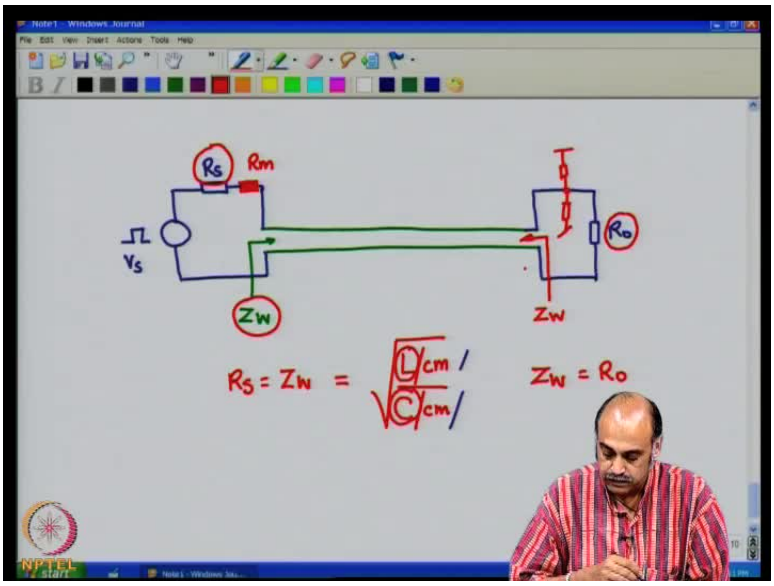
An Introduction to Electronics Systems Packaging

Prof. L. Umanand
Department of Electronic Systems Engineering
Indian Institute of Science, Bangalore

Module No. # 04
Lecture No. # 18
Electrical Issues – IV
Interconnection

We continue from where we left off in the previous class. In the previous class, we had been discussing about reflections and of course the other parasitic elements RLN and C. We will try to see some methods of how we can mitigate the problems that arise out of these problems, these issues that we had been discussing so far.

(Refer Slide Time: 01:00)



The problem of reflection, as we saw in the last class, which arose, because you had two tracks and high speed digital networks. These tracks become transmission lines and at one end of the transmission line, we have the source side impedances or the source impedance R_s and on the other side of the transmission line you have the load impedance termination R_0.

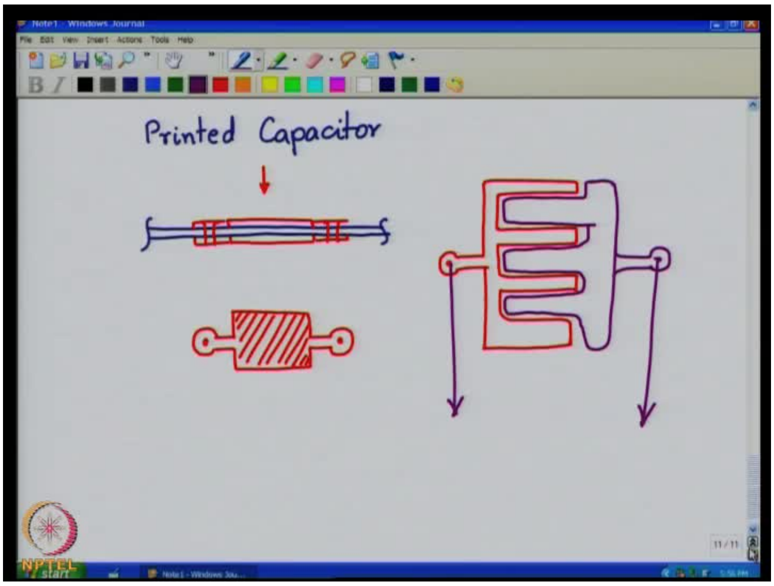
If the source impedance and the wave impedance are not matched, then you can have impedance at the source boundary and if the wave impedance as seen from the output side and R_0 are not matched then you can have reflection at the output boundary.

The solution to this, of course, is how R_s equal to Z_w or Z_w equals R_0 on the side and we have seen in the last class, instead of a Z_w we have used Z_0; it is common to use at $_0$ to denote the characteristic impedance of the transmission line.

