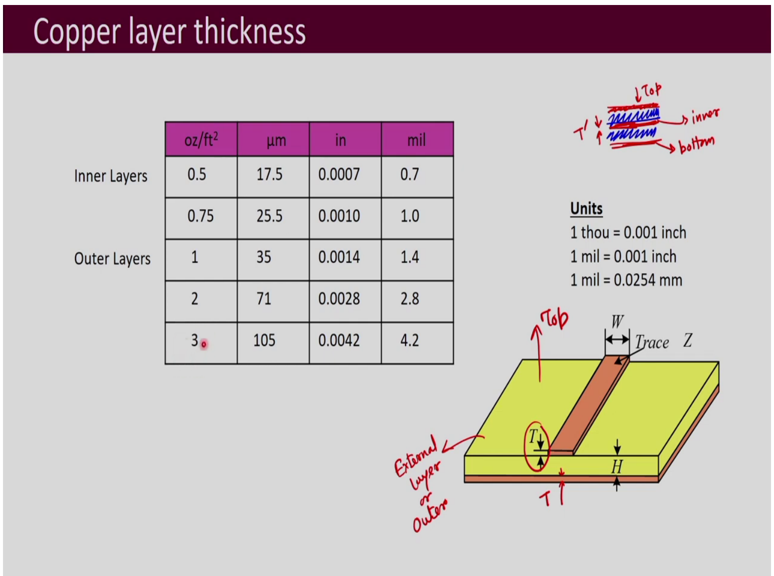
Design of Power Electronic Converters
Professor. Shabari Nath
Department of Electronics and Electrical Engineering
Indian Institute of Technology, Guwahati
Module: Hardware Design
Lecture 67
PCB - III

Welcome to the course on Design of Power Electronic Converters, we were discussing hardware design, and we had started discussing PCBs in that. We have looked into what are PCBs, the general terms, and PCB making process. And also, we had seen into the issues related with PCB of electromagnetic interferences signal integrity, I gave you a very brief idea about it. And I also showed you the various parasitics that are present inside a PCB. And that, one should be very careful in doing the layout of the PCB from those perspectives of EMI and signal integrity.

Now, let us continue further with our discussion on PCBs. The apart from the care that has to be taken in doing the layout from your perspective of stray RLC that are present in the circuit, and present in the layout of the PCB and the associated effects with it, what are the other rules that has to be followed in laying out the PCB? That we have to know.

(Refer Slide Time: 1:50)



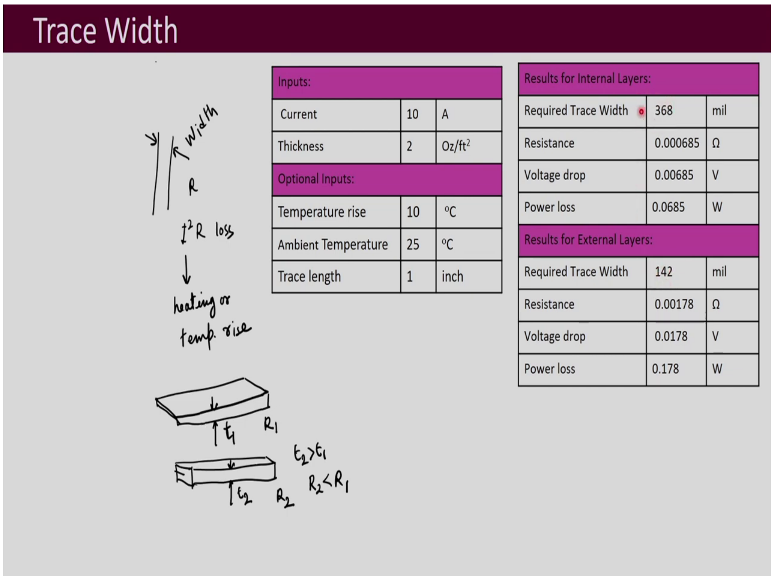
So, for that first thing we should be knowing is about the copper layer thickness. So, if we have this copper layer on one side, so, let us say this is the top side and the top side your copper layer thickness is this, that is your T. And on the bottom side also there is copper layer and that is also the thickness is T, let us say it is usually same. So, for a two-sided PCB, both the layers are can be called as the external layers, or also it is called as the outer layers.

And if we have multi-layer PCB, that means, you have a copper layer like this on the top and then in between we will be having some insulation, and then further again there will be inner layer. So, that is your inner layer. And further again, there will be insulating material. And then again, we will have the bottom layer. So, this inner layer, or whatever is its thickness, that thickness is usually different then the top layer, and the bottom layer thickness.

