Power Electronics with Wide Bandgap Devices
Dr. Moumita das
School of Computing and Electrical Engineering
Indian Institute of Technology, Mandi

Lecture-17
LAYOUT

LAYOUT

Welcome to the course on power electronics with wide band gap devices. Today I am going to discuss about the PCB part. So what are the constraints that we need to consider for designing high frequency PCBs for the wide band gap device applications. So basically when we are going to consider wide band gap devices, the operation of the converter will be at high frequency. If the operation is at high frequency, then there are several factors we need to consider for designing PCBs. Let's see what are these factors.

Refer slide time (01:15)

To achieve high frequency PCB :

1. Minimum on-board noise
2. Minimum cross talk
3. Reduction of ground bounce
4. Simultaneous switching noise reduction during operation
5. Provide correct signal line termination.

Board Material :- It controls how much noise & cross talk is contributed from fast switching input-output signals

$$F = \frac{Q_1 \cdot Q_2}{4\pi \varepsilon_r^2}$$

signal can propagate faster if ε_r is less.

Dr. Moumita Das: Assistant Professor, IIT Mandi 2

Basically, if we are going to consider this PCB design to achieve high frequency PCB we need to consider some factors so these factors i am listing down here so the first is that minimum onboard noise So, noise is one of the important thing which affects at high frequency operation. So, basically when the operation is at slightly lower frequency the noises effect will be much less as compared to the high frequency operation. So, we have to consider a PCB design where this noise effect will be less from the board itself. So, we will see in details how this noise is coming from the board.

So, the second point is that minimum crosstalk. So, crosstalk already I have discussed earlier in details. So, basically crosstalk effect of one particular line to other line. or one switch to another. So, that effect we have to minimize that also we will see in details ok.

Then reduction of ground bounce. So, basically what happens if all the connections in the PCBs are connected to ground at some point. So, then we have to make sure that at every point there is a decoupling capacitor connected in order to reduce the noises from PCBs. So, that is called ground bounce. Now, fourth point is that simultaneous switching noise reduction during operation.

So, it can happen that there is there are actually multiple switches. So, then we have to make sure the simultaneous switching are not happening. So basically there may be some gap between switching of one particular switches with respect to another switches. So that part we need to look after. Fifth point is that provide correct signal line.

So, we have to provide termination of each signal line. So, whenever there is a signal line with respect to each line it will be having some impedance. So, we that we have to actually provide termination for that particular impedance otherwise noises will travel back and forth by using that impedance line. We will see in details these X points. So, basically you can see these are the different points which is going to affect the high frequency PCBs for wide band gap device applications.

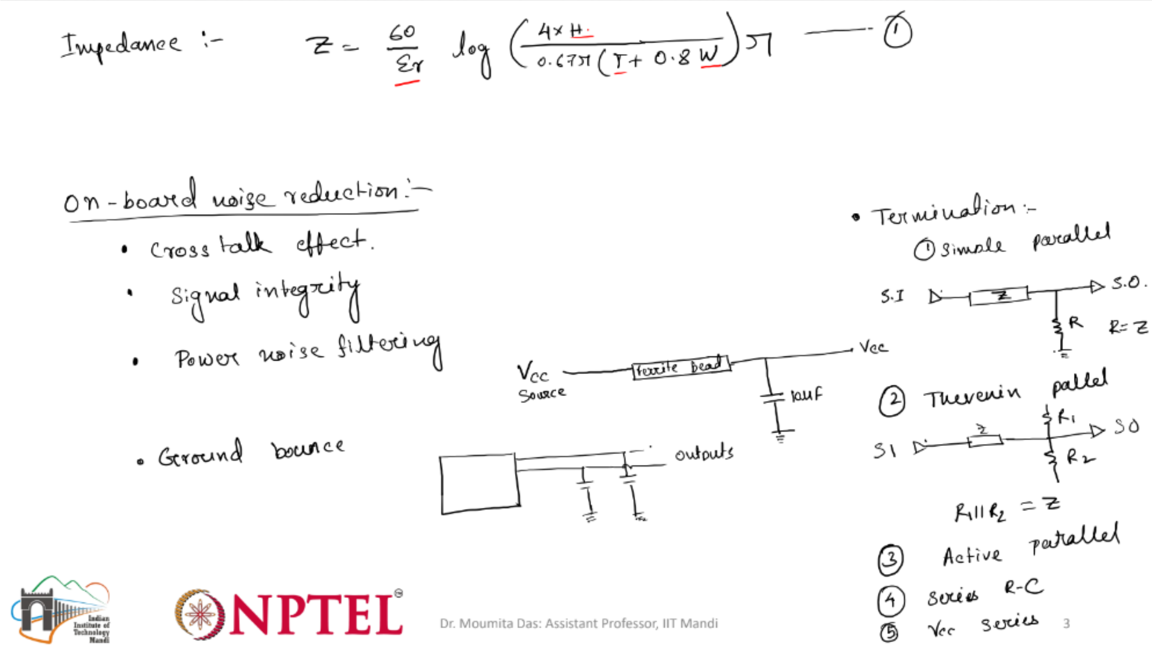
Now when we are considering this different effect, so basically so then what we have to do? We have to provide some method so that we can reduce all these problems. So one of the important thing we need to consider is that board material. So whenever we are designing PCB or actually PCBs are made. So, there are like different materials we can consider. So, based on the board material also it is going to affect the operation of the converter at high frequency.

