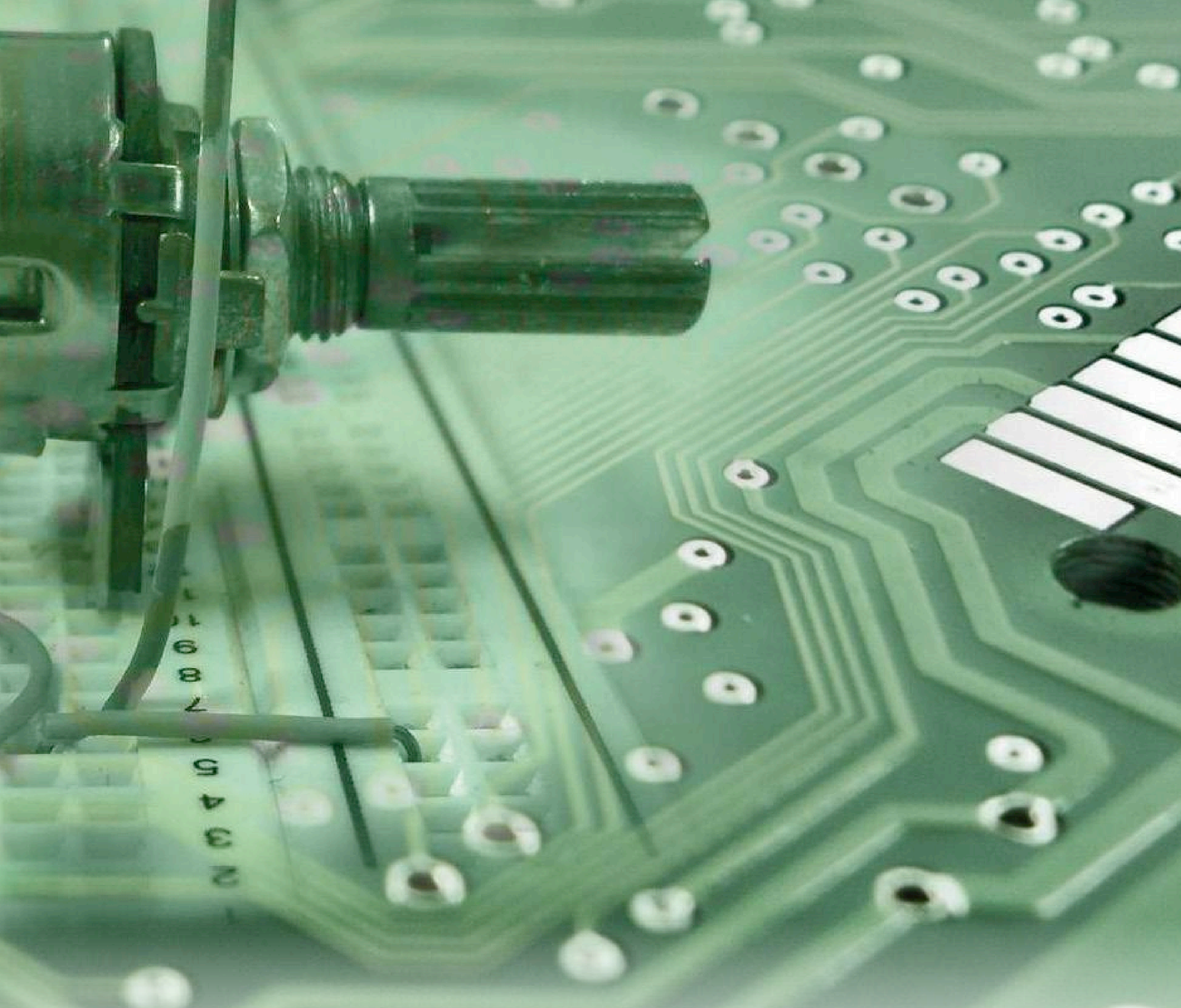
By **Clemens Valens** (Elektor Labs)

Even if such a naive approach may work for simple, low-power and low-frequency circuits, it is wrong. Simple circuits need good PCBs too. Good PCBs not only make a circuit work reliably and as intended, they also limit EMC problems as much as possible, they provide test points for system assembly and repair, and they are easy to fit in the final application. Because that is what a circuit board really is, a component in a system, and as such it has to be as good as any other component used in that system. A PCB design isn't intended to please its designer; it must please the end user, whoever that may be (**Figure 1**).