So, what are the usual copper layer thickness that are used? So, that is usually like this your 0.5 ounces per feet square, or it is also given in microns, or in inches, or in mils. So, this is the conversion that is written over here in the units, and ounces is actually unit of your weight. So, in terms of copper weight also the copper thickness is specified.

So, when we want to get a PCB designed, or made by one of the PCB manufacturers, you have to tell the thickness that you want, whether you want 1 ounce, or 0.5 ounce, or 2 ounce, 3 ounce, how much you want. So, for inner layers usually lesser thickness are used, so that 0.5 and 0.75 are usually used, for outer layers, 1, 2, or 3 ounces. This is what is the thickness that is commonly used. Most of the time when the PCBs are made with 1 ounce, but for power electronic PCBs where the currents may be very high, there you have to specially tell the PCB manufacture to put more copper thickness, and you can also go for your 3 ounces of copper.

(Refer Slide Time: 4:56)



Now, another important thing to note down is the trace width in your layout. Now, when we have a trace, like this, so what is its width, that decides the resistance of the trace. So, it is very simple, it is like this when you have a wire, so, depending on the cross-sectional area of the wire, your resistance of the wire is going to change. So, that is so also the same with your trace width as well.

So, the bigger the trace width is, the more it is, the lesser is going to be the resistance. And so, lesser the resistance so, lesser is going to be the I square R lost, and so, correspondingly lesser will be the heating, or the temperature rise. Temperature rise that will be associated with that trace. So, now, in the PCB traces, there will be I square R losses that are going to happen, and that will increase the temperature of the PCB.

And beyond a point, if the temperature goes, so then that may damage it. So that is why we have to maintain it in some limits and we do not cool entire PCB is generally whatever is naturally the cooling that is happening that is what which cools the PCB. So, we have to be very careful about the trace width.

Now, when we talk about the trace width, it is not just the trace width, we also have to know how much is the thickness of the copper layer that is used. And that you can know the reason very easily let us say this is the copper trace, where the thickness is let us say t1, and then you have another copper trace, where the thickness is much more let us say t2. And this t2 is greater than t1.

So, the resistance R1 for this trace and R2 for this trace. So, R2 is going to be lesser than R 1. So obviously, the second trace will have less ohmic losses as compared to your trace one, and if we have the same temperature rise, then for width trace 2 we can pass more amount of current. So, how the trace width have to be calculated? You have to first find out what is the current that the trace has to carry. And you also you should know what is your thickness of the copper layer, and then you can also have an idea of how much temperature rise you can withstand, and those simple equations you can use for calculating the trace width.

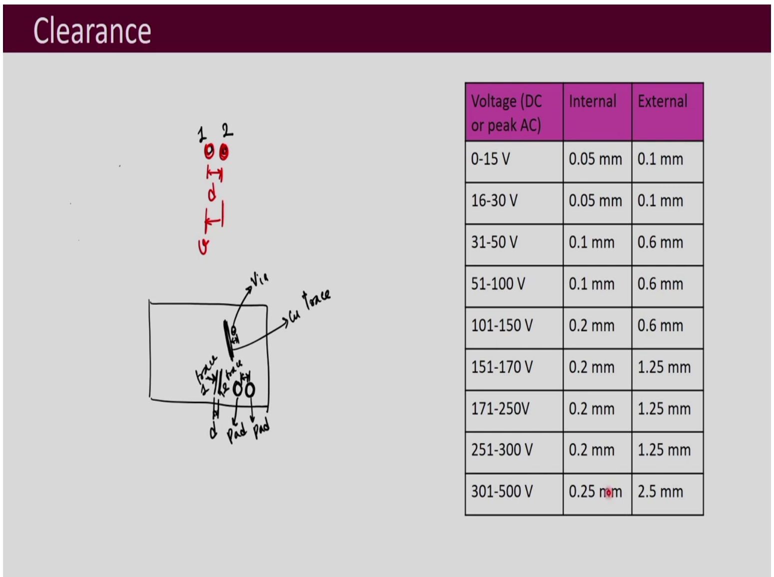
So, here an example is shown and many readymade calculators are also available you can Google, and sometimes in your design software itself, the PCB designing software itself those calculators are present, and you can just substitute your current values there, and it will give you the calculation, and how much your trace width is going to be.