So, board material is one of the important point. So, board material if we have to consider. So, based on the material what will happen it controls How much noise and crosstalk is contributed from high frequency operation or fast switching operation. input output signal. So, basically this board material will give some force between the signals.

So, this force we can actually define as charges between two lines. divided by permittivity of the board material. Now, this force will cause some noises to be present in the board. So, depending upon the board material this noise can be more or less and how this board material is going to affect? So, you can see here this is the term which is known as permittivity, it comes from the board material. So, if this permittivity is high, so then force will be less, if it is less then the force will be high.

So, basically signal can propagate faster if epsilon r is less. So, this we can actually choose based on the availability of the board material, we can choose the lower permittivity material, so that signal propagation will be faster.

Refer slide time (08:50)



There is one important thing is that impedance.

$$Z = \frac{60}{\varepsilon_r} \log\left(\frac{4 \times H}{0.67\pi\,(T + 0.8\,W)}\right)\Omega \quad \text{—} \quad (1)$$

So, basically what happens whenever any line is present in the board, so then that particular line will give rise some impedance. How we can define this impedance? This impedance we can actually define in terms of Z which is equal to, So you can see here, so there are different terms present.

So these are all coming from the board itself. So h is the height of the board, t is the thickness of the board and w is the width of the board and epsilon r is the permittivity of the board material. So all this will give rise some impedance. So whenever there is a line, so then that particular line will have some impedance now this impedance need proper termination otherwise there will be noise which will be traveling back and forth through this impedance okay so now that termination can be done by using different method. So, then let us see.

So, basically in order to reduce, so these are the noises which can be present in the board itself and at high frequency the effect of the noises will be more. So, we need to provide a way to reduce these noises. So, to reduce these noises, onboard noises what we need to provide onboard noise reduction So, then what we have to do? We have to reduce crosstalk effect. Then signal integrating means we have to provide the signal path such that There is like signal I have already shown you in the other lectures. So, how the signal line should be? It should not be the bend should be such that there will be less noises present.

So, basically straight bend can cause more noises to be present. So, that is why the signal whenever there is a bend point, so that bending should be such that noise effect will be less. Power noise filtering So, every point we have to see in order to filter this. So, what we have to do if there is a VCC signal input signal let us say VCC input. So, that has to pass through LC filter.

So, LC filter generally it present ferrite bit so and then it will be connected close to the C. C is generally 10 micro farad in that range and then the output will be connected close to the PCB board. So, BCC source may be connected bit away from the PCB board, so then there can be this LC type of filter which can reduce noise between this source and the point which is connected to the PCB okay then power this is with respect to the power noise filtering now if we have ground bounce effect so that generally comes during the switching of multiple switches. So, like some switch it is turning on and off simultaneously other switches are turning on and off. So, that will cause the ground bounce effect.