Any way, the wave impedance Z_w should be made equal to R_s on this side, Z_w made be equal to R_0 on this side, and what is Z_w equal to? Nothing but root of inductance per unit length by capacitance per unit length. So, if we need to decrease the wave impedance of the tracks, we need to increase the capacitance. That is, you need to make the lines wider, so that you can reduce the wave impedance and then bring about a matching between the source side and the transmission line characteristic impedance; likewise, the matching between the load side and the track.

In case that you may not be able to manage with just the track widths and spacing, you may have to include an extra component here to match the source side and the wave impedance, and probably even an extra termination to match the load side with the transmission line or the tracks. Now, one could also change the values of L and Cs to bring about change in the character of the transmission line such that you bring about a matching of the impedances.

(Refer Slide Time: 05:14)



So, on the tracks, apart from putting external components, one could also make printed capacitors and printed inductors. So, if you say printed capacitor, they are made by the tracks themselves.

One of the simpler ones is something like this. Let us say you have a laminate. On the laminate, on one side you have a rectangular piece of area and on the other side bottom side another rectangle piece of area.

And if I look from the top, it would appear something like this - the conductor portion and you can bring it out to a pad like this. This could act as a capacitance and to that capacitance; you could connect external capacitance in parallel at these pads to achieve whatever capacitances that you would need.

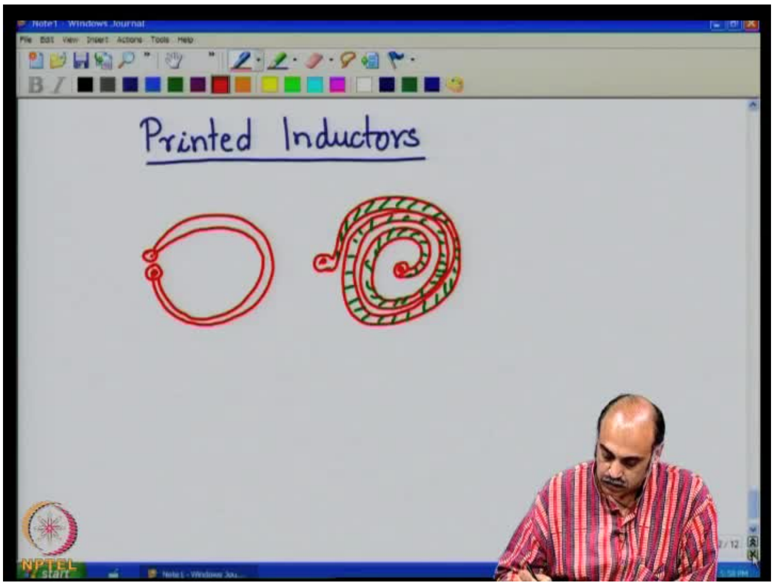
Now, use of printed capacitors whose advantages are they are very fast capacitors and they have the least series inductance equivalence series inductance of capacitors is the least and their drift with temperature is very low compared to external capacitor capacitors that we use. This is one way of printing a capacitor on the printed wiring board itself. Another topology would be something like this. One plate of the capacitor is something like this - shaped like a comb and two combs within each other form a form quite a good capacitance.

So, this would form another capacitance and you see that the contact point is all along this length. The dielectric between this comb, the right side comb and the left side comb is the one which is responsible for the capacitance between these two sets of conductors.

So, one can print something like this and then probably measure the capacitance and make charts out of it for a meter length or for the length of the combs or the width of the comb, the distance between the comb, teeth, the developer of the printed wiring board should start making charts that will aid him in future boards.

Now, once he makes the printed capacitor he will not know the value of the capacitance. So, he will have to measure the capacitance by using these terminals. These terminals should go into a bridge - an impedance bridge - and using an impedance bridge one can accurately determine the value of the capacitance and use that value for fine tuning the circuit parameters like the impedance or impedance matching.

