1 What is a PCB?

What is a PCB? The PCB is at the heart of any electronic device. To explain their importance, let’s look back to the days when a television was a novelty, a fat novelty.

For a TV to show a moving picture, something needs to convert the signal in the air into an image. For this, there are a lot of electrical cables and components behind the screen. Look at this TV from 1955, what a mess!

One thing is to build the TV and connect all those cables to the right places, but imagine trying to repair something and figuring out where to start! And, trust me, most TV sets needed repairing now and then.

Instead, if you look into this TV from 1970, you can see that most of the cables were replaced by a PCBs! Now you can see how components – those round yellow and red cylinders –are now placed nicely on a brown plate – a PCB. If you needed to repair this TV set you had a much easier job than in 1955, and if needed, you could just replace the entire PCB.

So, the brown plate is a simple PCB – and instead of cables, it has thin copper conductors. This means it can do the same job of connecting components in a much smaller space, and it’s easier to handle both in manufacturing and in repair. The PCB is one big reason why our TVs got thinner, and all electronic devices got smaller. And better. And less expensive.

Now, as the TV set develops, it needs even more connections. But how can it get more conductors – when they will need to cross each other, right? In the old TV, cables cross all the time, don’t they? It *is* a problem, but let me explain:

On a regular cable, the conductors (the copper wires) are wrapped in plastic insulation to make sure that the electricity has no chance to stray from intended track. So, even if cables cross, electrical signals stay where they belong.

Well, we really don’t want to wrap all our super-thin copper conductors with plastic, so we need to find another way to keep conductors apart. One way is to have conductors on both sides of the PCB with a layer of insulation between them. Then we can have twice as many conductors without crossings. The green plate in the picture from 1970 is such a PCB - it has conductors on both sides, and in the next lesson I’ll explain more how this works.

And – just so you know – in reality, there are thousands of different components that all look quite different; these are some examples. But for now, we will use the red and yellow cylinders as symbols for any kind of component.

This picture is from an early version of an LED TV. It’s a different technology, but it still needs PCBs - you can recognize both the brown, simpler, and the green, more complex PCB.

So, to conclude:

Our customers and customers’ customers make all kinds of machinery and devices. Our job is to help them connect their components using a PCB that fits the available small space. It still has to work perfectly. Quality first. Always.

Now that you know what a PCB *is*, it’s time to explain how it *works*, and why it’s just like the London Underground.

Yes, you heard me! It’s very similar, just smaller.

2 Levels and layers

Now you know that a PCB is basically a way to organize conductors in a small space, while avoiding unwanted connections. The more connections we need, the smarter we must design the PCB.

So, what to do when we need more conductors than we have space for? We can do what the London Underground does. They have a similar problem – they move passengers, and we move electrons, but the principles are the same.

This is a map of the London Underground. With all those tracks crossing, how come they don’t crash all the time? The answer, of course, is that the tracks are built on different levels. Trains don’t crash because they don’t meet.

I mentioned that a PCB can have conductors on both sides, separated by insulation. It doesn’t stop there - we can add layer upon layer of conductors and insulation, making sure that conductors don’t meet unless we want them to.

However, just like passengers want to move between levels to change trains or get back to street level, we want the electric signals to move between layers in the PCB.

Where passengers use stairs or lifts, electrons use holes - called **vias** to move between layers. A via can connect two layers, all layers or anything in between. There will be much talking about vias throughout this course; they are key in smart PCB-designs.

Today we can make PCBs with up to 120 layers for the most complex needs. You can imagine the effort it takes to design them to make sure all electrons go where they are supposed to go, and nowhere else.

So, layers and vias make PCBs smart.

THIS IS SO EXCITING! The next lesson is about conductors and how they are different.

3 Conductors are different

Now you know that electrons can **move** in **conductors**, that they **can’t** get into **insulation**, why it’s smart to have **layers**, and that connections **between** layers are called **vias**. In this lesson, you will get a closer look at **conductors**, because conductors are **not** all alike. They serve different needs, just like trains do.

If you are going from central London to Heathrow Airport, you want a **fast** train. If you are going from Bond Street to Marble Arch (which are very close), speed doesn’t matter much – as long as there is enough **room** for everyone to get on the train.