In what follows, acronyms and jargon are used. Refer to the **Glossary** further on in this article to better understand the text. Also, even though this article is quite long, it is far from complete. PCB design is simply too vast a subject to be treated comprehensively in one article.

System integration

The first step — skipped by most people — involves studying how the board is supposed to be integrated into the system. What shape should the board have? Where should the mounting holes be? How many? And the cables and wires to and from the board? Where do they come from, where do they go? Are there any height limitations? Possible heating issues? How is the user going to interact with the board? Will there be pluggable cables? On the rear or on the front, on the side maybe?



To many people, designing a printed circuit board (PCB) is a mere detail or afterthought. The circuit has been tested thoroughly and then, without thinking about it too much, they plunk it onto a board to make it easier to move around without wires coming loose or components dropping off. How wrong they are…

Rotary encoder or pushbuttons? Display? LEDs? Are there possible issues with the material the enclosure is made of? Oh yes — another afterthought — what about the power supply? Even if the system consists merely of your circuit mounted in an enclosure, then, before you set out designing a PCB for the circuit, choose a suitable enclosure and adapt the PCB's size and shape to it.

Unless you have excellent metal, wood and plastic working skills, or have access to CNC and laser cutting equipment, try to limit the mechanical work to a minimum.

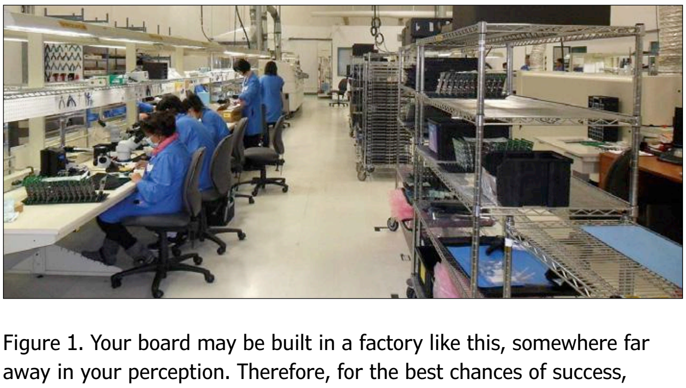


Figure 1. Your board may be built in a factory like this, somewhere far away in your perception. Therefore, for the best chances of success, design it in such a way that it can be done with as little explaining as possible.

Board manufacturing

Another important question is: how is the board going to be made? Are you planning on etching it yourself at home? Are you

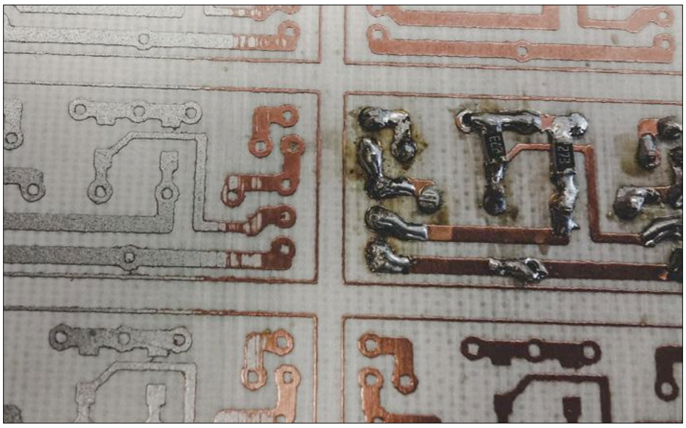
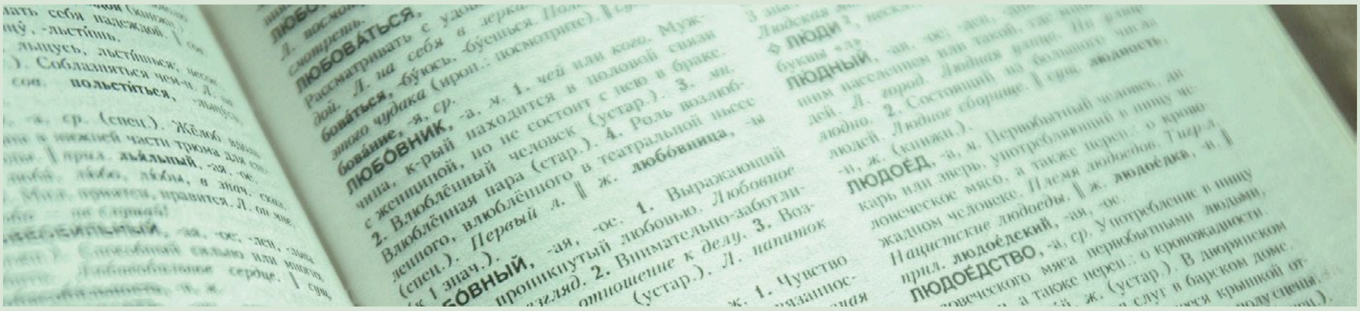


