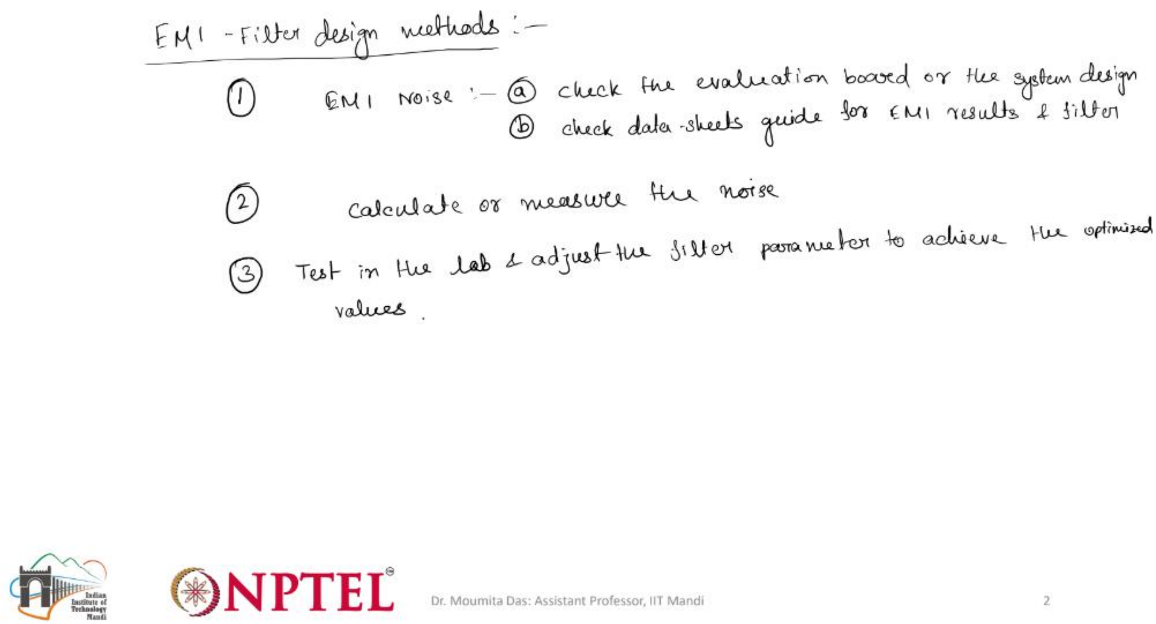
Power Electronics with Wide Bandgap Devices
Dr. Moumita das
School of Computing and Electrical Engineering
Indian Institute of Technology, Mandi

Lecture-22
Electromagnetic Interference (EMI - Passive Filter)

Electromagnetic Interference (EMI - Passive Filter)

welcome to the course on power electronics with wide band gap devices today i am going to discuss the filter design part for EMI so in order to reduce the EMI noise especially conductive type of noise the method we need to follow so that can be by using passive filter or active filter Today I am going to discuss about the passive filter part.

Refer slide time (01:10)



So, generally the EMI filter designing it needs some step right. So, what are those step? So, if I initialize the EMI filter design. then it follows certain step and this is applicable for both the passive and active type of filter right the first thing is that we need to have information of the EMI noise how we can do that in order to find out what are the noise present in the system then we have to do so basically we have to look into the evaluation board or the system PCB board or the overall system itself. So, check the evaluation board or the system design. part in order to know about this noises, right.

So, let me just denote it as A, B, C. Then what we need to do? Check data sheets guide for EMI results and filter. so how we can actually find out this noise that we can find out from the data sheet guidelines to find out the noises and also can find out what will be the filter designing part right then we have to adjust so then we will get the information of the noises the second step is that we have to either calculate or measure the EMI noise first step is that we have to look into the evaluation board or follow the design steps of the system if we are following the design step then we have to calculate the EMI noise possible EMI noise can be present in the system if we are looking into the design part of the system. But if we are looking to the evaluation board then we have the hardware system so then in that case we can measure the EMI noise from the system itself.