When we transport electrons, **sometimes** speed is important, and **sometimes** it’s better to have room for many electrons moving together. For this, the **size and shape** of the conductor are important.

A conductor designed for high speed will be **straight** and have **smooth** walls, so that **nothing** slows down the electrons. On the other hand, if it has a lot of sharp **curves** and the walls are **rough**, electrons will go **slower**.

**Other** conductors are **wide** to provide room for **lots** of electrons to move at **once**. When they arrive, there is **room** for them to **land and move on**, just like a busy train station needs wider platforms for passengers to walk out of the station.

So, the **shape** and **size** of the conductor decide the potential **speed** and how many **electrons** we can transport at a given time. In PCB design, some conductors must transport signals with **almost no** power available. This means that the conductor must have very little **resistance**, so that nothing stops the electrons from doing their job.

The material **surrounding** the conductors matters **too**, just like for trains. The hyperloop train is a futuristic idea where a train would run **in a tube** to protect it from wind or rain, or anything that might limit the speed. This could **reduce** traveling time from London to Paris **from 3 hours to 20 minutes**.

The idea is futuristic for **trains**, but some modern electronics **already** work like this. Mobile transmission works like this, with conductors being protected by waterproof and noise proof materials. The new **5G net** tolerates almost no interference and signals that can go extremely fast, even with mass transportation.

So, conductors are all different. And would you believe it? Holes are not just holes. And that’s the next lesson.

4 A hole is not just a hole

A hole is not just a hole. You may remember that vias are holes between layers where electrons can move.

Looking closer, you can see that the hole is not just a hole, but the walls are plated with copper for electrons to move in.

Some of these holes need to have thick copper inside the walls to transport many electrons at the same time. It is always important that the copper thickness is even, otherwise we get bottlenecks where electrons have to slow down to go through, like passengers trying to get on to a busy escalator.

Let’s look at vias in an 8-layer PCB, seen from the side – you can see the 8 copper layers. Between them are grey layers of insulation, and the top and bottom are protected with green soldermask.

Vias come in different sizes. A via that goes through all layers, to the other side, is called a through hole. A through hole takes up a lot of space on every layer, so usually a via will connect fewer layers. These are some versions of shorter vias.

* A blind via is drilled from the surface, but not all the way through.
* A buried via is a hole that never sees the air - it is drilled between inner layers only.
* A microvia is usually between 2 layers only, and very small, with a diameter of less than zero point fifteen millimeters (here’s a toothpick to compare, it’s much wider, 2 millimeters).

You can imagine that it’s difficult to plate a 0.15 mm via with copper, evenly. So microvias are kind of V-shaped, which makes it – not easy – but possible. It’s still hard though, and we have high demands on our factories.

Microvias can go a bit deeper, but due to the slight V-shape we need them to be wider on top. Or we can stack them, or have them in stairs (called staggered via) to reach several layers. But we have to be careful; microvia can be vulnerable if we stack them on top of each other. So why not just drill all the way through? Because a through hole must be wider than a microvia or it’s impossible to plate good quality. And if it’s wider. it takes up more space. Also, a long thin hole is vulnerable – the slightest movement in one of the layers could break the plating. Remember, our PCBs must be able to function perfectly, even under quite unfriendly conditions.

One of a PCBs first challenges is to survive the manufacturing process, which is the subject of the next lesson.

5 Are PCBs printed?

Here’s a question:

Are PCBs - (LÄS SAKTA!) Printed Circuit Boards - (LÄS SOM VANLIGT!) printed? The name indicates that they are, doesn’t it? And in a way it’s true! The method we use to create conductors resembles screen printing. So, what are conductors again?

In PCBs, conductors connect components, like tiny cables.

Now back to printing. Screen printing is when you protect a paper with a screen before you apply paint. See? Everything protected stays white, so you get the pattern you want.

Now we look at the PCB. You know that the core has a copper foil on top, and this foil will become our conductors.

The PCBs design is projected on the copper. Areas that will become conductors are protected, and the rest of the copper is etched away. When the protection is removed, you can see that everything protected stays copper.