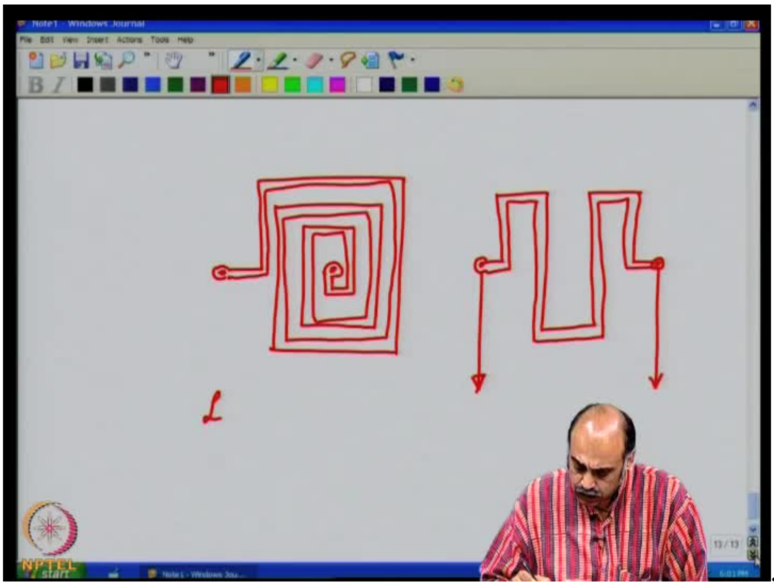
(Refer Slide Time: 10:16)



Just like the capacitance, you could also have printed inductors. Printed inductors are also quiet common. You could use few topologies. You have a pad here and then you have a circle another pad so this kind of circle or loop is equivalent to one winding inductor.

You could have multiple winding inductors by having a slightly modifying topology like a spiral. So this kind of a spiral would give you an inductor. You should understand that this spiral will give you multiple winding inductors and the dielectric forms the core.

(Refer Slide Time: 12:12)

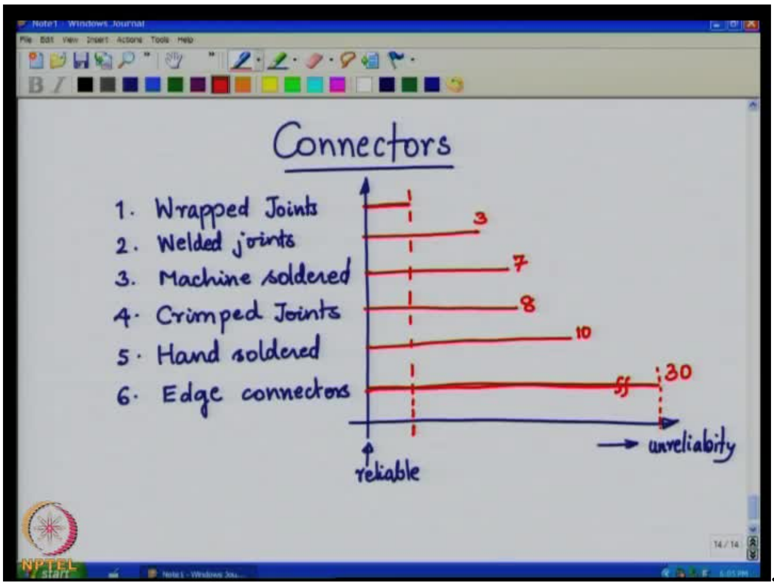


Instead of spiral, one could also use a kind of a squarish spiral like this. This kind of spiral is also, this is also a possibility to get a printed inductor with many windings. I will just write it as a simple wave like that. That is more like it. So this can also be used as inductor structures.

So like this, you can make values of the inductors and then give these terminating pads to the impedance bridge and measure the value of the inductance. That would give you the inductance probably per unit length or per unit ton for a particular type of PCB laminate and the developer should start making charts out of it and then over a period of time he can have nomographs to quickly design the artwork for the printed wiring board.

Now, these printed inductors and capacitors can be used at the ends of the tracks to adjust the values of the wave impedances in order to avoid the reflection problems. This is one issue. Another problem that you would see in a printed circuit board, when many printed circuit boards are put together to form a bigger system is that of connectors.

