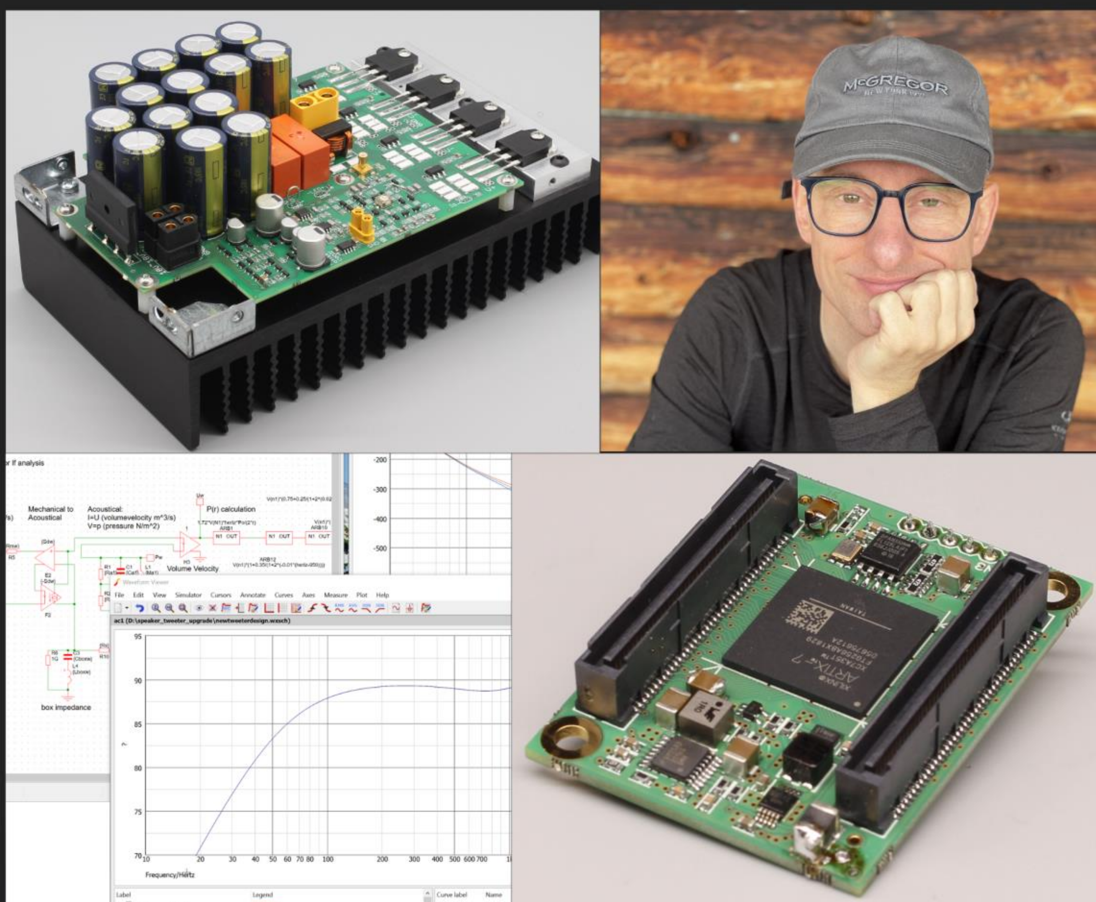
The Electronic Product Development Checklist

Hans Rosenberg



THE ELECTRONIC PRODUCT DEVELOPMENT CHECKLIST

Hans Rosenberg

www.hans-rosenberg.com

Version 5

Electronics is full of dozens of sneaky landmines that can quietly ruin your design.

You won't find them in any university course, you only discover them the hard way, after they've blown up your project.

Each one can wreck key parameters, forcing weeks of stressful debugging when you don't even know what went wrong. I've seen a single mistake cost **$100,000**.

The good news? Once you know where these landmines are, they're easy to avoid, and that's when your designs start meeting **all key specifications on the first build**.

I've spent 42 years in electronics, 30 of those professionally hunting down every one of these traps. All of that experience is in my Electronic Product Development Course: https://www.hans-rosenberg.com/EPDC_information

This checklist is a condensed summary of that course. Unfortunately, many of the most critical details are too in-depth to include here. It has already been downloaded over **8,100 times** by engineers worldwide, including many with decades of experience, who have said they immediately recognized the issues and failure points I have identified.

Keep it on your desk and you will avoid a lot of unnecessary problems already.