So, example is shown here you can see that for a current of 10 ampere, and thickness of 2 ounces, and if you also specify the temperature rise, in the ambient temperature and the trace length, this is what you obtain for inner layers, you need 368 mils as the weight, and these will be the resistance, voltage drop, and the power loss, I square R loss that is going to happen.

And if we have if this has to pass through external layers, that means your outer layers, so this is what you are going to get. So, what do you see here to route the same thing on the outer layers, the width is much lesser. Here, it is 142 and inner layers it is 368. And one more thing that you should be careful about is its inner layers, the cooling is going to be lesser because from both sides, it has got the insulation material.

So, for the heat to be dissipated there is no lesser path, it has to pass through the insulation material, the cooling is going to be lesser than the outer layers. And so, that is why inner layers for the same amount of current to be passed, the trace width is going to be larger.

(Refer Slide Time: 10:27)



Voltage (DC or peak AC)	Internal	External
0-15 V	0.05 mm	0.1 mm
16-30 V	0.05 mm	0.1 mm
31-50 V	0.1 mm	0.6 mm
51-100 V	0.1 mm	0.6 mm
101-150 V	0.2 mm	0.6 mm
151-170 V	0.2 mm	1.25 mm
171-250V	0.2 mm	1.25 mm
251-300 V	0.2 mm	1.25 mm
301-500 V	0.25 mm	2.5 mm

Then, one more important thing which you should be careful about while doing the layout is the clearance between any two points. So, when we have let us say two pins, this is pin 1, and pin 2, and there will be a copper around it. So, how close these two points can be, what could be the distance between these, that is the clearance. Now, how much it could be that depends on the voltage.

So, let us say here in between with respect to this point 2, this voltage is your v voltage, v volts. That can be 10 volt, 15 volt, 20, 30. How much the voltage is going to be? So, that decides this distance, d. Greater the voltage, greater has to be the clearance, the distance between these two points. And, what is the reason behind it? That simply that if you make it too close, then there can be a breakdown of between those two points.

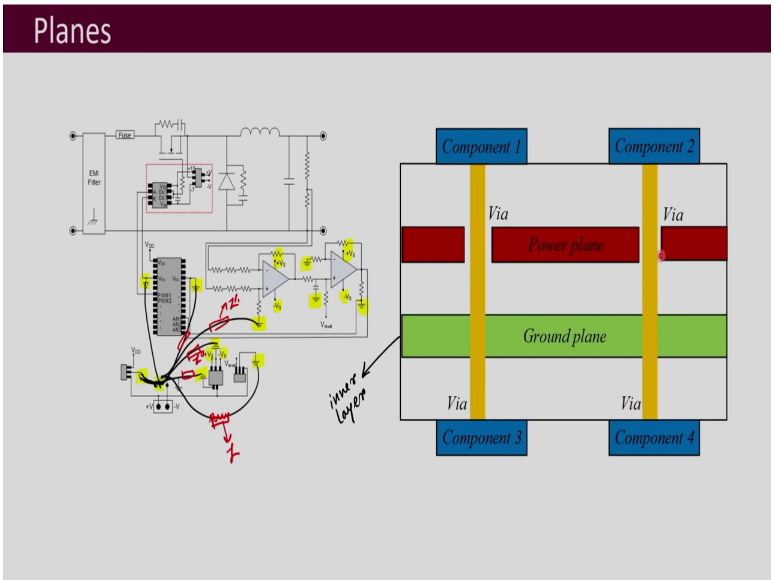
Now, which two points are we talking about in the PCB? This could be any two points which are close to each other. That means, if this is a PCB, if you have via, let us say here, so, this is a via, let us say. And if you have a trace, copper trace, which is near to it, so, in between that also the clearance you have to keep, if you have a pad and there is another pad adjacent to it. So, in between them also that you have to be careful about the clearance, and in between two traces also, if there is so, this is trace 1, and this is trace 2 in between them also whatever is this distance that you have to be careful about the clearance.

So, between any two points in the PCB, close points your which has got copper in it, you have to see whether the clearance is accordance to what minimum it should be, according to the what is the voltage between those two points. So, here it is shown one table is shown. So, if we have a inner layers, so for that the clearance is different, and for external layers, the clearance is going to be different. So, from 0 to 15 volt, how much should be the clearance? And 16 to 30 volt, what should be the clearance? So, like that this table is given, and you can note down from here, and you can have checks in your PCB, whether you are following or not.