(Refer Slide Time: 14:36)

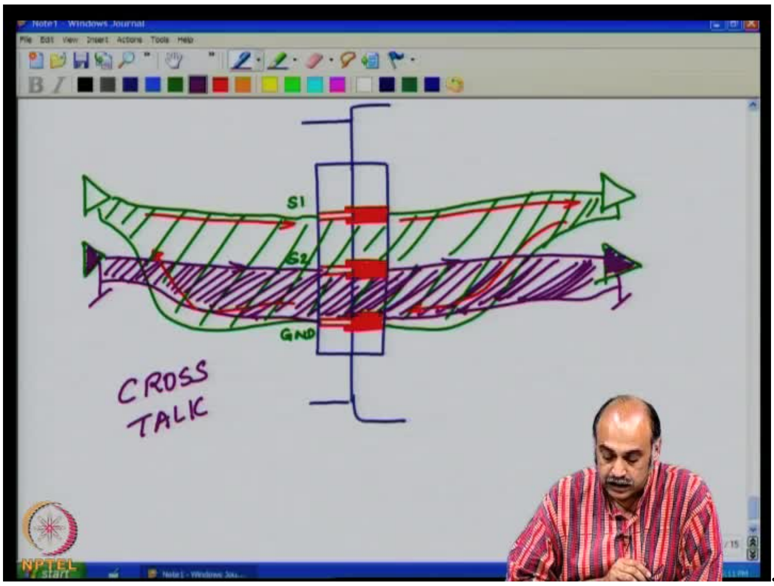


Connectors are sometimes from the reliability point of view, the weakest link reliability wise so that is what the Mills standard would say. There have been various connectors if you put them on a list you would have the wrapped joints which is supposed to be the most reliability wise you have the welded joints, the machine soldered joints, you have a the crimped joints, you have the hand soldered joints, by hand soldered I mean hand soldered by a professional and then you have the multi pin edge connectors.

So you have various types of these connectors which are used for interfacing the circuits on one printed wiring board to the external world. If I have to plot on the scale unreliability, so on this side, very reliable; this is the reliable end and reliability decreases or unreliability increases.

Now, if you look at these connectors, the common connectors that you would encounter in almost all the PCBs and look at what is the reliability relatively. The wrapped joint is the one which is supposed to be the most reliable. If you say that is the reference and that say you give a value 1 very reliable then the edge connectors or the ones which are least reliable from the reliability point of view and this we give a value 30. The welded joints are normally given a value 3. These are not to scale. The machine soldered joints a value 7, crimped joints more or less like the machine soldered joints and then you have the hand soldered joints 10. So, you see that the various connectors have various reliability associated. Keep that in mind while choosing the connectors for your printed wiring boards.

(Refer Slide Time: 18:23)



After having chosen the connectors, electrically connectors also pose different problem and that is the problem related to the inductance the mutual inductance issues.

There are two major problems, let us say you have a connector like this. This is on one board. Left side and the right side is on another board so two boards are coming together. With these type of connectors, let us say we have this is one connectors male and female and then you have let us say one more such connectors and let us say you have a male and female of those connectors they are connected together here and I will take a third one.

The reason that I am taking these three pins, you should note that a connector will have many more pins multiple pins but, I am taking three pins which are representative and let us say this third pin is the ground pin and then this is signal pin one this is signal pin two.

Now, you have some circuit tree on this side and then you have some loads on this side. I am just showing it as blocks. These could be digital loads and digital sources on this side, digital loads or analogue sources, analogue loads both are possible.

When you are talking of high speed digital circuits, let us say the drivers on this side source you have a track which comes and connects here and this goes track goes and connects to the input of the buffer stage which is in the another board.

