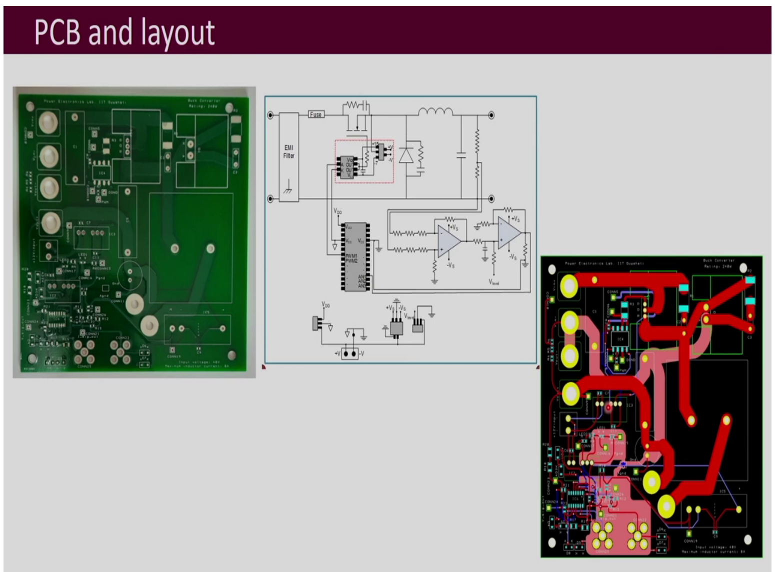
Design of Power Electronic Converters
Professor Doctor Shabari Nath
Department of Electronics and Electrical Engineering
Indian Institute of Technology, Guwahati
Module: Hardware Design
Lecture: 65
PCB - I

Welcome to the course and Design of Power Electronic Converters, we were discussing Hardware Design. Today's lecture we will discuss PCBs.

(Refer Slide Time: 00:38)



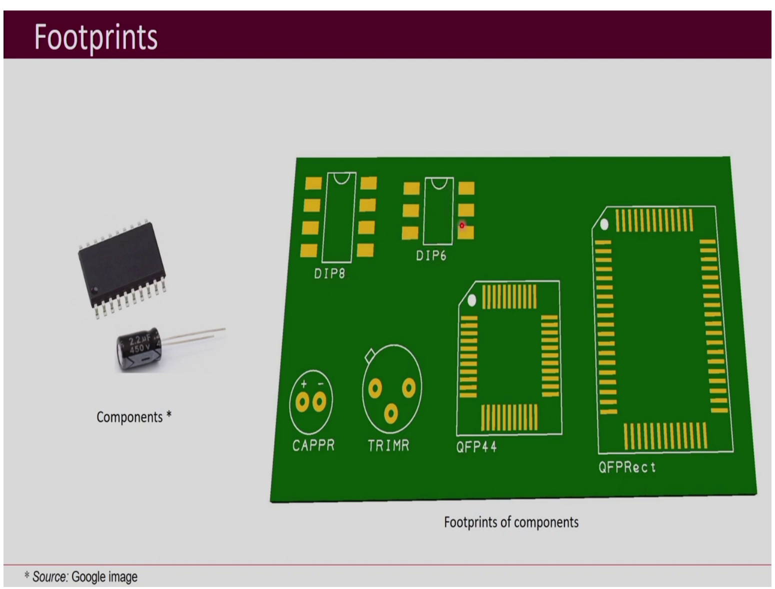
PCBs or Printed Circuit Boards must be very familiar to you, this is a picture of a PCB. Now, to make this PCB what is done is that, first you need to make a schematic of the circuit for which the PCB has to be made that not only contains the power electronic circuit, it also contains all the rest of the components that you would like to place on the same printed circuit board. For example, it will be having the drivers in it, it can also have the controller, it can also have the sensing circuit, the analog signal processing circuit, etcetera.

And then after making the schematic, this PCB is designed. This is the picture of the design of a PCB. Now, to design the PCB, what is done is that the size of the board is chosen and then the different components are placed on it at different positions. Now, how you would like to place the components inside the PCB is something very important. So, after placing the components at different positions, then the points which are connected to each other, are connected using copper traces.