Figure 2. A home etched panel with many holes left to drill, and panel cutting still to be done. Is it really worth it to do this yourself?

any good at it? If not, you may want to avoid vias, and stick to one layer. Double-sided boards require excellent alignment of the top and bottom films, and metalizing vias can be a lot of work, especially when there are a lot. Having said that, many home etchers achieve great results with double-sided boards; it is mostly a matter of skill and experience.

You may want to stay away from ground and other copper pours (planes) since you don't have a solder mask to avoid solder bridges. Finding and cutting almost invisible shorts is an unpleasant job as well. Maybe use thicker traces to prevent them from getting eaten away by the etching fluid? To avoid copper bridges, don't place traces too close to each other. Use larger pads because you're not so good at drilling (**Figure 2**)? A PCB engraving machine has similar requirements but, unless you are a very good home etcher, it has higher precision. The trace density can be higher, but not too high either. If the mill-

A Small Glossary of PCB Design



Auto-router: Holy Grail of PCB design software developers

Bottom: the lowest layer of the board stack, also still commonly known as solder side

Class: resolution or density of a PCB; the higher the class, the smaller PCB elements and clearances are allowed to be

Clearance: distance between two or more PCB elements

DRC: design rule check, to verify that PCB elements respect a set of required design parameters like minimal trace width, minimal drill size, minimum clearance between pads, etc.

ERC: electrical rule check, to verify that nets are connected, not overlapping, no shorts exist due to copper pours, etc.

Excellon: data format for CNC drill and route machines

Fiducial: special mark on a film, mask, board, panel, etc., to help aligning them with cameras, stencils, machines, each other, etc.

Gerber: ASCII vector data format for two-dimensional, two-color images

Heat relief: pad-copper (plane or trace) connection preventing solder heat from being absorbed by the copper

IAR: inner annular ring

Layer: surface carrying PCB elements like traces, pours and components

Metalized: see Plated

Mil: one thousandth of an inch

Millimeter: one thousandth of a meter

Net: a desired connection between two or more pins

OAR: outer annular ring

Pad: PCB element for connecting a pin

PCB: printed circuit board

PCB element: object printed on a layer, includes the board outline

Pin: connection point of a component like a pad, pin, lead, etc.

Plane: large copper area; called power plane when connected to ground or a supply voltage

Plated: covered with a conducting material

Pour (copper): see Plane

Push and shove: routing a trace by pushing and shoving PCB elements surrounding the trace to create enough clearance for the new trace

Ratsnest, rat's nest: a visual representation of all unconnected nets

Resist: protected against solder(ing), see Solder mask

Routing: transforming nets into traces.

Silk screen: a non-conducting layer of graphic symbols and text, usually white, also known as component print

Short: an undesired connection between two or more pins

Solder mask: mask with openings where solder(ing) is allowed. A solder mask not only prevents short circuits, it also avoids solder migrating away from the pad which may result in bad solder joints or pulling an SMD device out of alignment.

SMD: surface-mount device, a device using SMT

SMT: surface-mount technology, for parts with terminals that are not to be inserted in holes

Stack: neat pile of layers

Stencil: mask to apply solder paste on a PCB

Terminal: see Pin

THT: through-hole technology, for parts with terminals that are to be inserted in holes

Tombstoning: the partial or complete lifting of two-terminal SMT components during reflow

Top: the highest layer of the board stack, also known as component side

Trace: a connection between two or more pins on a PCB

Track: see Trace

Via: metalized hole connecting traces on two or more different layers

Via stitching: the use of many vias to connect a copper element on one layer to another element on another layer; often used for conducting heat or on high-frequency boards ing is too deep, traces may disappear. If on the other hand the milling is not deep enough, shortcuts may appear. If the board is not flat, both problems may occur simultaneously. Another thing to keep in mind is that contrary to etching, milling leaves the unused copper on the board. This makes soldering harder (again, no solder mask) but can also create dangerous situations with circuits that connect to high voltages like the AC line grid because the clearances between traces are no longer respected. This can also pose problems in circuits with high-impedance inputs. It is possible to mill away the unused copper, but it will take much longer to fabricate the board.