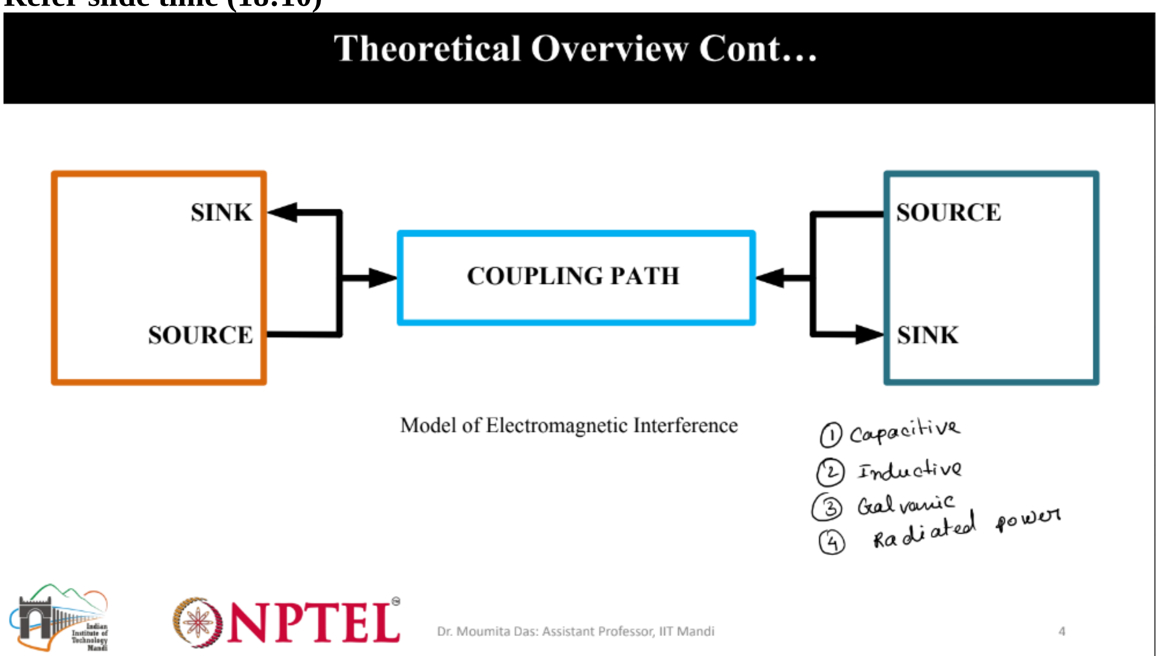
So, now in order to reduce that what we have to do again we have to connect filter. So, decoupling filter. So, basically if there are like multiple output which are connected and each are like switching simultaneously then what we can do we can actually connect capacitors with respect to each signal. So, then it will reduce the ground bounce effect. As I mentioned that there can be impedances in the line and which needs termination.

So, how this impedance termination is going to be? So, basically if there is a impedance we have to provide matching impedance for the termination. So, that it can reduce the noises due to this impedance present. So, this termination can be of different type means like this impedance termination can be classified in different types so they are the first one is simple parallel so then here what happens if there is a signal so this signal is connected to the impedance which can be calculated by using this particular formula. So, Z impedance. So, this Z already present in the PCB board.

This is the signal input. Now, the signal output Will be affected due to this impedance present What we have to do? We have to provide the matching impedance So now in this simple parallel method So the matching impedance will be connected in the parallel So that will be connected to the ground So this is the matching impedance Okay now there can be another method by using thevenin type of termination now this is like active type of termination source series type of termination so another is the thevenin thevenin parallel termination so there what happens so this z will be present then due to thevenin so there can be two impedances connected this way, signal output, signal input. So, here in the first one in simple parallel one, r should be equal to the value of z. And in case of the Thevenin 1, since it is R1 and R2, it is connected in parallel. So, R1 parallel R2 should be equal to Z. So, ultimately matching impedance we have to provide for the termination.

Similarly, there will be active parallel R1. then series RC, then VCC series, termination. So, these all are the classification of the termination. This will reduce the effect of the impedance present in the PCB board.