Now lets get into it 😊

GENERAL IDEA: THE ‘THINK 5 STEPS AHEAD’ FRAMEWORK

A lot of the ideas in this document are related to a framework I came up with after designing my first piece of electronic equipment (which was a total disaster because I didn’t even think a single step ahead 😊) called ‘Think 5 Steps Ahead’. This framework relies on asking yourself a number of questions when working on each part of a design. You can also use something similar for software development. The best thing is to always have these questions in mind during the whole development process.

1. Think of everything that might go wrong with this part of the design. Maybe there are things you’re not 100% sure of until you build it. Can you introduce options in the design to counter that if it happens? Maybe you need to build a small prototype of a subcircuit?
2. Can this piece of the design be assembled easily? Sometimes you think of something great, but it is very hard and time-consuming to assemble, which makes it bad for series production and maintenance.
3. Are there any reliability or lifespan issues? Check thermal loads, aging, maximum voltages/currents, things like that.
4. Can you combine functions in a single part? This can greatly increase the simplicity of a design and reduce cost. Somewhat jokingly: ‘The best part is no part’.
5. Can you simplify this part of the design further? Using the KISS principle (Keep It Simple, Stupid) everywhere can greatly improve your design. Incredible projects like the SR-71 Blackbird were done with this idea in mind to great success. SpaceX and Tesla also use this philosophy. It doesn’t mean designs are not complex; you just find the simplest solution.
6. Can you reduce the number of unique parts used in the design? This can often be done with resistor and capacitor values. This greatly simplifies sourcing parts and manufacturing.

THE STEPS OF AN ELECTRONIC DESIGN PROCESS

In order to go from an idea to a functional electronic product on a PCB, there are a number of steps to take. They are listed below:

1. Schematic design, usually done assuming ideal components.
2. Choosing 'real world' components with the right specs instead of ideal ones.
3. Determine a board outline, including all mounting points and critical positions for interfacing like LEDs, buttons, etc.
4. Component placement on the PCB.
5. Check library parts.
6. PCB layout.
7. PCB assembly.
8. PCB testing and measurement.

This document mainly focuses on points 1, 2, 4, 5, and 6. While the others are also critical to making a good product, these 5 points can really hurt the electronic performance of a PCB, which is what we'll be focusing on in this document. We can't go into the details of schematic design (that would fill a whole bookshelf), but we can set a list of standard things to check that will ensure good operation. Each chapter of this document will deal with the given points.

STEP 1: SCHEMATIC DESIGN

In this chapter, we'll look at schematic design. Firstly, we'll explore effective ways of drawing a schematic so there is less chance of problems. Secondly, we'll examine a comprehensive list of common errors that can easily be prevented by checking for them.

How to draw a schematic design to make your life easy

Drawing a schematic design the right way is crucial to improving your chances of getting your design right the first time. In electronics, it's very easy to make a small mistake with big consequences. This is why it is important to draw a schematic in the most comprehensible way so that errors can be easily spotted. It's hard enough already, let's make our job as easy as possible.

An extreme example of what not to do, which I've encountered multiple times, is a schematic with, say, 20 schematic sheets, where each sheet contains a single component and everything is connected with labels (by labels, I mean net-names). There is no way you can easily understand what the circuit looks like. That means it's also very very hard to evaluate if there are problems.

Another example was very similar, all components were on the same sheet, but everything was connected using lables: Also very hard to comprehend what is going on.

So what can we do to make it easier to read?

- Minimize the use of labels! There are three very clear exceptions:
  - Power supply and ground labels/symbols
  - A databus with a clear group of signals that belong together.
  - Single digital I/O lines or control lines. It's usually implied that they go to an FPGA, MCU, or another part of your circuit, which is often on another sheet.
  - Connections between sheets, where each sheet contains a complete circuit block.
- Try not to use labels to connect items on the same sheet; instead, place the schematic blocks in such a way that you can easily draw a wire. This likely represents the signal flow more accurately as well.
- Group related circuitry on a single sheet with minimal labels on the I/Os. Use a larger sheet if necessary. Maybe you need to put components a bit closer together to achieve this on your schematic sheet. Do whatever you can to make sure you get a clear overview of that circuit block.
- Avoid having many connections crossing each other, which makes it difficult to follow the signal path (if you can, sometimes it is impossible to avoid).
- Arrange sheets in a logical order, from input to output.
- For large projects, include a block diagram that provides a clear overview of how everything is connected.

Schematic design checklist

Detailed schematic design is out of scope for this checklist, but there are a number of common mistakes that can be prevented and can happen in almost any schematic design.