You can manually mill a board with a Dremel-ish tool or suitable milling bits like those ball-shaped drill bits you may have seen your dentist use. Like him/her, practice and skill are required to obtain painless results.

Then there is, of course, the professional route. Pooling services that combine your design together with designs from other customers on one supersize board are widely available on the Internet. Prices and delivery terms vary wildly so it's wise to compare several providers. The costs depend mainly on board surface, number of layers, board class and production delay, and adding options will make it more expensive. Some services allow you to take options away, others don't and you may end up with features that you don't need. Pooling services in places remote to you may seem attractive but shipping delays are not always respected and packages may get lost. Some services offer a price per board including shipping, others charge setup and/or tooling costs.

Usually there is no reason to try to make the PCB as small as possible. It may be cheaper to manufacture, but doing so makes the board more difficult to route, and more complicated to integrate let alone repair.

A word on milling: sometimes non-circular milling is required to fit a part or to allow access to something. Unfortunately, milling requires yet another tool, meaning that it usually increases board costs. Some manufacturers do not add costs if the milling tool bit is the same as the one used for the board outline. Also, milling may not be possible as detailed and accurately as you would like, so, before milling away shapes, check with the manufacturer to see what is possible and what is not.

Component placement

Components should not be placed anywhere or anyway you like, not even when the circuit allows you this virtual luxury. For ease of installation, inspection and repair, components should be arranged in rows and columns, and oriented uniformly. Although for one-offs to be assembled by you this seems unnecessary, by doing so your life may become a little bit easier, as well as those of the people that assemble and repair boards for a living.

For electrical reasons components should — in general — be placed as close to each other as possible. Of course, there are many more criteria to take into account here, like current return paths, avoiding crosstalk and other unwanted inductive or capacitive couplings, etc. In short, it is important to carefully plan component placement. A power amplifier simply does not have the same requirements as a high-precision measurement device.

However, the soldering technique used to mount the components may impose yet another set of constraints. The environment too can have an impact on component placement.

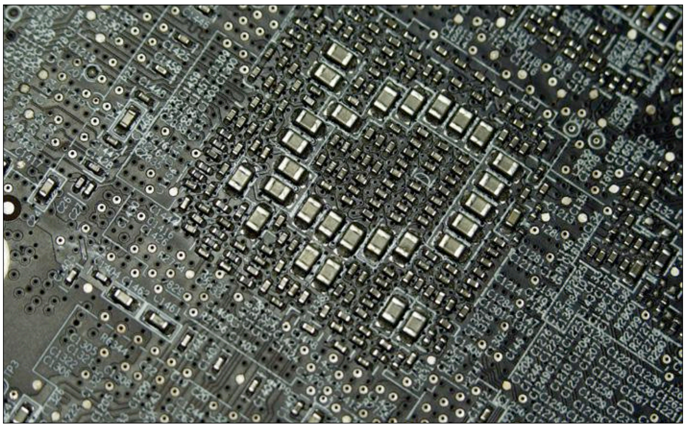


Figure 3. To provide proper decoupling for ICs with many pins, the decoupling capacitors can be placed on the other side of the board. Note how the component designators are placed outside the densely populated area in an intuitive way.

Humidity and/or dust may pollute the board, requiring larger clearances around traces and components to avoid shortcuts and leaks. This all is closely related to any norms and standards your board may be required to meet. Even though this may seem a non-issue for most DIY boards, there are good reasons why these standards exist, and reading up on them might actually teach you a thing or two.

Place THT components on the top side, also called the component side. Place SMT parts all on the same side, top or bottom; avoid placing them on both board sides as it makes manufacturing more expensive. Again, even though this may not be an issue for one-offs and small series, it is good practice to try to avoid SMDs on both sides where possible (**Figure 3**).

Boards that have SMT components on them need at least three fiducials as reference points for pick and place machines. Large ICs with many fine-pitched leads may require fiducials too to ensure correct alignment. DIY boards can, of course, do without fiducials, but why not try to make a habit of doing things the professional way? It doesn't cost anything.

Orient all polarized components such as capacitors and diodes in the same direction (unless signal integrity or other good reasons forbid you to do so). It is a great time-saver during board assembly, inspection and faultfinding. Always indicate the polarity on the silkscreen.

Beware of mirrored footprints.