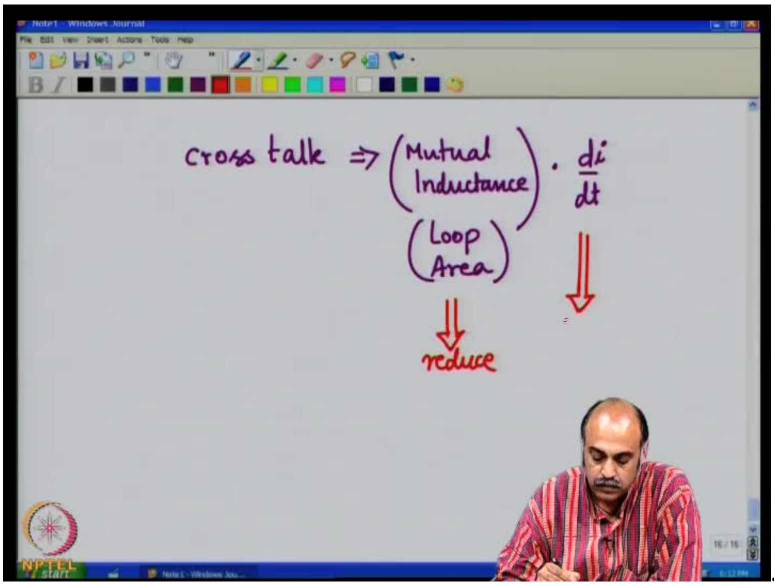
Now, from here you will have a return. The return will go to the ground pin and this ground pin will return back here so normally. That is how the many of the circuits will get interconnected you have a signal going from one board to another board and then there has to be a return path for the current.

So if you look at the current, the current goes like this, goes through the connector and then you have the return path, so on like this. If you look at this connector, this current path is a loop and this loop as we discussed earlier can induce flux to the flow of this currents and this flux induced flux can link with other loops.

For example, let us say you have another driver here as I have shown. Let us say this is the other driver. Let me show it in another color and this is the receiver corresponding to that driver. Now, let me remove this here and draw the track for this driver. It goes through like this, goes to this input and then from this ground it will go into this pin and then from this pin to this ground. So you have the forward current and the return current.

So if I take the first circuit, the loop area is so much. This would be the loop area. If you take the second, the loop area is this portion; so you see that quiet a significant portion of the loop areas are common, so which means a current flowing through the violet loop can interact with the green loop and the current flowing through the green loop can induce an e m f on the other loop and this can result in what is called as Crosstalk. So therefore, we need to be very careful when we are laying out and routing the lines through the connectors.

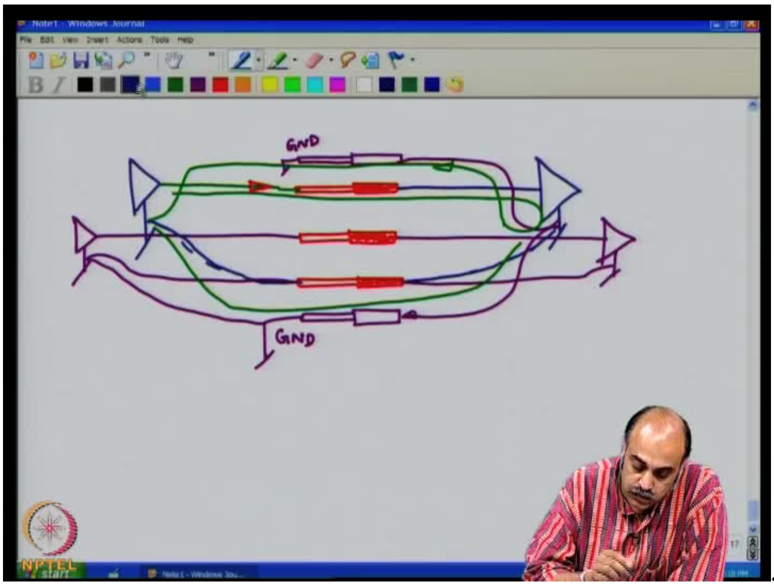
(Refer Slide Time: 24:27)



If you take the mutual inductance, mutual inductance has two components. The cross talk is basically is a function of the mutual inductance and the important part here is, d_i by d_t, the rate at which the current flow; the rise time of the currents.