- Have you checked for nets with only 1 pin? This usually means you have an unconnected pin somewhere.
- For labeled nets: Did you check all of them to make 100% sure they're all correctly connected?
- Is the polarity and voltage for electrolytic capacitors correct?
- Have you checked for 'double names'? For example, you have a net named VCC and a net named 5V. You really intended for them to be the same net but accidentally used different names on different sheets or on the same sheet, leaving them unconnected. The same can be true for AGND, DGND, or GND. Pro tip: Never use multiple ground planes unless they're galvanically isolated.
- When using ICs: Have you read the datasheet and are you applying all the guidelines for the specific ICs you're using? Be careful! Some datasheets suggest it is a good idea to split grounds: This is wrong information because they don't understand grounding! Use a single ground plane.
- Is there enough power supply decoupling?
- Do you have relays in your schematic, and have you applied clamping diodes across the relay coils to prevent damage to the switching component?
- Do you need to include fuses for protection/safety anywhere?
- Have you copied parts of a schematic from another project? In that case, check the net names very carefully. Some PCB design packages don't warn you if there are names that are already in use. This can lead to unwanted connections.
- Have you added test points for voltages, currents, or signals which may be very valuable to be easily measured?
- Are there any parts of the circuit design you're not sure of? It might be a good idea to add some options in the schematic to try different configurations for those subcircuits.
- Have you added a GND measurement pin? This is very helpful when measuring your product later on.
- Double check transformer directions. Make sure common mode transformers are correctly wired if you have them.
- Make sure that loose inputs are connected to ground or an appropriate input voltage (such as for ADC inputs, for instance) to prevent noise due to uncontrolled switching.
- Check component values, it is easy to forget a 'k' in a resistor value for instance. 10 instead of 10k.
- Check the mounted / not mounted properties of all components so you get the right components assembled on your PCB.
- Are all devices within their operating conditions, like maximum voltage, current or power dissipation?

STEP 2: SELECTING ‘REAL WORLD’ COMPONENTS

A schematic assumes ideal components. We have to select real ones that are suitable for the application. The goal of this selection process is to minimize the ‘damage’ we do to the schematic. What I mean by damage is that real-world components are never as good as ideal ones, so you have to find the best match. Cost is also a consideration here. We’ll go over all common components in the next paragraphs.

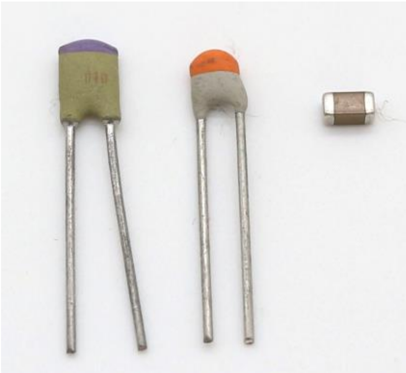
CAPACITORS

Real-world capacitors have a huge variety of specifications that can really ruin circuit performance in an unfortunate situation. This is why this paragraph is ‘annoyingly’ large. There are a number of things to consider:

1. **ESR (Equivalent Series Resistance):** Varies greatly across types. Low ESR means high Q. High ESR can really ruin certain circuits like resonators, filters, RF circuits, etc.
2. **ESL (Equivalent Series Inductance):** Also varies greatly with type. Causes series resonance. In general, you should always operate a capacitor under its resonance frequency; otherwise, it acts like an inductor.
3. **Derating:** Capacitance reduction due to DC bias voltage across the capacitor. Can be up to 84% with class 2/3 ceramic capacitors.
4. **Maximum voltage:** If you exceed it, lifespan is reduced, or you may get smoke or a bang, or both.
5. **Temperature coefficient:** Can greatly decrease capacitance over its temperature range. Class 2/3 ceramics really suffer from this.
6. **Leakage:** Electrolytic capacitors have leakage; check that your application can deal with it.
7. **Polarity:** Some capacitors have polarity; if you connect them the wrong way, you get a nice puff of smoke and maybe a bang as well.
8. **Microphonics:** Class 2/3 ceramic capacitors suffer from the piezoelectric effect, basically turning into microphones. This may be an issue in high vibration environments in combination with very sensitive sensors.
9. **Lifespan:** Electrolytic capacitors have a lifespan depending on temperature and load current.
10. **Price:** Often there are multiple capacitor solutions for a given value that meet specifications. In that case, it is good to check the price difference.

We’ll go over the different types of common capacitors and their behavior so you can pick the best one for your application.