Now, you can see that this is a copper trace, this is another copper trace, like this, there will be many traces. You can see that these are all different traces using which the connections are formed. Now, this placement of components and putting up the copper tracks or the traces together is called the layout of the PCB. Now, presently, we have a lot of software, which we can use for designing PCBs.

Earlier people used to do it by hand in the olden days, but now, many software are available some of which are very sophisticated software for PCB designing, where not only you can draw the schematic and make the design but also you can do the layout of the PCB. You can also simulate the circuit which means you can do circuit simulation in the same software and you can also check for the parasitics of the PCBs and signal integrity and EMI issues which we will be discussing a little later.

(Refer Slide Time: 03:24)



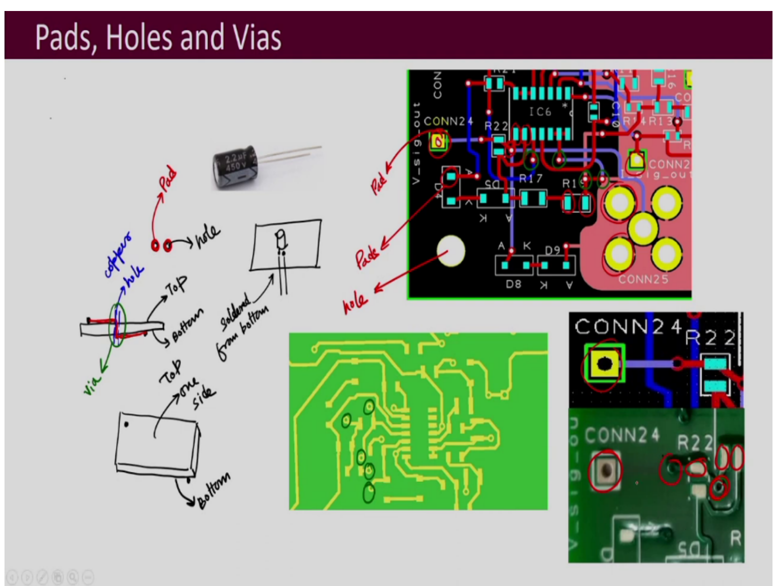
To design the PCB, the first thing that is required is the footprint of the component. So, whatever different components that you would like to place on the PCB, you have to obtain the footprint of those components according to the package. For example, if we have an SMD device like this, and let us say if it is an 8 pin SMD So, this is the footprint depending on the package, if it is a

DIP8 package or if it is 6 pin of the same package. So, something like this is the footprint. So, it is a footprint that is basically when you place the component on the PCB. So, then the points at which you are going to solder, that design or that diagram is the footprint.

Now, here like this, this is a through-hole component. So, then the footprint will have holes in it and then copper across it. So, this is a footprint of a through-hole component like a capacitor. And these are also footprints of some SMD components. So, when you choose the footprint or you make the footprint, you have to be very careful about the dimensions of the device or the component.

And if it is a component for which footprint is available. We usually the PCB designing software they have got extensive libraries, where you will get the footprints of many of the components that we use. And as I have told you before that some of these components are manufactured in different packages. So, when you choose the footprint, be careful of the package that you are choosing because different packages of the same component will have different footprints.

(Refer Slide Time: 05:31)



Now, let us say if we have this through hole device, so, when we place it on the PCB, so, the footprint will have these holes and then it also has to have a copper pour across it, or which you can call it as the pad and the black one this back circle that I had drawn is the hole. So, if this is the PCB, so the cylindrical capacitor is placed like this and then there are two holes and then it goes through it. These are these two legs and it is soldered from the bottom side.

So, here you can see that re this is the hole and then this is the pad which is shown here for one PCB. Similarly, this is another one which you can see, which is a hole in the pad. And like that, there are many of the pads and holes which you can see, they are usually for through-hole components, there will be a hole and then across it there will be a copper filling which is called as the pad through which the component is soldered.

And if we have an SMD that means, we do not need a hole. So, in that case, it will be simply a pad like this, it is shown here you can see that these are all just simply pads with no holes in it. And then the PCB can just have simply holes with no pads in it for example, if you want to screw the PCB to the enclosure that means the box of your power converter, so, they you do not need to solder you just need to fix the screws so you just need a hole.