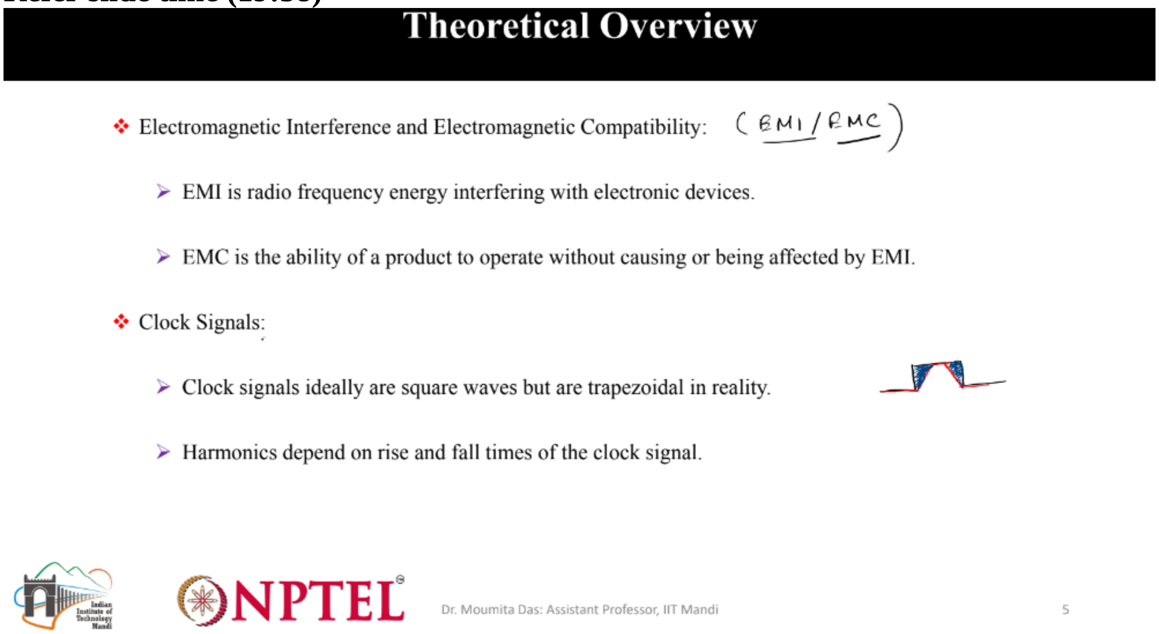
Refer slide time (18:10)



now you see that like this is the diagram of like every path which is present in the PCB. So, each line in the PCB board it act as a source and sink. So, if you see here in this particular block diagram it is shown source and sink one with respect to the orange color and it is connected to coupling path which is connected to another source and sink. Each line if it is act as source and sink and this is actually connected to coupling medium. What are this coupling medium? So, this coupling medium, so basically coupling path what is shown here. So, it can be either let's say capacitive or inductive or galvanic.

Radiated power So now what happens it can cause interference between for this present of coupling medium. So, this coupling medium can be capacitive, inductive, galvanic, radiated power. So, anything it can be.

Refer slide time (19:58)



So, this line will interfere with any of this component and it will cause interference okay. So, you see, so these are actually different component which generally affects the PCB board.

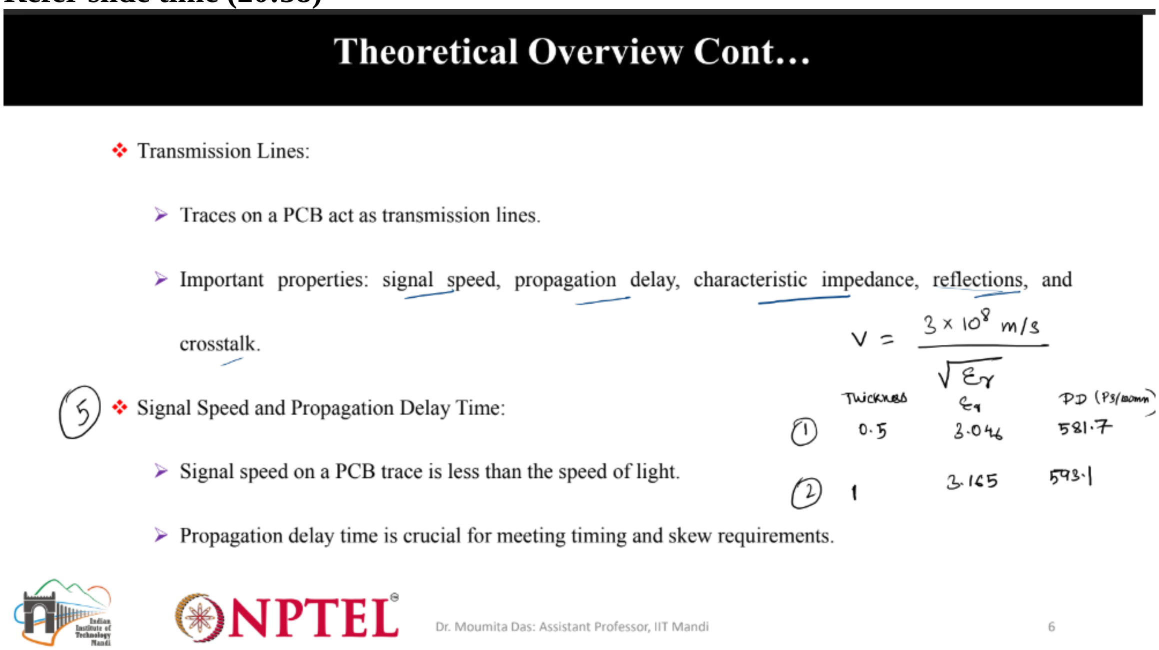
So, I have listed all these components also I have discussed earlier. So, the first thing is that electromagnetic interference and electromagnetic compatibility basically EMI or EMC. So, what happens this EMI is a radio frequency energy interfering with the electronic devices. So, this interference as I have discussed in the previous slide, so it can be with any of the coupling medium. So, this coupling medium will be present in the PCB by default.