So, the rise time of the currents is a major culprit. If you are talking of very slow signals then cross talk is not an issue but, when you talk of fast rise time signals, then this can be pretty high coupled with the mutual inductances of the loops. This is directly related with the loop area so these two together can cause cross talk.

(Refer Slide Time: 25:52)



So, how do you reduce cross talk? Two possibilities you can bring down this (Refer Slide Time: 25:33) or reduce this. Let us say for example, you have one connection in this fashion and you have a driver circuit which is connected like that and it goes on to the receiver side circuit. Of course, you have a ground and there is a ground return path somewhere.

Now, the current i, which is being driven from the source side into the load side and then the d_i by d_t of d_i by $d_{t.}$ of this, is one major culprit. If you can reduce this d_i by d_t, then the cross talk can significantly be reduced.

If you do not need such a high rise time then one way is to put an RC in series with the circuit in this fashion. Let us say what you are going to put here is something like this.

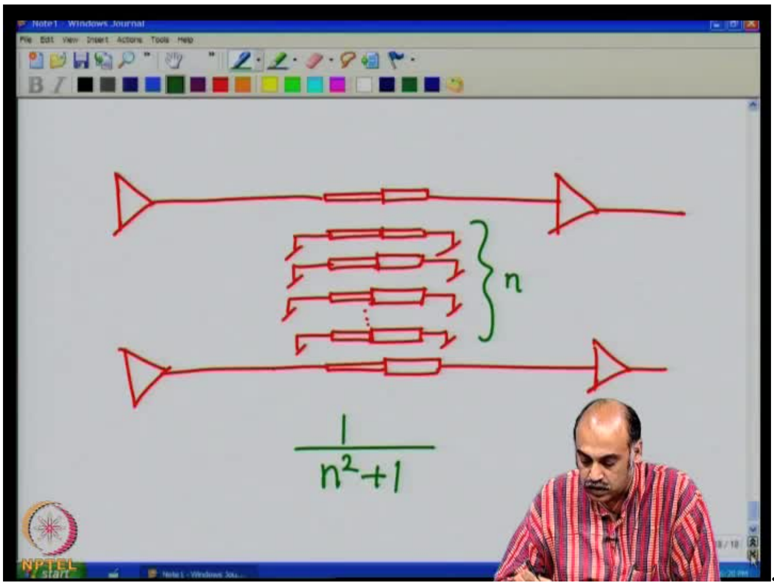
Then any rise time here is smoothened out at this point and from this point onwards, you see that the current in this portion of the loop the d_i by d_t is very much reduced compared to what would have been if you had not put this is one way. This is a Brute force method of doing that. But, this is going to reduce the bandwidth of your high speed circuit but, however keep in mind that you can do it this way and when you are going about it be careful do not put the capacitor in this side because if you put the capacitor on to this side then you are still having a pretty high loop area not only that then your passing the high d_i by d_t currents through the connector and connectors can act as an antenna and radiate e m i induce electromagnetic interference to other circuits so you have to ensure that you put the capacitor on the source side itself.

This is one aspect of improving the cross talk by reducing d_i by d_t. There is another way that is by interposing ground lines. Let us say you have you this (Refer Time: 29:00) let us say you have this ground connector and the return path is coming through here like this. Now, this current let us say for example, the current that is flowing here is split into two parts and that two parts is like this. Let us say like in the previous case we had one more signal which is coming from another drive going to another loop then coming through the same return path.

So, this is the scenario that we had now. Let us interpose two more ground lines. Let us say one ground line here probably another ground line here so this also ground meaning new grounds we connect it like that so what happens in the return path this line is split here and here so what returns through this is lesser much lesser than if it was a single ground. So in the grounds, the current gets split and then each of the loop so if you look at this loop if you take this loop goes through like that and the one return path is through this to this ground this is the return path and the other return path is through this ground let us say and back here.

Now, what happens is that through each of these ground only half of the or part of the current flows through and the strength of the magnetic signal is lesser and as in on one side it is clockwise and another side anti clock wise you have some cancelling effect.