So, most of the PCB designing software they come with your design rule check, which is called as the DRC check, where you can actually set these, that this point and I mean this trace is carrying this much voltage, and this much current, and then it will automatically check it whether any point you are violating the clearance, or how much should be the trace width it will whether it is according to that or not those things can be set automatically in the software these days.

And then there is also electrical rule check means if you have accidentally shorted some point, so electrical rule check in the schematic and also in the PCB, the software can run and will tell you the errors in your PCB designing.

(Refer Slide Time: 14:39)



Now, you should be also familiar with planes in the PCB. So, this plane concept is more relevant when we want to do routing of the grounds, or of the voltage supply pins. Here, if we see that if the example of schematic that we have chosen to explain the different terms related with your hardware designing, you can see here that there are various grounds, you can see that this is one analog ground, another ground, another ground, and this is another ground point, another ground points, so this another digital ground points.

So, there are so, many ground connection points which are present in this, and this is a very very simple schematic, if you have practical converter then yours, usually your schematic is much more involved it has many more components. So, many of the components are then connected to the ground. So, if we have to form the ground connection then one is that you do it with traces that means, let us say here, this one trace is placed in another trace is placed like this, and then another one coming here, and then here like this, you can do a connection using copper traces, but that means what?

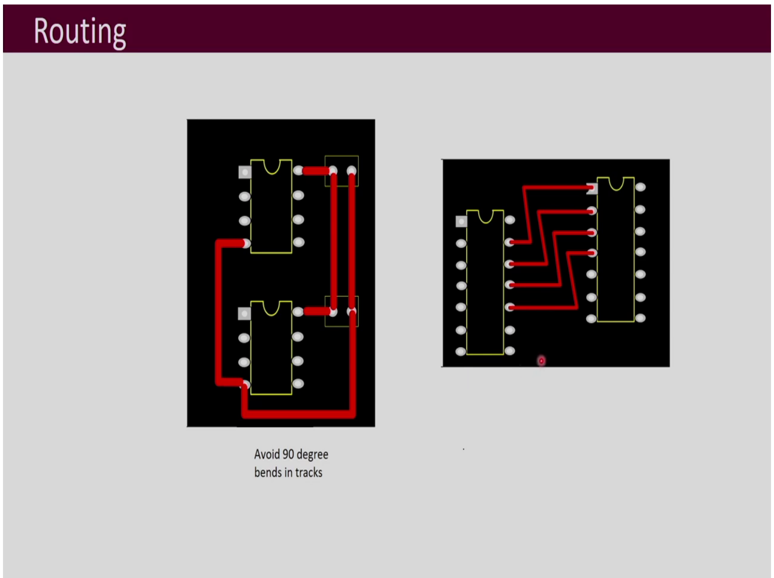
So many traces have to be laid out, further from the last lecture, we saw that each one of these will have its impedance with it, and all these impedances will be different. So, each will be having its own impedance. And so, that will lead to the problems that we saw in the last lecture. So, we would like to avoid that, it is better to have the ground laid out in a way where the resistance and the impedances are minimum, and we know that the more the width is the lesser is the resistance.

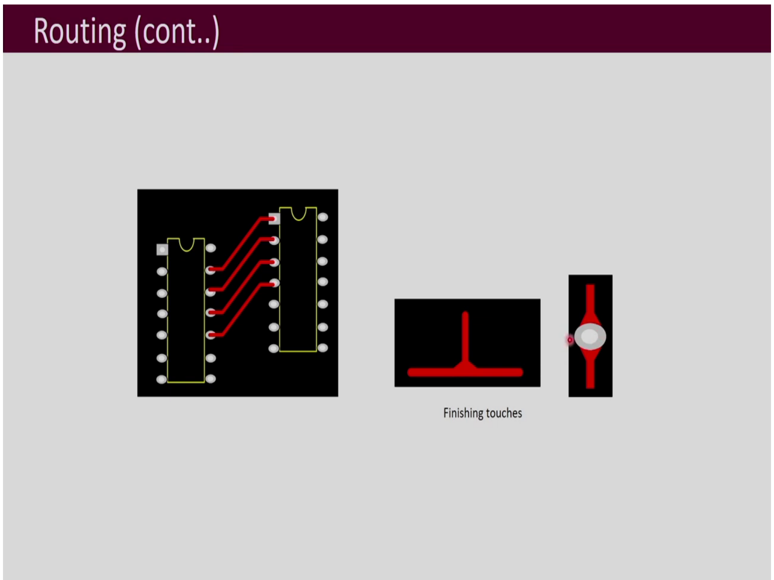
So, we can have entire layer of copper dedicated for ground. So, that is called as the ground plane. So, it is like this inner layer. So, this is one inner layer of copper, which is dedicated for ground connection that means no other traces are laid out in that plane. And if we have four components, let us say, and they have to be connected to the ground, so, using vias, you can directly connect it to the ground plane. So, this is the front view, which is shown here. And similarly, we can do this for your power plane also, because many times the voltage also has to be supplied to many ICs, or components.