In reality, it’s a bit more complex, but the principle is like this. In every copper layer, copper that’s not needed for conductors is etched away. What’s left are very thin and very precise conductors, layer by layer.

Customers will place their components on the outer layers of PCBs, so we prepare for that. Every component has two dedicated places, called ***pads***. Pads lead electrons between the component and conductors in the PCB.

Customers use ***soldering*** to fasten components to PCBs. It works like this:

A small bead of solder paste, a small tin ball, is placed for each connection. (This is the same thing seen from the side.) Then the component is put on its pad.

When all components are in position (I’m just showing one here, but you get the idea), the whole thing goes into an oven. The solder paste melts and sticks to both the PCB and the component. Afterwards, when it cools again, it’s a solid metal connection. Now the component is securely fastened, and electrons can pass through easily.

Since customers are always trying to save space, it’s smart to use materials that have more than one function. ***Soldering*** serves both as a glue and a conductor.

Soldering components is the customer’s job, and our job is to make their job as easy and precise as possible. We know that the solder paste goes fluid in the oven, and it must be stopped from flowing all over the place. That’s why we apply a soldermask – an insulator that protects areas where we don’t want connections. Electrons are only allowed to go where we want them to.

6 What layers are made of

What are layers made of. PCBs are made in layers. In the TV from the 70’s, the brown PCB had conductors on the surface, so that’s a 1-layer PCB. The green PCB had conductors both top and bottom, so that’s a 2-layer PCB. Layers are ‘layers of conductors’ – there are also other layers, of insulation for example, but they are not counted. So, an 8-layer PCB has 8 layers of conductors. It’s very logical.

In theory, a PCB can have an unlimited number of layers, but factories have some limitations depending on their equipment. Right now, in 2022, most of the PCBs we sell are 4-22 layers. In this lesson, I’ll explain what layers are made of, and some aspects of PCB production.

Most PCBs start as a 2-layer core - insulation with copper foil. The insulation is made of **prepreg**. Prepreg is soft at first, but when **cured**, which means heated and compressed, it becomes hard as glass, and it’s now called **laminate**. Now we have a stable core to build on it.

**PREPREG**

Fibreglass pre-impregnated with epoxy; an insulating material that copper conductors can sit on.

Cores are made in laminate factories, so we buy them ready made as a copper clad material - laminate with copper foil top and bottom.

If we need more layers, we add more cores. Since prepreg is soft before it’s cured, it can be used to glue cores together. This is another example of saving space by using the material to serve two purposes. Just like soldering is both glue and conductor, prepreg is both glue and an insulation.

In the previous lesson about vias, you probably asked yourself how it’s possible to drill buried vias inside a PCB. Good question, I’m impressed you thought of that! There are two answers:

* One is magic, but that’s a different course.
* The other answer is that we drill buried vias **before** gluing all layers together.

The soft prepreg must go through the press, so that the epoxy is cured and hard before we can drill for vias. In complex PCBs we have several press cycles and sets of buried holes.

When all layers are ready - conductors, buried vias and everything - we press them all together in a multilayer press. The prepreg will soften every time it’s heated, so that epoxy fills every gap between conductors and layers. Now the entire PCB is cured and hard, and we can drill and plate blind vias and through holes from the surface.

So, does it seem difficult? Actually, it’s **very** difficult, partly because PCBs are really, really tiny. How tiny? Check out the next lesson.

7 How small is small?

How small is small? NCAB’s customers want PCBs that are small and perfect. But how small are we talking about?

This is an ordinary toothpick. And this is a 2-layer, standard 1.6 mm, PCB from the side. Can you see it?

I’ll expand them, so they are easier to see. The toothpick is 2 mm thick, but with PCBs, millimeters are too big to be practical. For measures below 1 mm, we use microns instead which are 1000 times smaller.

Here is the laminate, 1,5 mm, and the copper, 35 microns on each side. If there were inside layers in this PCB, that copper would be even thinner, down to 18 microns. And then the soldermask, 10 microns each side. So, a standard 2-layer PCB thinner than a toothpick, is 1.6 mm, but we often make them even thinner.