Stay on the grid

Work on a grid for as long as possible. I like to place the components and do the initial routing on a 50 mil grid. When things get too dense, I will switch to a 25 mil grid. When a dense board is almost done I may switch to a 5-mil grid to squeeze the last trace segments in. Occasionally a 1-mil grid may be required to precisely position an element while respecting the design rules (DRC). Mils, mmm, it doesn't really matter which unit you choose, as long as you stick to it.

Pads

Component leads and terminals are soldered to pads on the PCB. For many parts the pads not only provide the electrical connection, they double as the mechanical mounting points for the part. It is therefore important that the pad is large enough

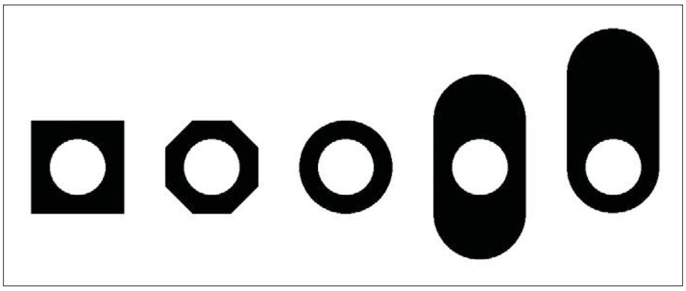


Figure 4. A selection of THT pads available in the popular Eagle PCB design tool.

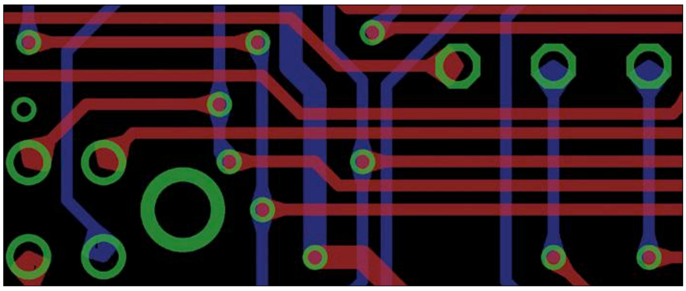


Figure 5. Tapered pads in Eagle, created with the teardrop script (ULP).

for this. Especially high or heavy parts with few terminals (e.g. large capacitors) need big pads to prevent them from accidentally being ripped off . The copper is glued to the board, and, especially when heated up (by soldering or current) delamination is a possible risk.

SMD parts usually have oblong shaped pads. Lead-free solder doesn't flow as well as traditional 60/40 solder, especially in the corners, which is why people have started to use pads with rounded corners. SMD pads must be large enough to hold enough solder paste to correctly solder a terminal. Refer to the component's datasheet for the footprint or landing pattern to use.

THT pads can have all sorts of shapes (**Figure 4**). Square pads are often used to indicate pin 1 of a part, say, a connector. Octagonal pads are popular, but round is the best shape as it maximizes the copper area (good for mechanical strength as well as heat dissipation) while simultaneously minimizing the space required to meet clearance rules.

Teardrop-shaped and tapered pads — where the pad gradually merges into the connecting trace — exist too (**Figure 5**). Besides giving a slight retro touch to the board, they do provide stronger pad-trace connections. This is important for boards that have to be able to bend, like flexible PCBs. Not all PCB design tools can do this.

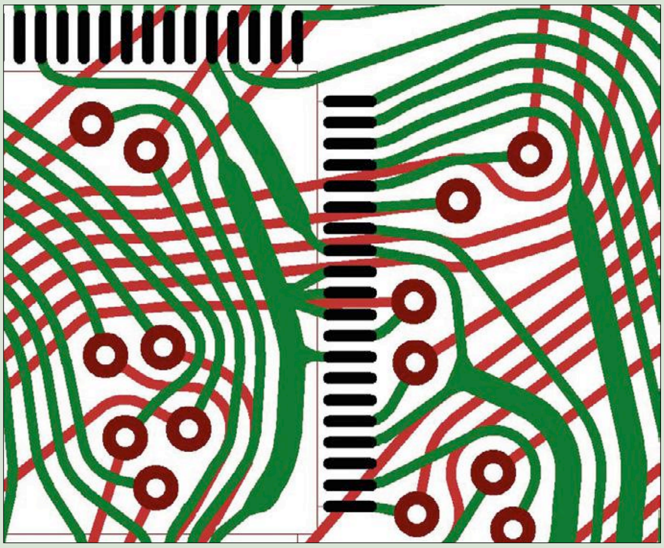
Then there is heat relief (**Figure 6**). When a component lead must be soldered to a very thick trace or a copper pour, the copper surrounding the pad may absorb the heat, making soldering difficult. To avoid the heat flowing away, pads can be connected with thin traces — so-called spokes or thermals, usually four — to the surrounding copper (this is where the inner annular ring makes its appearance). When doing reflow soldering (i.e. in an oven) this is less of a problem because the whole board is heated up to the same temperature. Therefore, there is usually no real reason to use heat-relief techniques

Automatic or manual routing?