Ceramic capacitors

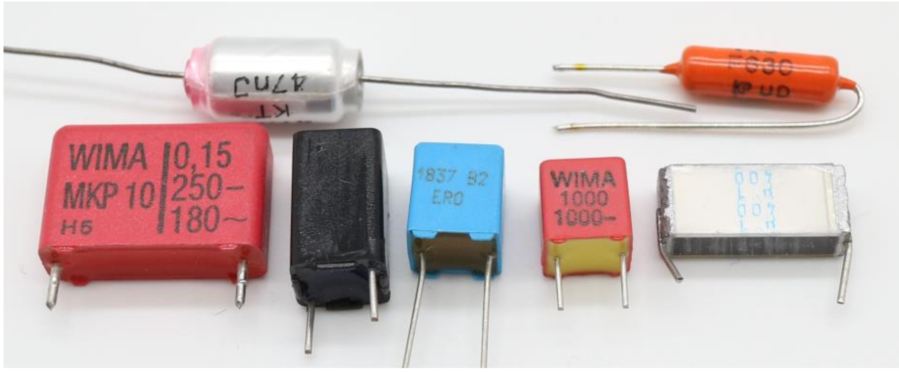


The most common capacitor. They are very cheap and available from sub-pF values up to 100uF (large 2220 SMD package). Watch very carefully what type of capacitor you use depending on the use case. The loss, accuracy, and value can be greatly affected by the type. For instance, capacitors with an X5R ceramic dielectric can lose 84% of their value when used at their rated voltage; this can lead to all sorts of problems. In general, there are two main classes which are discussed in the next two paragraphs.

NP0 / C0G (Class 1): Low loss, very stable and more expensive. Values are relatively small (max 220nF/25V in a 1210 SMD package). The value is hardly affected by temperature and can be very accurate at 5%. If you pay more, you can get down to 1%. The capacitance is not affected by voltage unless you use them at a higher voltage than specified, in which case you'll probably break the part. You use these if you need to make accurate filters, RF matching circuits, oscillators/resonators, filters for PLL circuits, low distortion coupling, etc.

All others, dielectric code starting with X/Y/Z like X7R (Class 2/3): Moderate loss, not very stable, very cheap, and high values available (max 47uF/25V in a 1210 SMD package). The capacitance can reduce drastically due to temperature and voltage applied. You can get them at 5%, but that is a bit of a useless number since the value decreases as you put a voltage across them. The X5R dielectric is the worst I've seen with an 84% loss of capacitance at the rated voltage. The worst-case temperature dependence I've seen is 70% when going from 25 Celsius to 75 Celsius! These capacitors are generally used for power supply decoupling. In general, you need to use a much larger value than you want since you'll lose a lot of it. You can also use a capacitor with a much higher voltage rating than you need to mitigate this effect. I use a lot of 100nF/100V X7R capacitors, for instance, which I use up to 35V. Another nice pitfall is when you're making an RC time constant for a timer. The non-linearity of these capacitors will make the time constant much lower than you expected. If you use these as a coupling capacitor (for an audio signal, for instance), you may cause very large distortion due to the value change versus voltage applied. If you really need a large capacitor, the aluminum polymer capacitor can be a very economical solution.

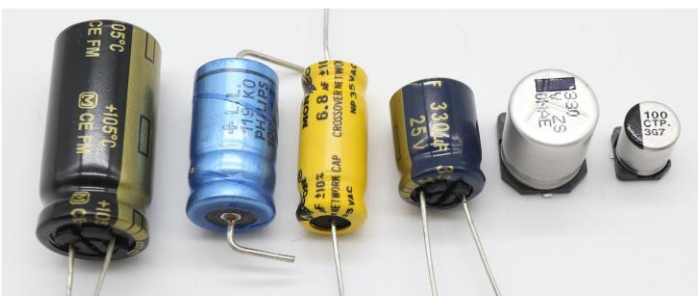
FILM CAPACITORS



These capacitors use a plastic dielectric and are generally all pretty stable, low loss, and accurate. They are more expensive than ceramic capacitors and available at very high voltages. The most common dielectrics are polyester, polycarbonate, and polypropylene. They are generally through-hole devices, although there are also SMD film capacitors using a polyphenylene sulfide dielectric. These capacitors can have values in a range from 33pF/1000V, which is 5x6mm in size, up to 330uF/250V, with the size of a soda can for loudspeaker filters.

If the value needs to be very accurate or you need very low loss, make sure you check the datasheet. For the lowest loss, use polypropylene.