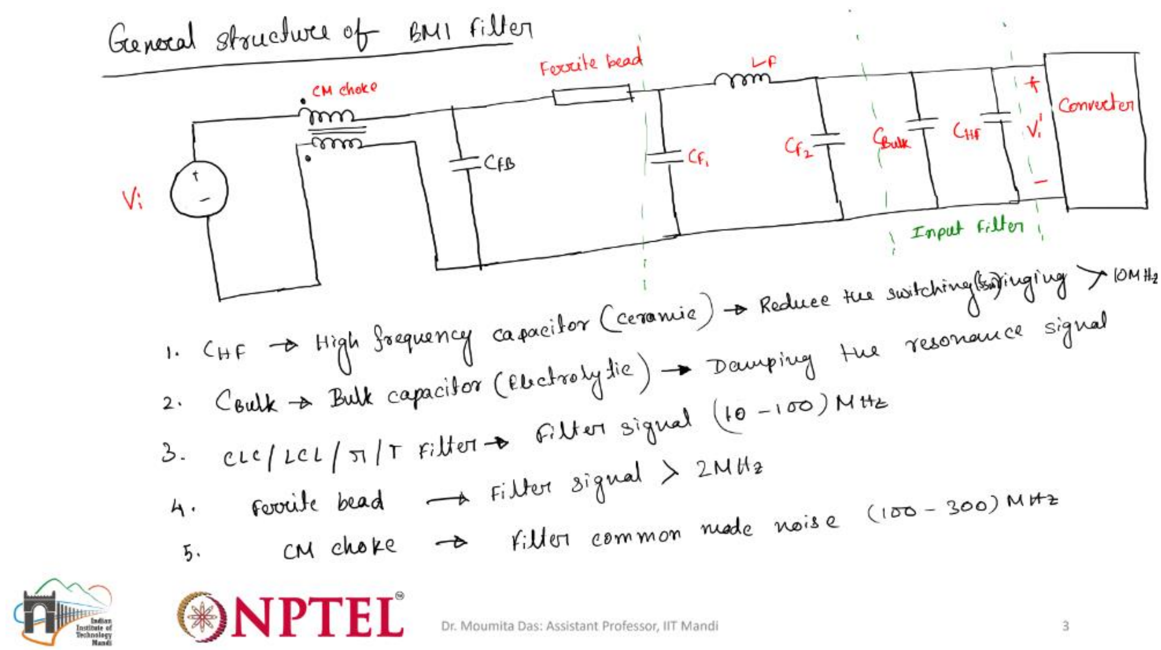
So that is why the second step will be calculate or measure the noise. So in the last class I have already discussed about this CM and DM noise. Right. So that measurement technique also I have discussed. So using that method you can measure or if you have to calculate then you have to look into the analysis part.

So I have shown you part for the CM and DM noises for buck and boost converter. Same method you can apply for other converter to find out the noises from the design steps. Okay. Now third is that now once we have the information then we have to design the filter. Once we design the filter we have to check whether that filter is suitable or not.

We have to check in by connecting in the system or we have to check in the design steps whether it is attenuating the required signal noise or not. Okay. Test in the lab and if it is suitable then it is fine no problem if it is not suitable then we have to adjust the filter parameters to achieve the optimized values okay so these are the different steps we need to follow this is the overall steps this we need to follow for like designing of any type of filter for the EMI noise find out the noise source and find out that can be find out from the board or the design by looking into the data sheet guidelines or the application not like how we can find out these noises that already I have discussed in the last class second is that calculate or measure the same thing whether we have whatever we are like measurement method we are following that already matching or not And then we have to find out in the experiment to find out the actual value of the filter right. So, this is the general method of EMI filter. Now, if we have to consider the designing of this filter.

So, today I am going to focus mainly on the common mode noises ok. So, how to mitigate this common mode noises and what are the different types of filter we can actually consider. So that part will come next but what is the general structure of this EMI filter? So if I say general structure that means that filter in place of filter it can be any type of filter but other than the main filter part there will be additional component also. connected in the system before actually connecting to the main system, main converter, all these components will come. So, what are those components?

Refer slide time (08:40)



So, general structure if I say, general structure of EMI filter how it looks like say there will be just input which we are getting from the supply.

Here I am considering the DC supply so that is why this is input DC supply. Now this DC supply will be connected to one CM choke or the common mode choke. It is nothing but one coupled inductor Right. So this choke will be connected after the source. This is CM choke.

I will just write down this name later. And then it will be connected to one filter capacitor. Right. This again will be connected to ferrite bead. This name I will write later.

This will be connected to another capacitor. And then this is inductor. This is input capacitor. And this is One like bulk capacitor will be connected here. See bulk and then high frequency capacitor will be connected.

And then this will be connected to the converter part. Then it will be connected to the converter part. Okay. So let me just write down the name of this different component. This is CM choke.

This is ferrite bead. Then this is bond CLC type of filter. So let's say CF1, CF2, L, F. Right. This is then connected to one C bulk capacitor and then which is connected across C high frequency capacitor and this is connected to the converter for which we are actually connecting the entire filter part to reduce the EMI noise.