So, this will cause EMI type of interference. Now, EMC is the ability of a product to operate without causing or being affected by EMI. So, we have to actually look after these two things whenever we are operating any system at high frequency, what is their EMI and then what is the EMC of that particular system. If it is okay then fine, otherwise we have to design filter in order to reduce the noises due to EMI. Now, this filter as I have discussed earlier it will be in terms of LC.

So, there can be decoupling capacitor in multiple places or there can be ferrite beads or combinations of decoupling capacitor and the ferrite beads. Next is the clock signal. So, basically clock signals are ideally are square wave, but it is actually trapezoidal because whenever we have provide any signal like any type of signal, we decide it to be square wave or we expect it to be square wave. But generally what happens, this turn out to be trapezoidal like this. So, you can see here the black one is something which we expect, but red one is something which we get.

So, due to these, so you can see here, so basically these are the places which will cause noises to be present. So, this different like the shaded portion which I have shown here, this part we have to see how to minimize this particular part. So, harmonics depends on this rise and fall time of the clock signal. So, as I have seen here, so basically these two shaded portion it is due to the rise time and fall time of the clock signal and it will cause harmonic to be present due to the clock signal.

Refer slide time (20:58)



Then third is the transmission lines. So, basically traces on PCB it acts as transmission line. It is not the power line, it is the transmission line of the PCB. So, whatever line you are drawing in the PCB that is PCB transmission line. and It is the one which decide signal speed, propagation delay, characteristic impedance, reflections and crosstalk. So, now one of the important thing is that signal speed or the propagation delay.

So, how to find out this propagation delay? So, propagation delay can be find out. So, basically it can be represent as

$$V = \frac{3 \times 10^8 \, m/s}{\sqrt{\varepsilon_r}}$$

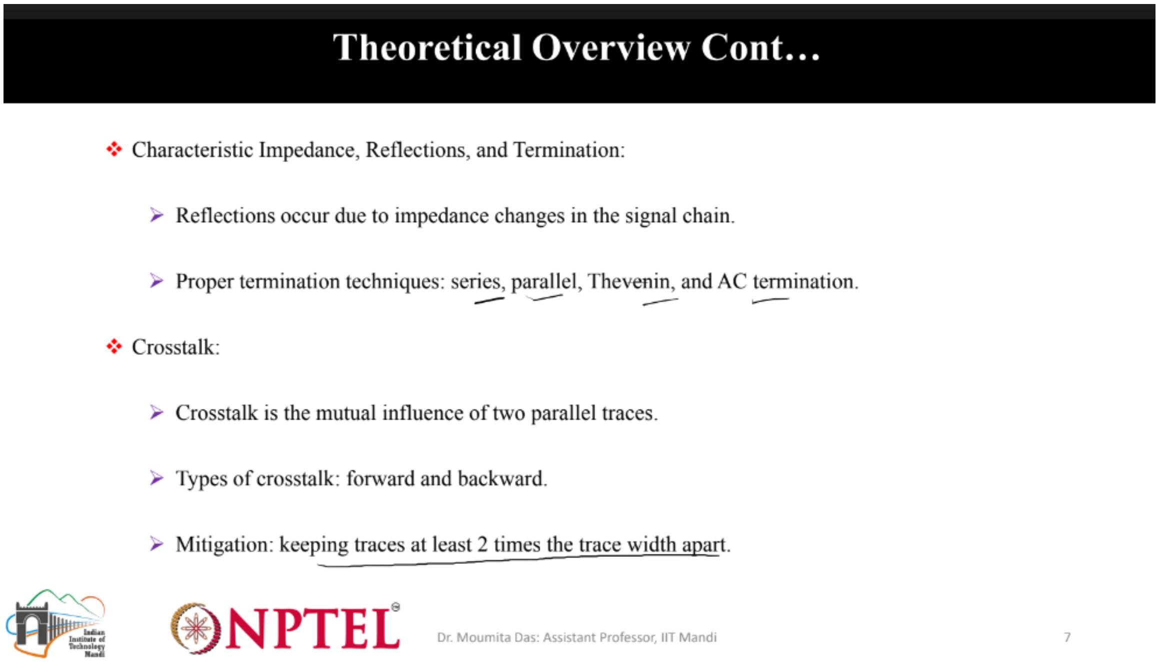
So, whenever any signal is flowing through the transmission line, so the speed of the signal will not be equal to the speed of the light.