ELECTROLYTIC CAPACITORS



These capacitors have moderate to high loss (depending on the type, check the datasheets) and are generally larger in size and value. They offer the biggest bang for your buck when it comes to capacitance. These capacitors are generally used for supply decoupling. Make sure that the leakage these capacitors exhibit is not a problem for your circuit. You can get them from 1uF/25V up to 1.3F/25V (size of a soda can with screw terminals). I'm using 25V here for comparison with the other types of capacitors; you can get them from a few volts up to around 700V. Stability is pretty good versus voltage and +/-10% over their temperature range (-40 to 105°C for high-performance electrolytic capacitors). The most important thing to realize is that they are polarized: They have a + and – terminal. If you connect them wrong, they will pop open or explode, releasing dirty brown muck and smoke (great for viral YouTube videos 😊). Another crucial thing to realize is that electrolytic capacitors (elcos) have a lifespan. After a certain amount of time, they break down. This lifespan is negatively impacted by temperature and the current running through the capacitor. The lifespan decreases with current and temperature. This is why electrolytic capacitors generally have a current, temperature, and lifespan specification like 3000 hrs @ 105°C (at maximum rated current which is usually specified as an RMS current at a given frequency or multiple frequencies). A general tolerance for an elco is 10 or 20%.

POLYMER CAPACITORS

Polymer capacitors are relatively new; they are 'dry' electrolytic capacitors (elcos). Normal elcos have a liquid dielectric that runs out after a certain amount of time. This means polymer capacitors can have a much longer lifespan (not all of them have a higher lifespan than normal elcos, so check the datasheets). The losses are much better than normal elcos, making them an excellent replacement for high-value ceramic capacitors since they don't lose a large part of their capacitance when biased or when the temperature goes up. The temperature variation is around +/- 10% going up with temperature, similar to a normal elco. Basically, this is a 'normal' elco with a lower value range, lower losses, and potentially a longer lifespan.

TANTALUM CAPACITORS

I don't see these being used much anymore. I think they've mostly been replaced by polymer capacitors. However, they're still available for purchase. Essentially, they are improved electrolytic capacitors.

THE 'HOLY GRAIL' OF CAPACITORS: SILVERED MICA

If money is no object and you want insane stability, accuracy, and lifespan, you need a silvered MICA capacitor. The highest value you can get is 47nF/500V, which will set you back $44! You only use these if you have some crazy project with ridiculous specs.

RESISTORS

Resistors are far less problematic than capacitors. Most resistors will perform their job quite well assuming you do not exceed the maximum allowed dissipation (crucial to check this!) or voltage. There are a few considerations, however, when designing high-performance analog circuits. Some resistor types will display higher distortion or 1/f excess noise. This is noise that increases as the frequency drops. If your circuit is sensitive to this, use thin-film, metal-film, or wire-wound resistors. Other than that, the very cheap thick-film resistors will do the job just fine. If extremely low distortion is needed, avoid thick-film resistors and use a resistor with a lower temperature coefficient and a higher allowed dissipation to reduce heating-induced distortion. Another type of resistor is the carbon type, but they are very old-fashioned and I would not use them because of bad specs.

When a lot of power is dissipated, make sure the resistor can handle it and can get rid of its heat. This could be done using resistors that can be mounted on a heatsink, or by using thick traces or larger copper polygons to get rid of the heat, using your PCB as a heatsink. Large copper polygons will add parasitic capacitance, so keep the frequency range of the signal in mind.

Some resistors are capable of handling high peak loads for a very short period of time; these can be useful in certain power applications. Check the datasheets for this if you need it.

The ‘holy grail’ of resistors is the bulk metal foil resistor; as you can expect: ridiculous specs and price.

INDUCTORS

Inductors are usually not used in large numbers. There are three main uses: DC-DC converters, RF matching, and filtering/resonators. When using them for DC-DC converters, the best practice is to carefully read the datasheet of the DC-DC converter chip to make sure that you’re using the correct type of inductor. When using inductors for RF matching or filtering, the loss of the inductor can negatively influence performance. To ensure that the inductor you choose does not affect performance too much, it is good to simulate your circuit with a model that takes into account the Q-factor of the selected inductor. Also, make sure that the resonance frequency of the inductor is well above the frequency of the signal the inductor is used on; otherwise, it acts as a capacitor!

TRANSISTORS, DIODES, MOSFETS, ZENERDIODES ETC

We can assume that during the detailed electronic design phase, the correct components were selected. However, there may be thermal issues, can you get rid of enough heat and can the parts take the dissipation and current?

STEP 3: BOARD OUTLINE AND MECHANICS

Before you start placing components, you have to know the shape of your board and all mechanical constraints. It wouldn't be the first time I started drawing a layout only to realize I forgot to put mechanical mounting holes in the corners of the PCB so it can be mounted. You may also have connectors or buttons that you want in a specific location. You may need heatsinks or components that need to be mechanically connected to an external heatsink in a specific location. This step may be an iterative process with step 4 to gradually optimize the mechanical layout of the PCB.

STEP 4: COMPONENT PLACEMENT

Component placement is the next step after we know the outline of the PCB. It is crucial to do this right since we do not want to introduce unnecessary parasitic resistance, inductance, capacitance, or crosstalk due to bad component locations. We can follow a number of rules to minimize the impact.