(Refer Slide Time: 32:00)

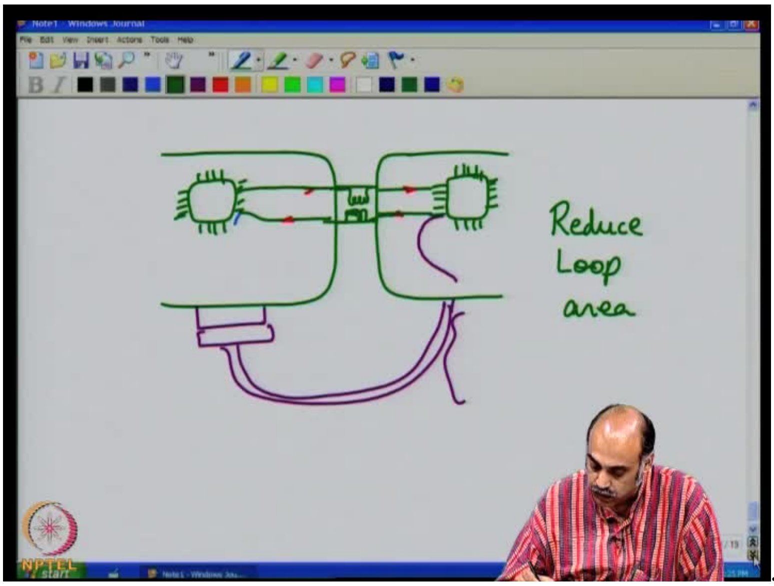


This is one way of putting grounds on the outer side of the connectors which are supposed to have cross talk. Another way is to interpose between the two signal lines, you have ground connectors in between as many ground connectors as possible more the better but, let us say you have many ground connectors like this ground on this side, ground on this side so on both sides you have grounds like this.

Now, this will increase the impedance between these two lines and as a result the cross talk will come down between these two lines will come down drastically and that is that will come down by a factor of 1 by n square plus 1 where n is the number of ground lines which you have interposed in between the two signal lines that matter.

So, then what are cross talk? Cross talk would have been there if there were no ground lines that divided by n square plus 1 will be cross talk with the ground lines. So this way interposing ground lines at the connector points or also in between track routings will significantly reduce cross talks in high speed circuits.

(Refer Slide Time: 34:05)



Another important point that needs to be addressed here at this point is the issue of series inductance effect. If you are having two boards and you have let us say some high speed circuit here and may be some high speed circuit here and you have a connector; now, you have the line going from here to here and the return line which we expect it to go from here to here which means that you have a path with goes like this and the return path. This forms a loop.

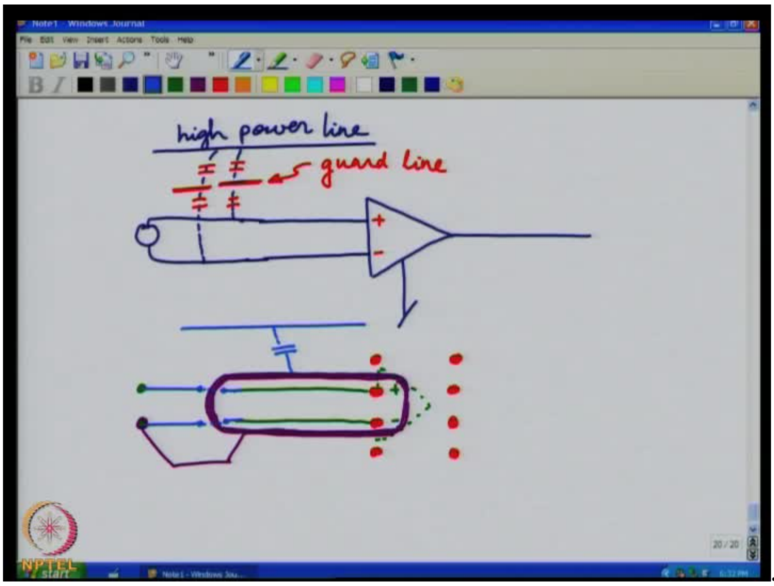
Now, this loop can radiate and link with other loops. This is a small area; however the problems occur if I am having a different connector. Let us say there is another connector at some other part of the board and to which is connected this board through a cable somewhere which may come through this or may be from some other portion of the board.