So, speed of the light is given here. So, how different it is going to be? So, it will be affected by the permittivity of the board material so you can see here the speed of the light divided by root under epsilon r so this is the factor it is going to decide how much propagation delay is going to be there so for different board material or the different value of all the other parameter this propagation delay can be different. Okay. And it is going to affect the signal speed. Okay. So, let us say if there is a like any like PCB board which is having let's say micro strip one and micro strip two two different types of the PCB lines and everything if i decide so let's say one system and other second one so each is having let's say thickness 0.

5, 1 so this is the thickness Then epsilon r permittivity of these two board material it is different. Let's say one is 3.046 another is 3.165. and then there can be other parameter also by based on this parameter we can actually find out the propagation delay.

So, the propagation delay so in by using other parameter it is calculated as 581.7 for one picosecond for 100 millimeter and another one is 593.1. So, you can see like different pore material, different thickness of the PCB lines all can affect the propagation delay So, we have to see that how to make sure this delay is not much for any other So, now this signal speed and the propagation delay time it depends on you can see this fifth one So, signal speed on a PCB trace is less than the speed of the light as I have told you. The propagation delay time is crucial for meeting timing and skew requirements.

Refer slide time (26:25)

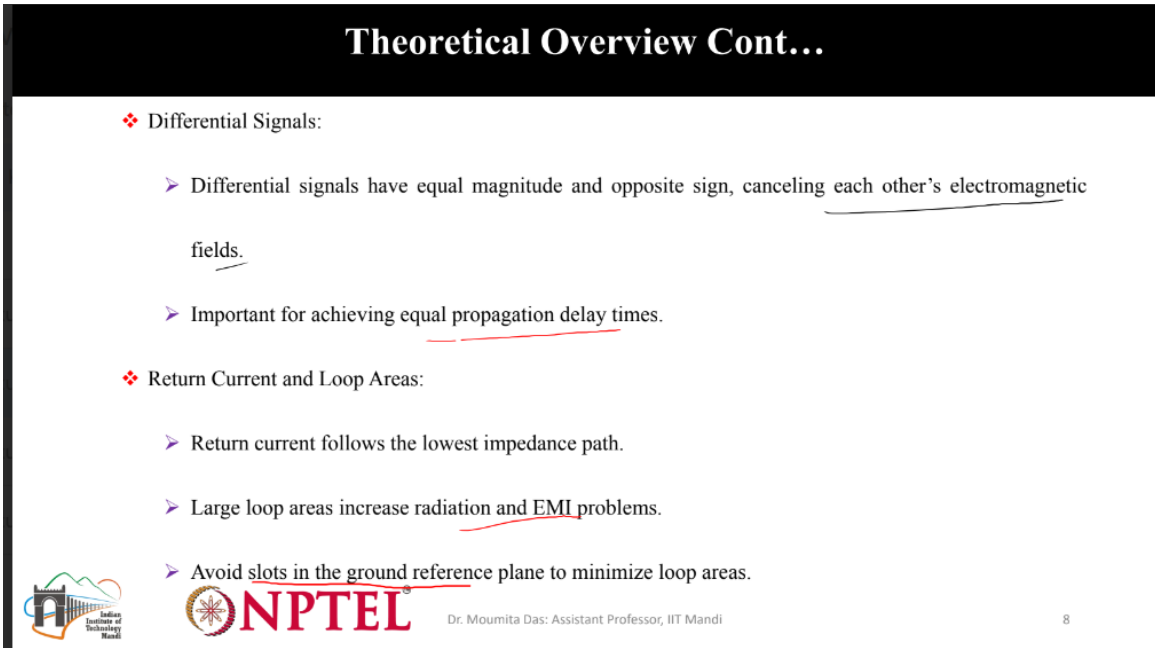