Group circuit blocks together in layout blocks

Circuits are usually made up of a number of subcircuits, like amplifiers, AD-converters, microcontrollers, PLL ICs, receiver ICs, etc. For each of those subcircuits, we do the following:

- Fetch the main component.
- Get the decoupling capacitors and place them as close to the main component as possible.
- Get all other supporting components for that block and place them as logically as possible. What I mean by that is: Watch the unrouted lines connected to these parts and check if you can place them in such a way that you don't need vias to make unnecessary crossings of wires. Also, keep traces as short as possible to prevent unnecessary parasitics.
- Do not place unshielded inductors in line with each other: They will couple with each other!

Place those blocks logically on the board

Now that we have those blocks, we place them logically on the board. This process can be iterative as we try to find the best possible solution. There are a few general rules to follow during this process:

- Follow the same order for blocks as on the schematic design. If the schematic shows a number of blocks ordered like 1-2-3-4 in a signal path with multiple blocks, don't put them on the board like 1-4-2-3. This will cause unnecessarily long lines and crossings. Order them on the board 1-2-3-4 as well. The basic rule here would be to keep lines between all blocks as short as possible, especially blocks that carry high dynamic range analog signals.
- Keep 'aggressors' away from 'victims'. An aggressor is a source of interference like a DC/DC converter or switching digital logic. Victims can be receivers for GPS, DAB, FM, AM, or ADCs, sensitive sensor circuits, etc. Try to put distance between them on the PCB. If interference demands are really high, you may want to place shielding cans over the aggressor(s) or the victim(s).
- Sometimes the previous two rules may conflict with each other, then you have to make the best choice considering both rules.

STEP 5: CHECK LIBRARY PARTS

Before continuing with the next step, which is layout, it is good to check the libraries you've used. This step can also be done earlier, but the fact that you have placed components now helps since you can print that board on a piece of paper using a printer. Printing also gives you a really nice idea of what it is going to look like in real life. This is the list of checks for this step:

- Print the PCB layers with components placed on them using a 1:1 scale (check the scale after printing by checking the board outline size with a ruler to be sure). This allows you to place the physical components on that printed piece of paper to check the size and pad locations. This really is an 'idiot-proof' way of making sure your footprints are correct.
- Check the pinouts of the components you're using. Are all pin and pad numbers correctly connected? Be careful here, some datasheets are doing everything they can to confuse you. Luckily, this is a minority.
- Check pinouts of connectors; ribbon cables can easily be mirrored.

STEP 6: LAYOUT

The key part of layout is to minimize the 'damage' we do to the schematic design. What do I mean by that? A wire or connection in a schematic is ideal; it is basically a superconductor. However, when you make a layout, this wire turns into a copper strip that has resistance, inductance, and capacitance with every other conductor on the PCB. Your job as an electronics engineer is to minimize these parasitic components.

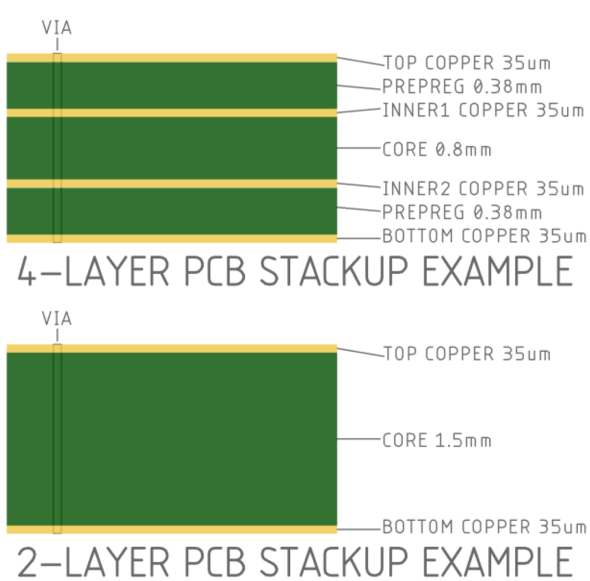
Layer stack-up

Before starting a layout, you have to determine how many layers you want to use. For simple boards, 2 layers with just through-hole vias will usually do the job. When it gets more complex, 4 layers with through-hole vias may be needed; these options are still very cheap. It must be noted that 4 layers are a lot better than 2 layers for high-frequency grounding when using the correct layer assignments.

When you're dealing with more complex boards containing BGAs and/or fine-pitch parts with lots of connections, you may need more than 4 layers and maybe more advanced via options like micro-vias or buried vias. Micro-vias can be placed inside an SMD pad without solder problems. Buried vias connect multiple layers on top of each other in a board that has more than 2 layers. You can, for instance, connect layers 1-2-3 or layer 2-3. Adding options like that will greatly increase the cost of your PCB but may be necessary to achieve a good layout.