CAD tool manufacturers and scientists spend and have spent many, many, many hours on developing and improving automatic routers and yet I have never been satisfied with the results of a single one of them. I also have never met anyone who was. One reason is probably that I have never had access to the best tools available, but the ones that I did try either crashed or gave up five minutes after I had left the office, never managed to route 100% of the board, if they achieved 100% then made me spend hours on cleaning it up, or were simply too complicated to set up. Because of all these reasons I prefer to route manually. (Also, I find routing a very relaxing activity.)

The automatic router often manages to route up to 90% of the board or better, but hardly ever achieves 100% unless the circuit is very simple. When it gets stuck before reaching 100% this usually means that there really is no solution left to finish the board. That doesn't imply that the board can't be routed, it just means that the autorouter has blocked all possible solutions. To get out of this hole you have to undo so much that in the end it would have been quicker to have routed the board manually from the start.

As a kind of compromise, some people use the autorouter to see where routing problems may pop up, then move some components to hopefully solve these potential bottlenecks and finally route manually. Others use the auto-router only on (trivial) parts of the board and then clean up afterwards.



The Eremex TopoR 'topological router' lacks preferred routing directions to optimize space usage. (Source: Eremex) on pads for SMT parts that are to be soldered in an oven. However, tombstoning — the partial or complete lifting of an SMT component during reflow — may occur when the thermal mass is very different on both sides of a small two-terminal device like a resistor or capacitor. Heat-relief can bypass this problem. Vias hardly ever need heat relief because, in general, they are not soldered.

Holes

Pads often have holes in them — vias have holes too. Mounting holes can be non-plated; holes in pads and vias are plated. You specify the plated-hole diameter for your vias and pads; it is the job of the PCB manufacturer to ensure that the finished hole matches this diameter. For a plated hole the actual drill size must be larger than specified because the plating material needs space. Drilling has limited accuracy and a hole may be drilled off-center (**Figure 7**). Therefore, to ensure that enough copper remains around a plated hole after drilling and etching the board, the 'outer annular ring' (OAR) should be wide enough. Inner annular rings (IAR) exist too. According to my dictionary, 'annular' means 'shaped like a ring', so, technically speaking, annular rings are "ring rings" (which reminds me of this great Abba song from 1973 of which I highly recommend the video on YouTube. They don't make music like that anymore…, nor apparel, Swenglish, guitars, hairdos, etc., but I digress.)

A common mistake is to specify the wrong drill size for a component lead — either too large or too small.

There is an issue related to vias and wave soldering: if a via is unmasked, solder may flow up through it and potentially damage a component mounted over it. Closing the solder mask for vias will usually prevent this.

Traces and planes

Keep traces as short as possible; you knew that, of course. This is especially true for high-frequency signals, but low-frequency and even DC signals benefit from short traces too. Not only are short traces good for signals, they also save board space. Some fast signals may require traces (or a pair of traces) of an exact length (and impedance) which is not always the shortest connection possible.

Do not use the thinnest pen from your PCB design tools. Thin traces may cost more and are fragile. The finest detail on the board determines the board's manufacturing class; the higher the class, the more expensive the board. Vibrations can cause micro cracks in thin traces resulting in bad or even broken connections. If the board has to be soldered by hand, or if rework is to be expected, thin traces may easily delaminate when too much heat is applied for too long. Home etching usually is not well-controlled and thin traces may get eaten away. Furthermore, a thin trace may not be capable of carrying the current that you had in mind for it. Adapt the trace width to the current that has to flow through the trace, the extra width is needed to keep the trace cool. Many online calculators are available to help you determine the best width for your trace. A thin trace carrying too much current will heat up and may eventually melt or crack; it is the heat that limits the current. Vias also must support the current. Covering traces with a thick layer of solder will increase the maximum current a trace can carry. Similarly, filling a via with solder or copper helps too. Providing openings in the solder mask will be useful here. Multiple vias "in parallel" are often used to improve current conduction, while limiting the risk of open circuits due to a broken via. However, one large via can be used too.