For example, here you can see that this is your V s plus V s here. So, two of them are using this V s supply, and minus V s supply. So, like that, if you have many, many miscellaneous components in it many of them have to be given the power supply, the voltage has to be applied to them, the source voltage.

And then, that also can be done easily by forming a plane and then like this, where like this via the component can be directly connected to that power plane. So, this power plane and ground plane is very much used where circuits where there are many, many components in them to reduce the impedance and to simplify the routing also.

(Refer Slide Time: 18:44)

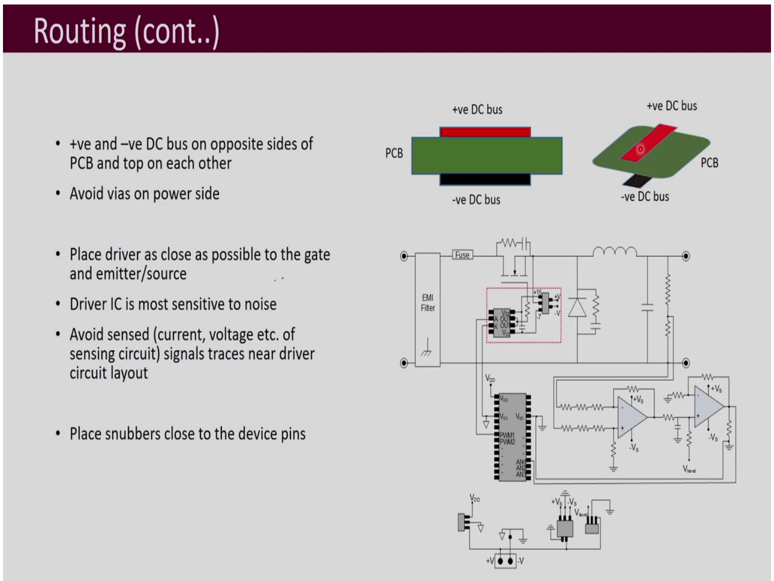




Then when you route it, you should be careful that you do not do these kind of 90 degree angles. And the reasons are there if you have this sharp corners so that are undesirable in the PCB routing, then routing should be done in uniform way and not like this. You can see that is a non uniform way, it is not a good routing practice, doing routing in this manner. Whereas, you can do it like this in a uniform way, where having 45 degree bends, that is more preferable.

And further 45 degree bends can also be here like this here you can see that finishing touches are there, this is one trace, and this another trace coming emerging from there. So there also instead of having 90 degree you can do this and create this 45 degree bends here, and similarly if you are taking our traces from this pad, from the circular bad there also you can create these kinds of 45 degree bends.

(Refer Slide Time: 19:55)



Now, some other routing tips that you should be knowing are that, let us say if we have a DC bus, positive DC bus, and a negative DC bus. Now, we know that that there is a stray capacitance here, but that stray capacitance is not undesirable where if we place a positive DC bus and negative DC bus on top of each other like this, because then that C is beneficial over there.

So, this is sometimes done, your positive with DC bus and negative DC bus, they are put one on laid out one top of each other. And vias on the power side, this means your power electronics side it is preferable to avoid the vias over there, because inner layer are not capable of carrying too much current, and usually on the power electronics side the currents are higher, several amps to 10, 20, 30, 40 amps of current may be flowing, and that much amount of current in the inner layers and through the via will create lot of heating. So, it is a general advice to avoid vias in the power electronics side layout of the PCBs.