Everything around PCB is small. The red and yellow capacitors and resistors in the TV from the 50s, have now become so small that they can barely be seen without a magnifying glass.

**DENSITY**

How many components can fit onto this area

A key design measure for customers is the **density** of the PCB. And more components mean more function on less space.

Just look at this PCB (with components mounted) inside an iPhone; it gives you an idea of density. Each of these small components has two areas that connect pads to PCB. This PCB can only be made in very few factories globally.

Just imagine fastening the smallest components in thousands to one single PCB! That’s the customer’s job, and ***our*** job is to make the customer’s job easy as possible. So, how can we make tiny PCBs easier to handle? Well, instead of delivering PCBs in one-by-one, we can make ***panels***. A panel can be moved around as one piece so that customers don’t have to handle one PCB at a time. This helps them with efficiency and cost of assembly.

**PANELS**

Sheets of identical PCBs that make the customer’s handling more efficient.

Components are placed by robots, and this process is amazing! I strongly recommend that you watch a video (there is a link after this lesson) with a robot at work. One such robot can place 33 000 components in an hour!

So, who can supply these little miracles? Well, NCAB can, and one reason why we do it so well is how we cooperate with our factories.

Even though we don’t own any factories at NCAB, we are deeply involved in the details of manufacturing. Our Factory Management people know exactly which factories are the best at which technologies, and are often personally on site to advise factories on how to achieve the best quality. We see **everything**, and that’s what the red eye in our logo stands for.

8 So, what is a PCB again?

So, what is a PCB again? Now you know that a PCB is basically cables, nicely organized in layers. It provides a way for customers to connect their components on the smallest possible area. And it **must** work **perfectly** every time. Check our website for the latest numbers on how we deliver on time and on quality. Last time I looked in (it was December 2022) it was 92.7% on time and 99.6% on quality. So, we can still get better!

Imagine what a pain for a customer to discover a defect in a PCB ***after*** they went through all that soldering. Or after ***their*** customer installed into whatever-it-is-they-do. Or after ***consumers*** starting using it. The longer it takes for a defect to be discovered, the worse it gets. We must protect our customers from that.

It’s not enough that the PCB has survived our factory’s manufacturing process and is perfect when it leaves the factory. Every part of a PCB – tiny as it is – must also be tough enough to go through everything that happens after it’s delivered.

A PCB can end up where it’s very cold, very hot, or in the Sahara Desert where days are hot, and nights are cold. All materials change more or less with elevated temperature, so inside these very dense and complex PCBs, we must use materials that change as little as possible. Copper, fortunately, is a bit elastic so it can remain intact when the surrounding materials move a little.

Water is also a challenge. You have probably seen how water expands when it freezes. Water freezing into ice can crack mountains! And PCBs, for sure. So, we must avoid water inside PCBs, and since they are so small, even humid air contains enough water to be harmful, especially if we use materials that absorb water. And then there is vibrations, and pressure, and radiation, and crocodiles…

So, different environments call for different materials. An important part of our job at NCAB is to understand where our PCBs will end up so that we can advise our customers on the use of materials. We have also decided to go further than regulations require in some areas – for example, NCAB has decided to use a bit more copper when plating vias, to avoid them breaking. It’s playing it safe - it costs a bit more, but we think it is worth it!

Still, it’s a constant balance between quality and price. Our customers want perfection, but they can’t afford paying for a level of quality quantity? that they don’t need. We must understand their needs ***so*** well that we can advise them what balance is.

Well, this concludes this introduction to the wonderful world of PCBs. Let’s just recap some words that may have been new to you before this course.

A **conductor** is where electrons can go. **Resistance** is how hard it is for them to move.

**Insulation** is where electrons can’t go. **Prepreg** is an insulator, often used in PCBs.

A **via** is a hole for electrons to move between layers.

**Pads** are places where components are fastened. **Soldering** is **how** they are fastened.

**Density** is how **many** are fastened into a certain area.

**Panels** are how PCBs are often delivered, to make it easier for customers to solder components.

**VIDEO:** Well done, you made it to the end. Now you know more about PCBs than most people!