So, this is the picture of just a simply a hole in the PCB. So, here you can see that this one is the picture in the actual PCB. This is the pad and then this is the hole. These are the PCB designs, the layouts and this is the image of the actual PCB, you can see here, this one is the pad and the hole these are just the pads.

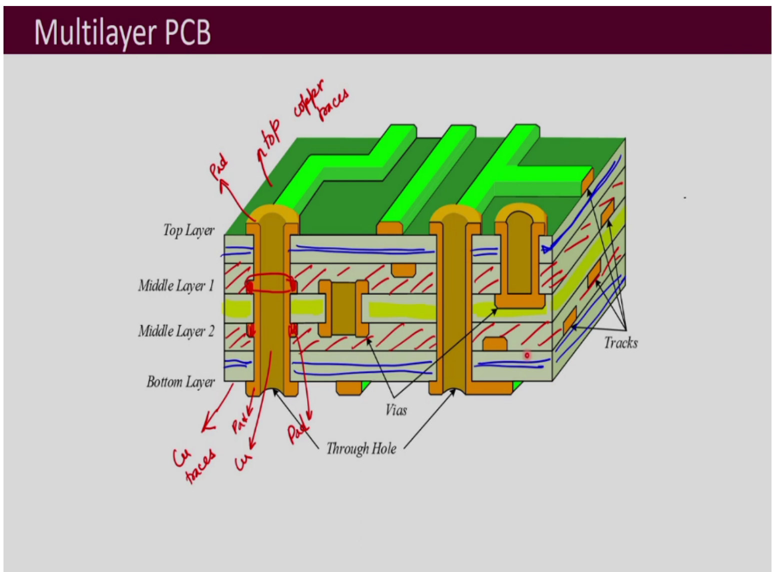
Then, if we have a two-sided PCB, two-sided PCB means on the one side, this is one side of copper and then that is the top side and of course, you will be having the bottom also. So, this side will be the bottom. Now, let us say if there is one connection point here and on the bottom side, there is another connection point. So, if we have to see the front view, let us say if we draw the front view. So, this is the PCB.

So, this is the top layer and this one is the bottom layer. Now, if there is one connection point at the top and there is another connection point at the bottom, and if we have to connect them, what is done is that the track comes from here from the top, and then it has to pass through this PCB. That means we have to have one hole here. And not only hole it also should have the copper in it, because it is a connection.

And then further this trace travels in the bottom layer and goes to the connection point. So, this thing is called as a Via. So, here you can see that this is a Via. You can see here these ones are pictures of Vi's that means the tracks are traveling from one layer to another layer and then you have a hole plus the copper filling in it or a copper pour in it. So, the difference between your through-hole pad and your via is that, in through-hole pad you have it is made for soldiering, whereas via you do not solder anything, it is just the copper drag which is passing.

So, these are also pictures of the Vias you can see here of in a PCB layout, and in this one here you can see that this one in the actual PCB picture, this is a via, these are some vias which are placed here. So, these pads hold vias and traces or tracks, these are some of the very common terms which are used while discussing PCBs.

(Refer Slide Time: 11:04)



Now, I just mentioned that there can be two sides to a PCB, the top side and the bottom side. But these days, we have multi-layer PCB, which means more than one layer of copper it is which usually means you have more than two layers of layers of copper in that PCB. So, what it looks like is shown using this diagram.

If this is let us say the top layer of the PCB that means this is the top side where you have the copper traces and then this is the bottom side or the bottom layer, where again, you have copper traces and then in between, there are middle layers. So, this one is a middle layer, which is a copper layer and this also contains tracks, copper tracks. So, this is middle layer 1. And between this top layer and middle layer 1, there is an insulating material. So, the one that I am marking with blue so is this insulating layer.

And if we have another middle layer 2, so then again in between, we have to have another insulation layer. So, this is another insulation layer between middle layer 1 and middle layer 2. And then further this middle layer 2 is again a copper layer where we will be having tracks or traces.

And then further, again, between the bottom layer and the middle layer 2, there will be an insulation layer in between as, so that is like a sandwich you have two copper layers and then there is insulation in between in there that is like a sandwich which is formed. So, multi-layer means it will have at least three layers in it, top, bottom and one middle layer and this diagram that is shown here is for the four-layer PCBs.