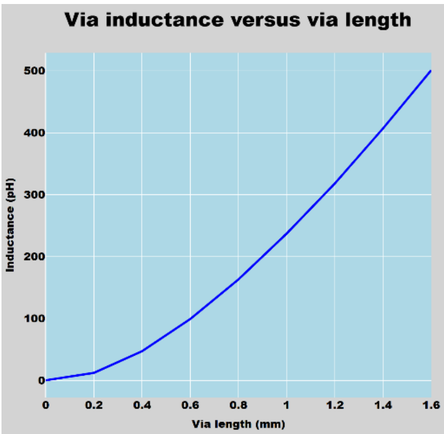
You have to know exactly what the layer heights in your stack-up are in case you need to make traces with a characteristic impedance. You should check this with your PCB manufacturer. You can basically get almost anything you want, but that may get very costly.

Below we see two stack-ups for a 2-layer and 4-layer PCB. A stack-up is a cross-section of the PCB showing all layers and their thicknesses.



Layer assignments

Once you've chosen a stack-up, you'll have to decide what layers to use for what purpose. Crucial here is that you want a ground plane as close as possible to the layer that routes the most AC signals. These can be analog signals or digital signals, anything other than DC. The reason you want the ground plane to be as close as possible is the ground impedance. Longer vias have a higher inductance, as you can see here:



A 0.38mm distance (from top copper to inner 1 in the 4-layer stack-up example) has an inductance of 45pH. If you go to the bottom layer, it is 501pH. This is more than 11 times more (In a 2 layer stack-up, this is your only option). This has a big impact on the impedance of the ground connections. You can check my YouTube videos for actual measurement examples of it. Having this in mind, it is easy to determine a good layer assignment for a 2-layer and 4-layer PCB.

2-layer assignment:

1. Top copper: components, signal traces and supply traces.
2. Bottom copper: Ground plane with as little other traces as possible

4-layer assignment (Components only on top):

1. Top copper: components, signal traces and some supply traces.
2. Inner1: Ground plane with only via holes as obstruction: NO TRACES!
3. Inner2: Supply traces and or connecting traces. Fill up the rest with ground, make sure this ground is connected to the ground of inner1 using vias.
4. Bottom copper: Supply traces and or connecting traces if needed, otherwise as much ground as possible with many via's connecting this ground plane to the ground plane of inner1.

4-layer assignment (Components on both sides):

1. Top copper: components, signal traces and supply traces.
2. Inner1: Ground plane with only via holes as obstruction: NO TRACES!
3. Inner2: Supply traces and or connecting traces. Fill up remaining space with ground
4. Bottom copper: components, signal traces and supply traces.

If you're working with RF boards, make sure there are no 'floating planes' on your board (parts of copper on the board which are not connected to anything). They can act as resonators (I also have a video on that on YouTube).

Setup your board technology

The last thing to do before starting the layout phase is to set the design rules in your PCB design software. You can get these design rules from your PCB manufacturer for their standard PCB processes. Within technical limits, you can change those rules, but that would make the PCB more expensive. So what are the most important ones?

1. Minimum trace width: 0.15mm is quite a normal value.
2. Minimum distance between traces/planes: 0.15mm is pretty standard.
3. Minimum drill size for holes, pads, and vias: 0.25 to 0.3mm is a good starting value.
4. Minimum inner and outer annular ring (this is the 'width' of the donut-shaped copper around a circular pad or via for inner and outer layers): 0.175mm is a good minimum value. So when you want a 0.3mm via, you need to add 2x0.175mm = 0.35mm to get the pad diameter, which would be 0.65mm.

There are a lot more rules which are a bit more secondary; however, if you're making a board for larger series or mass production, you should check all of them with your PCB supplier to make sure you're meeting all of them. Often they will give you feedback before starting production if something is really wrong, but you are ultimately responsible for following the process rules.