Naming nets is helpful during routing as they remind you of the signal you are working with.

Then, corners! Rounded traces are best because, contrary to angled traces, they have constant width. Changes in trace width cause impedance mismatches and reflections, but only at very high frequencies so most circuits are not affected by

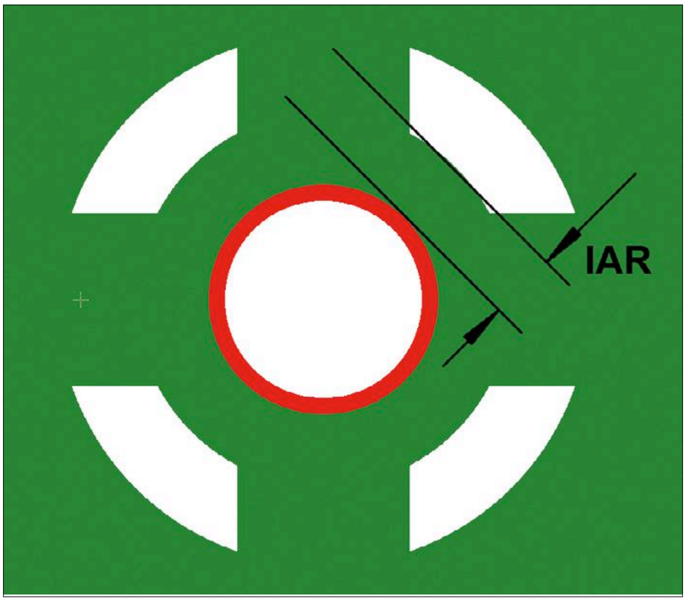


Figure 6. Heat relief prevents too much heat flowing away into the large copper pour during soldering. Here the minimum inner annular ring (IAR) may be specified even though it is much the same as the OAR. The red ring represents the plating of the hole.

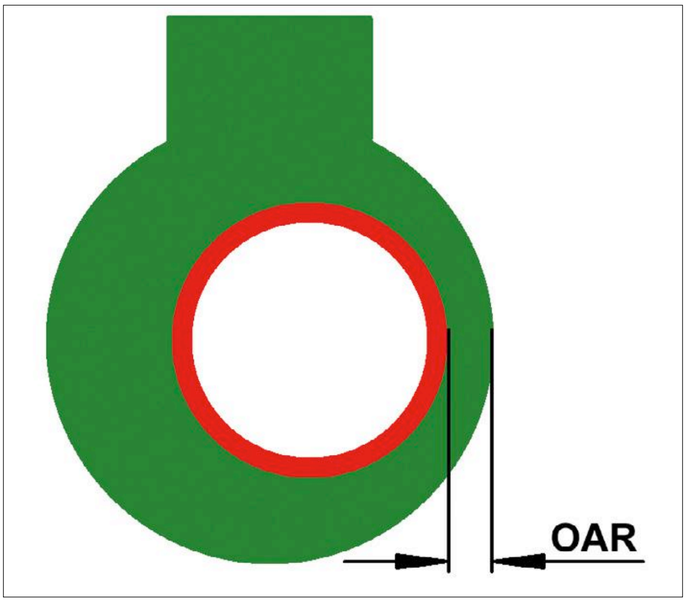


Figure 7. Due to manufacturing tolerances a hole may be drilled off-center. To allow for plating (the red ring) the hole must be drilled larger than specified. The finished hole (with plating) will have the diameter you specified. The ring that remains after drilling is the annular ring (AR). The minimum acceptable outer annular ring (OAR) shown here is specified by the design rules (DR).

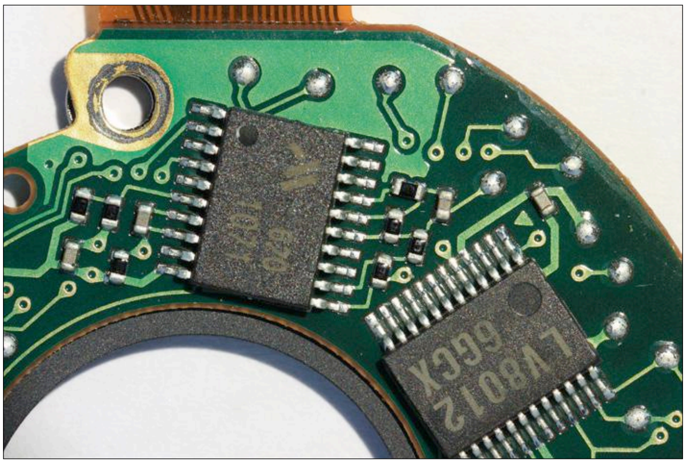


Figure 8. No component designators, only one pin-1 marking, traces with all kind of angles and teardrop vias. Once you know what you are doing it is *common practice* to be liberal about *good practice.*