Now, multiple layer PCBs are used for circuits, which are larger and bigger, where the two sides are not sufficient and four-sided four-layer PCBs, six-layer PCBs or even higher than that PCBs are very much used these days. Now, here further you can see that this is the connection of a through-hole component, and this through-hole component is having connections further in the middle layer also. So, that is what you can observe here this has got a pad, this side over here, this is that pad part which is also present in the middle layer.

So, these are your pads inside the middle layer as well. And this whole thing is filled with copper. And then there is a pad on the top and the bottom layers as well. And this is a via and this one is your via between your middle layer 1 and middle layer 2 and so it has got pads in middle layer 1 and 2. It does not have any existence on the top side or the bottom side. So, when you will see from the top at the bottom that via is not accessible to you.

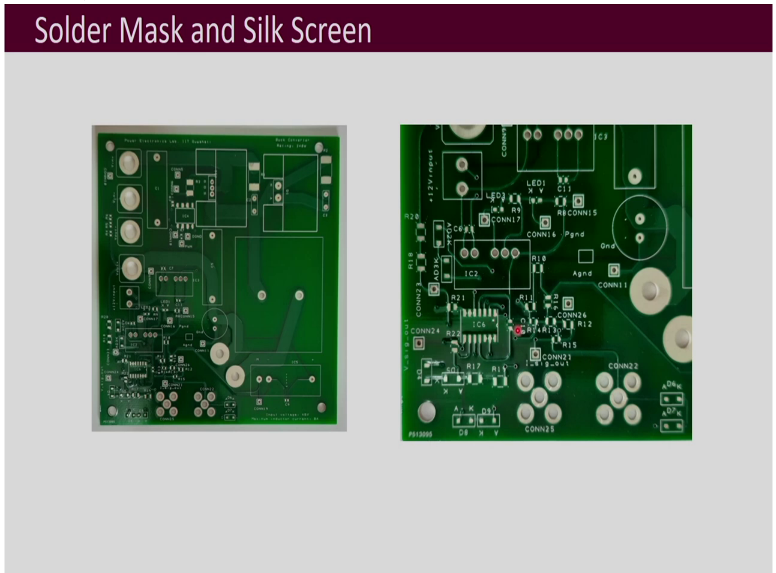
And if you see this through-hole, here this directly connects between your top and bottom, it has no connection in between means in the middle layer no connections are drawn out from this point. And here you can see this is an example of another Via where it is forming a connection between the top layer and your middle layer 1. So, further, you can see that these are the tracks in the middle layer 1 and then these are the tracks in the middle layer 2.

So, like that, from this what we understand is that. that in each of the layers, there will be several copper traces and in between them you can have vias and from the top to bottom you can have through-holes and then in that through-hole, it can have connections to the middle layer or it may not also have it. And components obviously, you are going to place either on the top or on the bottom. And the middle layers are mostly for your routing, forming connections between one point to another point and also for creating planes, which we will see a little later.

(Refer Slide Time: 16:35)

So, let us look into the PCB making process. Now, here I will look into it very, very briefly just to give you an idea of what it is. So, in PCB making we start with the copper board like something like this. And then whatever is the PCB design, that means whatever is the PCB layout that is drawn on the board, after that, apart from that layout, that means all the traces in the pads. Apart from that, the rest of the copper is etched out by some chemical process. So, this is something like this is what you get. And then after that, what is done is that all the holes are drilled in it. So, you can see that here, the holes are present, which are drilled, and also the copper traces are visible in it.

(Refer Slide Time: 17:34)



After this, or what is done is like a mask is put, now that is called as the solder mask. And that is for the protection of the copper traces. Because otherwise, if there is no mask on it, then the copper traces can get removed from wear and tear. So, here you can see that this green colour mask that you see on the PCB, is the solder mask. Now, that is why you see PCBs not having copper colour, but have different colours, like you get green colour blue, red, different colour PCBs, so that is the colour of the solder mask.

And after that, to know which component is what or to have an understanding of different pins or points in the PCB. It is usual practice to print the name of different components or to have some notation for the different pins. So, that is what you can see here like this Con2 R16 and, all these things that are printed on the PCB with this white ink, that is called the silkscreen. So, after you obtain it, that is what is finally what is your PCB is what which we use. Thank you