The layout phase

During the layout phase, we're placing all traces and planes to connect all the parts. It is good to use a logical order:

1. Place a ground plane over the whole surface area of your PCB on the correct layer. If you have enough layers to place a power plane on a layer, you can do that as well here.
2. Place critical signal lines: Analog, RF, and sensor inputs with a high dynamic range. Crystal traces and reset traces. These should be kept short; they're notoriously sensitive to interference. Shield these traces if needed with ground on each side. The reason you do this is to keep these as short as possible and to remind yourself during the rest of the layout phase that you should not get near these with interfering signals.
3. If you have large power consumers, place the supply lines for those to ensure those traces are wide enough. Your component placement should have solved this for the most part already; those power traces should be kept short preferably.
4. Route the different circuit blocks that you made during the component placement phase.
5. This one is critical: **Give each ground connection its own via!!** I cannot stress enough how important this is (I have a few YouTube videos demonstrating the impact of this). With a 4-layer board, this simple rule solves all grounding problems. With a 2-layer board, it is wise to make sure the via is the only path to ground (there should not be another path to ground via top copper and another via; this may give crosstalk). This rule ensures the best grounding and minimizes crosstalk. If you have a power plane, the same holds true for supply connections; give each supply connection its own via as well. Place this via as close as possible to the decoupling capacitor of the supply.

6. Route the connections between the different circuit blocks.
7. Finish any connections you've missed.

That should wrap up the layout.

PCB layout / final checklist

This is a checklist with common mistakes and easy-to-forget items when making a PCB layout.

1. Perform a DRC (Design Rule Check). This checks your board against the design rules you entered for the PCB process of your manufacturer.
2. Perform a connectivity check to make sure you have all connections.
3. Make sure you have fiducial marks. You should have three on three corners of your PCB, as far apart as possible. If you have an area with fine-pitch SMD components, you should place three extra fiducials in three corners of a 'virtual box' surrounding those fine-pitch components. This helps an assembly company with placement accuracy.
4. Add text and version info on the PCB.
5. Add layer markers on each layer. Usually, they are numbered 1-x, where x is the number of layers. Place this number in the copper of each layer, all on top of each other in one location on the PCB. This helps the PCB manufacturer identify each layer beyond reasonable doubt.
6. Make sure all components have the correct order codes if you are working with those.
7. Do you have ground connections on the PCB for measuring?
8. Did you use the smallest possible vias everywhere? This could lead to reliability issues. Only use these in fine-pitch areas where you have no choice. Larger vias also have less impedance, so they are preferred.
9. Have you named your test points? This can be very handy for verification.
10. Check the solder paste or solder cream layer. Make sure that all pads that need paste have a 'hole' in the solder paste layer. This layer is used to make an SMD stencil. Some items maybe should not have an opening in the SMD stencil, check those as well.
11. Check the copper balance. Copper balance means that you have to have roughly the same amount of copper averaged over all layers of your PCB. PCBs are produced by stacking them together and then squeezing them to 'glue' all layers together. Say you have the left half of your PCB with copper on all 4 layers and the right half without copper on all 4 layers, the layers won't bond in the right half during production. Filling up all unused space with ground and nailing those planes together with vias is a good solution. Also great for the thermal resistance of the PCB.
12. Put enough distance between components so the PCB can be assembled. If the distance is too small, that can result in problems. If you have a partner for assembly, check with them for design rules.
13. Make sure you check the DC resistance of high current traces. If you're supplying a high power digital IC like an FPGA, the voltages can be quite low while the currents can be quite high. You may create too much voltage drop for the IC to function properly.

STEP 7: PCB ASSEMBLY

Once your PCB is done, you can generate production files using your PCB design software. This means creating Gerber files for manufacturing a bare PCB and placement files and BOMs for the assembly company. The key here is to select a reliable manufacturer, which is outside the scope of this checklist. I will be posting a video in the future how to reflow SMD boards at home or in a small workshop with very cheap tools.

STEP 8: TEST AND MEASUREMENT

Once your board is back, you obviously have to test it. Test and measurement is a whole bookshelf on its own, so outside the scope of this document.

FINAL REMARKS

I hope you'll enjoy using this checklist. I'm always very curious to hear what you are struggling with as an electronics engineer and would love to help addressing those struggles with a YouTube video or maybe even a course in the future. If you have any questions or remarks please send an email to info@hans-rosenberg.com.

Best regards,

Hans Rosenberg

VERSION INFO

V2, 23-sep-2024

- Added course information
- Added notes on how to draw schematics in such a way that the chances of getting it right the first time are highest
- Added loose inputs to the schematic entry checklist
- Added paragraph on Tantalum capacitors
- Changed layer stackup picture colors to hopefully improve the image for color blindness

V3, 17-mar-2025

- Added a few items to the schematic design checklist:
    - Check component values, it is easy to forget a ‘k’ in a resistor value for instance. 10 instead of 10k.
    - Check the mounted / not mounted properties of all components so you get the right components assembled on your PCB.
    - Are all devices within their operating conditions, like maximum voltage, current or power dissipation?

V4, 26-apr-2025

- Added trace DC resistance check for the PCB layout / final checklist
- Updated intro to this document as my course is not ready for sale

V5, 10-Aug_2025

- Updated the introduction