these problems. Right angles are considered ugly, yet you will find many when you look closely at the vias on a board. Often a trace on one layer continues at a 90° angle on another layer, and even if it doesn't, the via itself introduces two 90° angles. Traces with 90° angles tend to be longer than those that use 45° angles. Sharp corners may also introduce delamination problems, acid pockets, or on the other hand, under etching. Sometimes it is impossible to avoid a sharp edge and in most cases that is fine. Personally I tend to stick to 45° angles whenever I can.

Using a power plane or copper pours (usual for GND) can save a lot of work and is good from an EMC perspective. However, be careful when using multiple supply planes, especially close to board edges and mounting holes where metal objects and fixing screws may create unintended short-circuits between planes. Always run an electrical rules check (ERC) when done to ensure that all short and open circuits have been corrected. When you use a plane or pour for a net, do not route that net but let the copper pour take care of it. Beware of very thin, sort of accidental, connections between plane segments that may get etched away and break the continuity of the plane. Such planes are unsuited for home etching. To prevent this from happening, start with a high copper pour clearance and see if the copper flows everywhere. Move traces and vias to improve copper flow. When you are satisfied with the pour you may decrease the clearance.

Note that planes have an influence on the copper distribution of the board. When the copper is unevenly distributed etching may become uneven, and can result in uneven copper thickness distribution making the PCB bend at extreme temperatures (as in automotive applications).

Use design rules

Use the design rules checks (DRCs) and electrical rules checks (ERC) to make sure all PCB elements respect them. Good design rules will help avoid traces from overlapping or coming too close to the board's edge, help you find illegal drill sizes and clearances and much more. Use them! Also make sure that when you think the board is ready, all nets are connected indeed. Do not leave warnings and errors even if they are acceptable for you because the person that inherits the design — or yourself when you have another look at it six months later — may get confused by them. If you must accept a warning, document the reason why. If possible change the design rule that gives you trouble to make it go away, but make sure the new rule is acceptable for your application as well as for your board manufacturer.

Markings and component print

Try to provide designators for all component outlines and make sure they remain visible after the components are mounted. I like to orient designators the same way as the components, in rows and columns (see also Figure 3). Do indicate pin 1 on all connectors, headers, and any other component where pin 1 is not readily identifiable. Also clearly mark the polarity of all polarized components such as capacitors and diodes (**Figure 8**). Avoid print under SMT parts, especially two-terminal ones, as this makes their "seat" wobbly and may result in tombstoning. Keep text legible and use labels where useful to guide the user. Note that the openings in the solder mask are usually slightly larger than the pads, and any silkscreen print crossing these openings is cut away so make sure text does not overlap them. Put legible text on all layers to prevent accidental mirroring of a layer. Label or number layers to ensure that they will be stacked in the right order.

Give the board a unique name or number and don't forget to mark its revision.

If the board's bottom side can have print too, use it.

DIY boards in general do not have a silkscreen, but that doesn't mean that you should forget about text and polarity markings; you can write on the copper layers too. So, where possible place marks and text while keeping an eye on their size to avoid that they will be lost during etching.

Testing

Provide easily accessible test points. If possible make the circuit produce useful test signals. Component terminals should not be used as test pads as a test probe pushed against a terminal may temporarily "repair" a bad solder joint, feigning a "good" connection that's bad most times. Vias can be useful for testing, but only when they were left open in the solder mask.

Is that all?

After reading this long article you may think that you know a lot about designing PCBs, but in reality we only scratched the surface. We did not discuss, to name a few, current return paths, multiple ground planes, heat management during soldering and heat management when operating, EMC compliance, handling high-speed signals, designing footprints, etc. PCB design is a vast theme combining chemistry, physics, electronics, mechanics and automation. It is rather amazing to discover that many aspects of PCB design never really have been studied in detail and are simply based on common sense and assumptions. On the Internet you can find many discussions on PCB design details, so, when in doubt, ask around.

Thanks to Malte Fischer for his useful suggestions.

(160397)

Web Link

[1] www.elektormagazine.com/160397