So, VI we are actually providing from the supply here at this input point and the converter which is getting the VI here, VI dash that will be after the filter part. So, the noise from the source will not affect the load or the converter and similarly the converter noise will not affect the source, it is both ways. So, basically either ways the noise will not be transferred and it will be absorbed or attenuated in the filter part itself, right. So, what are these components? So let me elaborate this thing. So basically you can see the bulk capacitor which is present.

So I will actually start from the end part. See high frequency. This is actually high frequency capacitor. Generally, for this particular capacitor, ceramic capacitor is considered. High frequency small value capacitor, so that is why ceramic capacitor will be enough for this particular part.

So, why we need this? To reduce the switching signal, switching ringing, the noises due to the switching. To reduce this we need this kind of filter. So, then what is the level of the frequency? So, this reduction of this noise which is of frequency more than 10 megahertz. this CHF which you can see here in this block diagram this is to reduce the noises more than 10 megahertz I have also classified different noises like which noise comes under what part like so that I have already discussed earlier you can look into that so now the bulk capacitor C bulk This is known as bulk capacitor. Right, so the bulk capacitor is generally chosen as electrolytic kind of capacitor but other capacitors if it is available in that range which is having low ESR that can also be selected.

Now why we need this bulk capacitor that is because this provide like constant kind of supply and ESR of this bulk capacitor it helps in damping the resonant signal. Now this part is done. This is you can say input filter of the converter. Now comes this particular part where LF, CF1, CF2 are present. So this is the part where basically We are considering either so here I have considered CLC type of filter.

So it can be any other type CLC or maybe LCL or maybe pi type or T type. And it can be also first order, second order, third order, any order filter. So that is depending on the requirement of the attenuation. So this is this filter. This provides signal attenuation.

I can write filter the signal. Filter signal in the range of 10 to 100 megahertz. Now you may say that we already have high frequency capacitor. that can actually reduce the noises more than 10 megahertz then why we need this filter so this high frequency filter is to reduce the noises due to the ringing which is coming from the switching part from the switching frequency part so i can also write down here switching frequency ringing so due to switching frequency right Then this particular part it is not from the switching frequency it is from the common mode noise. So this is coming due to the parasitic capacitance which is coming due to the presence of the ground right.

So that is why we need this filter to reduce the common mode noise. You can see here this filter after this filter the component which is going into the capacitors that will go to the converter. So this signal should not go to the converter. So that is why this filter is important to reduce the CM noise.

Now fourth component. So now comes the ferrite bead. So what it does? It does filtering signal which is more than 2 MHz. frequency range. So, it will be from 2 megahertz to 10 megahertz. In that range, this filter, this ferrite bead will filter out the noises, right.

Now, comes the CM choke part. so this CM choke what it does it actually filter the common mode noise between 100 to 300 MHz Right, so this is the thing. So you can see here different frequency and different filter part which is required as part of the EMI filter. So one part I left here which is CFB. So this is actually provided in between common mode choke and the ferrite bead to provide a noise reduction okay so this part is not particularly for any level of noises so this is connected between because you know cm choke it is already coming here as inductor and the ferrite bead is again coming after this so that is why one capacitor component is provided so that two inductor will not come in series with each other. so that signal which will be passing through the CM choke that will be going to CFB and after that the same signal will pass through the ferrite bead in order to reduce these different noises so first it will reduce the noises of very high frequency then the frequency level will be lower in order to reduce the noises in that particular level through the ferrite bead and after that CLC-LCL filter will come.

So, that will again reduce the noises which will be 10 to 100 megahertz. So, 100 to 300 through the first component, 2 to 10 through the second component, 10 to 100to the third component so whatever signal is coming in this particular like system so by using this different filter component we can actually filter out the noises which will be in the range of 2 megahertz to 300MHz So, after that it will go to the bulk capacitor. So, anyway the bulk capacitor will act as low pass filter. So, then it will reduce the noises which will be present after that and then this filter output will be provided to the converter. If converter is having any noises due to switching that will be taken care by the high frequency capacitor part.