So, that is why designing, while designing PCB we have to consider all these different factors and if it is for high frequency these factors becomes quite significant because any delay in the signal it is going to affect the operation of the entire system. Now, sixth one is the characteristic impedance reflections and the termination. This is with respect to the impedance which I have already discussed in details. So, reflection occurs due to the impedance changes in the signal change.

So, that is why we need proper termination. So, the termination can be using anything like the simple parallel, thevenin or or active any method you can use so that we can actually provide termination for the impedance so that there will not be any reflection. So, proper termination technique series parallel Thevenin and AC termination any of this we can use in order to provide the termination. But before that we need to calculate the impedance. Unless and until we have the impedance value we cannot provide the matching impedance for the termination because for termination we need to provide matching impedance.

Now another point is the crosstalk. So the crosstalk is the mutual influence of the two parallel traces. So the two lines if it is there so then one line can affect the other line. So that is why this type of crosstalk can be either forward or backward. so mitigation how to mitigate this crosstalk so keeping traces at least two times the trace width apart so this is the point it needs to fulfill in order to reduce the crosstalk effect.

Refer slide time (27:58)

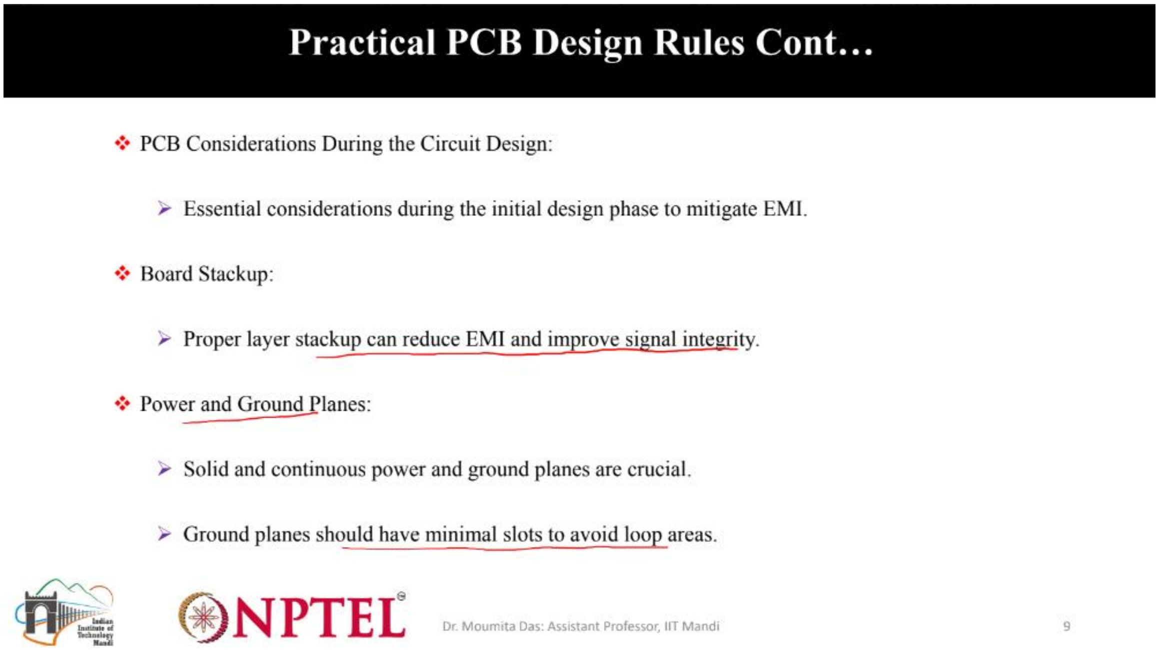


Now, differential signal. So, if we have differential signal then what will happen? One advantage is that if they have equal magnitude and opposite sign then it will cancel each other's electromagnetic field.