There can be a path through this. Let me show it in a different color. There can be a path from here; return path which goes through here like this and returns to a ground here. Now, this is a pretty big loop area and even a small current which flows through this undesired path can cause havoc to other circuits and may also not pass the electromagnetic e m i specifications So for such cases it may be desirable that we reduce the impedance in this path and increase the impedance in this path.

So, what we can do. Let me clear up some portions. We could have a common mode choke, so that a common mode choke where for these two lines together forward and the return they cancel each other. This inductance does not act as an impedance they forms an impedance for the line which for the current which does not return through this but, returns through another path.

Therefore, the impedance of all other external paths will increase and the impedance for the local path is reduced by putting this common mode choke and thereby you try to retain most of the return path current here and reduce the loop area. So the idea is reduce loop areas in your track routing; so when you root your track see that to you reduce the loop area. Keep the loop area to a minimum to ensure less problems with electromagnetic interference.

(Refer Slide Time: 38:27)



Next, one more issue related to analogue circuits that of capacitive coupling which you need to be careful. Let us say you have an OP-AMP amp kind of a circuit, an analogue amplifier, you have an input and the output, and you have a ground here and you are getting a source which is at a distance probably or a high impedance source.

This could be high impedance source or from a far distance like a thermo couple. Let us say you have something like this and then you are making a differential amplifier so when you have such a differential amplifier you need to see that there are no common mode voltages. Common mode voltages can be there; these common mode voltages by differential amplifiers will get differenced out and then it is not reflected in the output.

However this common mode voltages can cause a problem and deteriorate the performance if the impedance seen by this line and impedance seen by this line are different which could happen if there is a difference in coupling between this and this and this and this [Refer Time: 40:00).

For example let us say you have a high power line; this is a high power line and this will have a capacitive coupling to this. This is also a track; so from this track to this track you could have on more capacitive coupling. Now, if these two capacitive couplings are different then that could give rise to different signals here and deteriorate the performance of the output. So it is important to see that we increase the impedance of this line that is a positive line and the negative line to any external line, the impedance to be as high as possible or increase it.

So, one way of increasing the impedance between the line that matters and any external line is to see that we are able to split the capacitance. Let us say into two capacitances for example, so this is each of the capacitance now split into two capacitances in series and therefore, we have much higher impedance between.

So this basically is the component concept that we introduce something in between some something in between a line such that it splits the direct capacitive coupling from any external line to the line that matters which is the plus and the minus line. Now, this line is called the guard line. If you see for example, you have an OP-AMP and let me indicate by them pads because when you do the PCB routing, it may be better visualize in this way and let us say this OP-AMP has a plus minus imagine that the OP-AMP is like this here. Then you have lines coming out from here and gets terminated probably at connector or some other components drivers whatever maybe which is basically giving you the differential source here.

Now, these lines we need to guard from all external lines so what we shall do. We shall erase a part of this; then give the guard in this fashion let us say you guard these two lines by putting a track like this all round it.

So this track after you have put a track like this here, you need to give it a potential and normally we will give it a potential which is lower or there is a minus potential. So let us connect it to this potential or it could be connected to ground it is called the guard ground.

Now, from here onwards you could take in the wires and probably through the other layer you can make the connection to the inner line.

You see these lines are guarded from any external line which has capacity coupling to the guard but, not direct coupling to the lines of the differential amplifier. This will improve the performance of high precision difference amplifier to a great extent so you should try to use the guard concept not only here even in high frequency cascade stages you could use the guard concept to reduce the capacity of coupling effects on the various amplifier stages.

So these are some of the points that I thought I will put present to you. There are lots of points which you will learn as you go along and making the printed wiring boards and you will learn more and more from experience as you start doing it.

It is difficult to list down all the points exhaustively. However, this should give you a good start when you start making your PCB on a CAD package. So, before using the CAD package, brush up on these principles before; so that you may make the layout on routing in less number of iterations. Thank you for now.