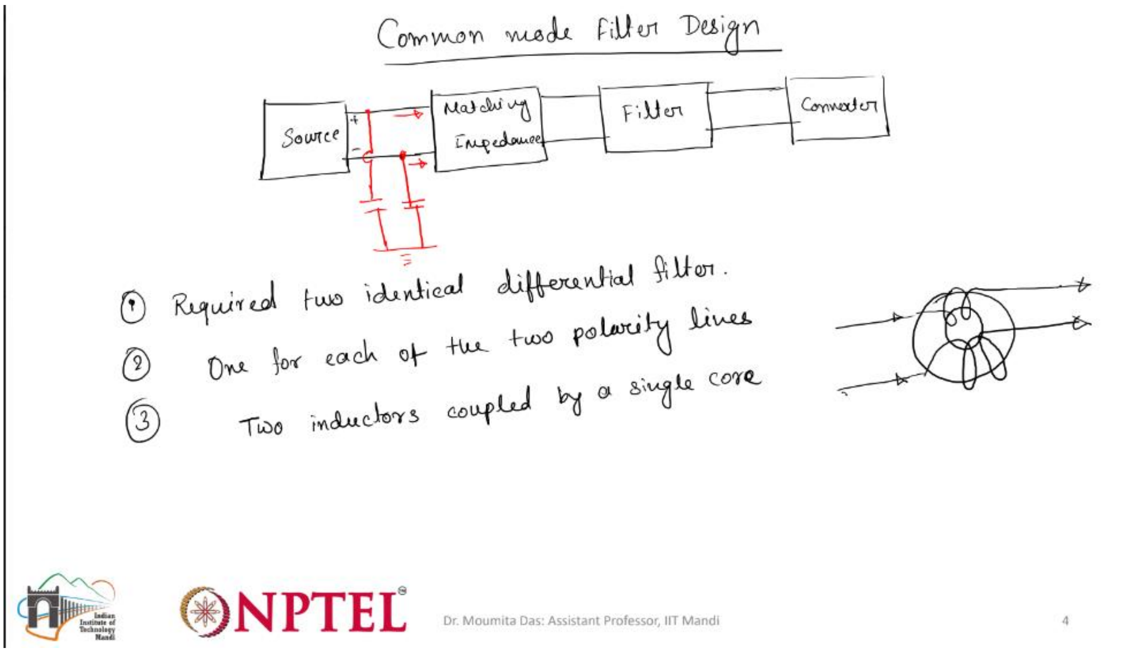
So this is how all this filter component is working in any system as part of EMI filter. So, by using all this component you will be able to reduce down the noises within the specified limit. So, the limit I have already discussed. So, there are like different limit levels as per CISPR like given limit specification. So, class 1, class 2, class 3, class 4 we can choose any of this and we can see whether the system responds within this limit or not.

If it is not then we need to design the filter. We can design filter but this component will increase the size of the overall system. So, that is why we have to see whether we need to connect all the components or Or any one or two components will be enough to attenuate the signal within the specified limit. Right. So, all this component we can connect that is not a problem that will help us to improve the system performance.

But it will increase the cost and size of the converter. So, we have to optimize the size also. That is why we have to choose the filter which is very much important for the converter. right so now in order to design that so basically now we know what are the different filter component and what method we need to follow now there are like a different types of filter available as you know the CLC LCLtype of filter so there are like first order filters second order filter third order filter different types of filter are available in order to reduce the common mode noise so you have seen the common mode choke which is used in this particular like the previous diagram where I have shown so that choke can be used to reduce the common mode noise but sometime it may happen that choke is not sufficient we may have to go to higher level of the filter of that particular choke so that choke itself is only one reactive component. So, if we consider only one choke then it comes under first order filter.

So, the choke itself can reduce the noises or the attenuate the noises in the level of 3 dB per octave so basically if we use only one choke that much attenuation will be possible but if we need higher attenuation in that frequency level what is given 100 to 300 megahertz so then we have to go for second order or third order filter so let's see how to design the filter any of this filter how to design it so we'll go in details of that okay so common mode So, when I consider this filter design, so generally what happens? So, the block diagram of this will look like. So, there will be source which you have already seen as the input voltage. Source, then we need matching impedance part. and then there will be the filter part and then the converter part comes.