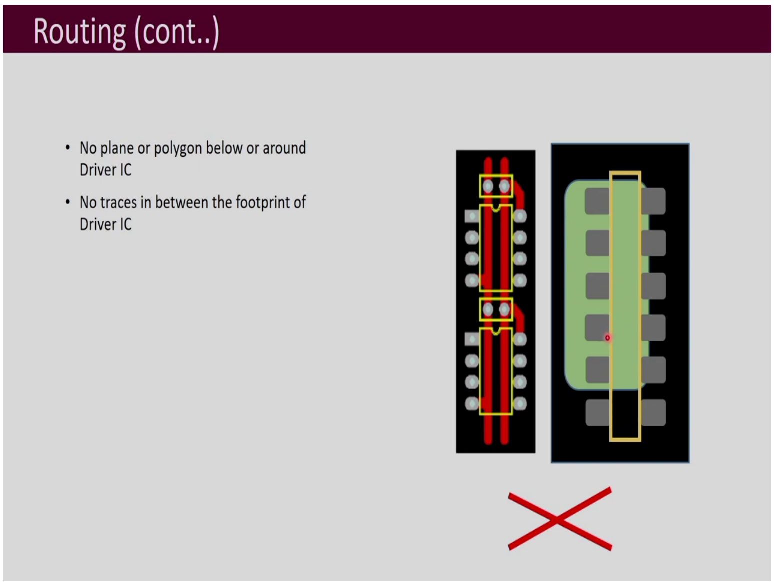
And very much care has to be taken when you lay out the driver circuit, driver is the part which is most susceptible to noise, and we have seen before the drivers varies very quickly, turn on and turn offs are happening, and the reference of the drivers circuit is source or the emitter of the transistor which is your most of the time floating. So, it is a floating circuit, and it is very, very susceptible to noise and electromagnetic interference. So, a lot of care has to be taken while you route the driver circuit.

And we should place it as close as possible to the gate, emitter, or the gate to source region. Because by now, you understand that from last lecture the moment you have placed it, you have routed using the traces you have introduced some parasitics also. So, the driver circuit is not exactly the way you see in the schematic, and drive's performance your turn on turn off all the performance will get affected by the way you have laid out. So, we have to place very close to the transistor, and the routing has to be done very carefully.

Similarly, your snubbers, they also affect the turn on and turn off process, and if we lay them if we place them farther from the device in the layout, then that also is going to affect the performance because of the parasitics that are going to get added. So, this snubbers also should be placed as close as possible to the device beds.

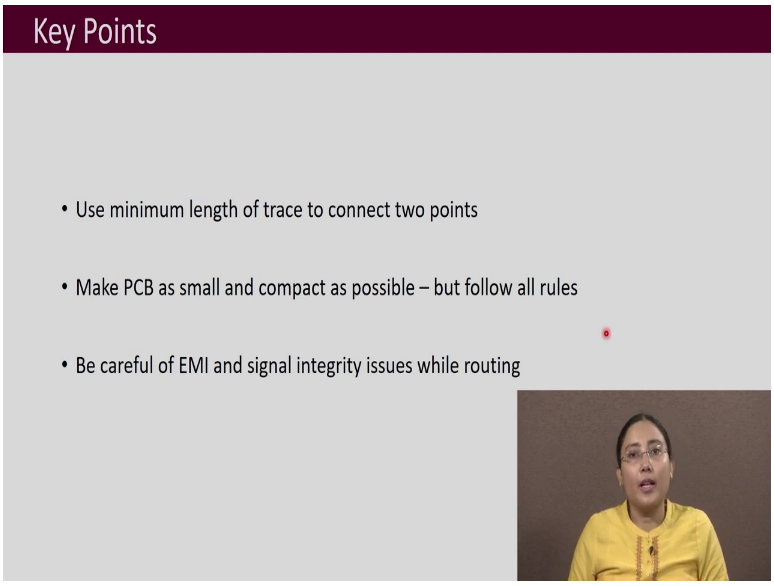
And, since the driver is very sensitive, avoid your sensed signals which are carrying like your analog signals, or sensed signals to cross the driver region, it should preferably not cross the driver section, although your currents, voltages in different sensed signal, those traces which are going to carry the sensed signal they should be away from the driver area.

(Refer Slide Time: 23:47)



And one more thing which you should know is that when you do the driver routing, it is preferable not to put traces inside the driver's IC area, like this kind of routing is not preferred for driver circuits, or placement of a polygon, or a plane below the driver area. Driver has got its own reference, the emitter or the source. Do not create a plane for that. For other types of circuits like your analog circuit, traditional circuit are usual to create planes, or polygons for the grounds and the references but for the driver it is not desirable.

(Refer Slide Time: 24:41)



So, the key points of this lecture are that, that use minimum length of the trace to connect between two points. Make a PCB as small and compact as possible. While following all the rules, you have to use the correct trace width, if do use the correct clearances, I mean they have to be greater than what the minimum requirement is, and further all the routing rules that are there, and taking care of the signal integrity and EMI issues. You have to design the PCB, and make it as small and compact as possible, but follow the rules. Thank you.