So, that is one advantage of using the differential signal close to each other. The electromagnetic field will be actually cancelled. So, this is important for achieving equal propagation delay time. Otherwise, you know the propagation delay for forward path and the return path may be different, right. So, that is why it is very important to have the differential signal lines close to each other.

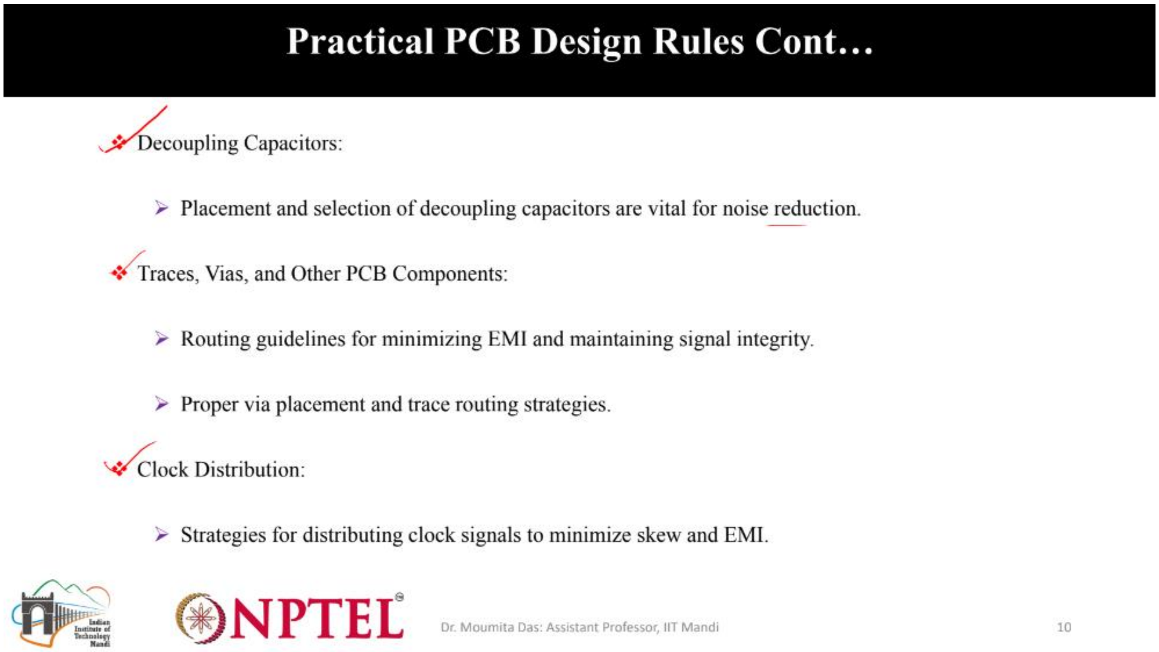
Return current and the loop areas. The return current follows the lowest impedance path and the large loop areas increase radiation and the EMI problem. To avoid slots in the ground reference planes to minimize the loop areas. So, that is why this part need to be look after in order to provide the solution for this EMI problem for this large roof areas, increased radiation and EMI related issues, ok.

Refer slide time (29:20)



Next is that PCB consideration during the circuit design. So, essential consideration during the initial design phase to mitigate the EMI, ok.

The next point is the board stack up. So, proper layer of the stack up can reduce EMI and improve signal integrity. Then one of the main important point is with respect to power and ground planes. So, solid and continuous power and ground planes are crucial. That will actually reduce many effect, many signal related issues and noises can also be reduced down. If we have continuous power and ground plane, if it is not continuous plane, if it is not multi plane, at least proper bus we can provide where the power and ground planes can be provided in the like in a bus line so that each signal where the termination is required so it can be connected very easily and also ground plane will be close to each of the termination point okay so that will reduce the noise So, ground plane should have minimum slots to avoid the loop areas.
Refer slide time (30:25)



Now, in order to reduce the noises, noises will be present in the PCBs, we have to place the decoupling capacitor in the multiple places. So, placement and selection of the decoupling capacitors, it is very important for noise reduction. And then traces and vias and other PCB components, routing guidelines for minimizing EMI and maintaining signal integrity is very important. And proper via placement and trace routing strategies is required. So, clock distribution, strategies for clock signal distribution to minimize the delay and the EMI effect.

So, these are the references. for this particular topic, you can just go through these references to know how to design the PCBs for high frequency operation. Thank you.