Refer slide time (26:17)



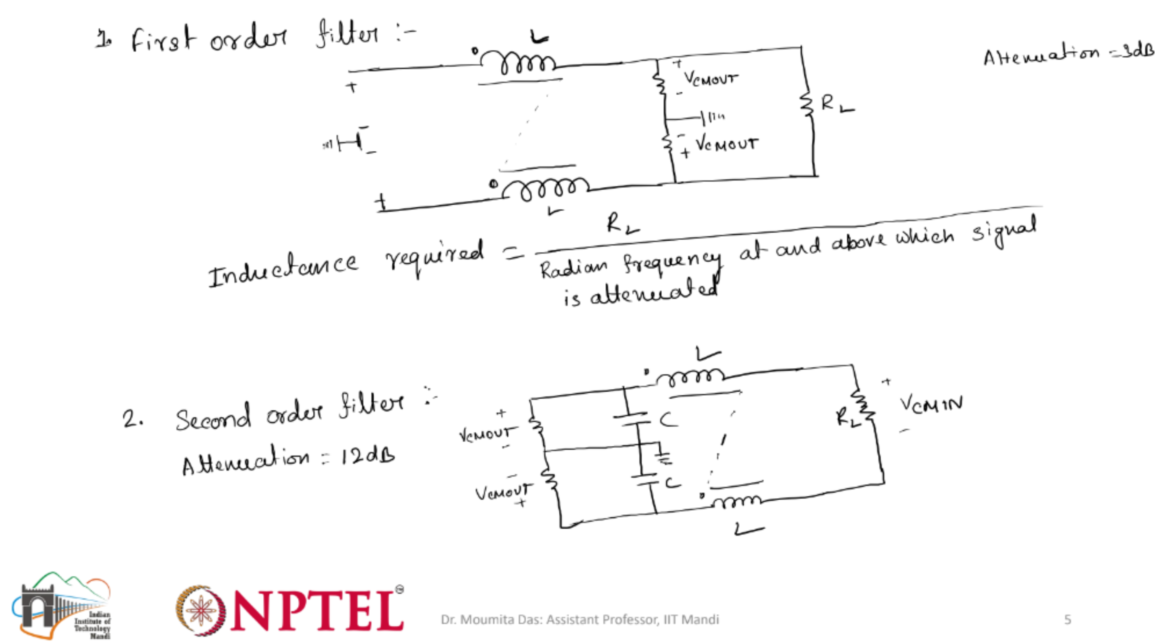
Right. So if we have to consider this common mode filter, The common mode filter basically it requires two identical filter. You know like the source if I consider so basically the noise which is coming it will be in both the lines. Means it will be in the line which is connected to the negative point of the DC. source and also the positive point of the DC source. So these two different lines will be having parasitic capacitances with respect to the ground.

So this both parasitic capacitances are responsible for the common mode noises. This will have the current in the same direction. So in order to reduce this noise we need it requires two identical differential filters. two identical filters are required, right. So, one for each of the two polarity lines as I have shown here.

So, if we consider the DC, so this is positive and this is negative. So, for positive polarity line we need one filter part and the negative polarity line we need another filter part ok. Third thing is that we are considering two filters but two identical filters the inductor if we are considering that should be mounted in the same core means reason of using this is due to the fact that whenever we are actually considering the single core so the core if we are considering this kind of code so then the signal which will be entering here so this will go and then mount will be like this and will go in this direction similarly another signal will come from the for the negative polarity then the signal again will go in this direction so this is actually coupled in single core so the fluxes of each line will cancel out so the net flux due to this two different lines will be zero means if there is a noise present so then only the net flux will be non-zero and which will be attenuated by the mutual inductance of this particular choke. The noise present in the system, common mode noise in case of ideal system if it is 0, so the net flux due to two different polarity lines will be 0.

So that is why we need to mount it in the same core. So that flux due to the positive line and flux due to the negative line it can cancel out each other. So that is the reason the presence of this common mode choke will not affect operation of the system if it doesn't have any noise. It will only come into picture if there is a noise and it will only attenuate the noise part. Right. by considering non-zero net flux and which will be attenuated through the mutual inductance.

Refer slide time (32:50)



Now, this filter is then can be considered in different types. First let us say this is first order filter. Now if we consider first order filter then it will have only one reactive component you can see that order is first means like only one inductor or capacitor anything can be present then the how this filter looks like so the filter will have this common mode choke which is like coupled in single core two windings are coupled in single core Right, then this will be connected to the matching impedance. Now these are the grounds and then this will be connected to the load resistance RL. Now how to find out the value of this inductance like common mode choke inductance.

So how to design that. So in order to design that we can find out so what will be the inductance required for any particular system that can be find out. So inductance means mutual inductance we are talking here. Inductance required equal to loads in ohms which is RL which will be in ohms divided by radian frequency at which signal is attenuated. So, once we have the radiant frequency for which the attenuation is required, basically the cutoff frequency. So, then this resistance load resistance if we have in ohm then from there we can find out what is the inductance required for this particular system and accordingly we can actually connect this inductance.

Now, as I said this can provide attenuation 3 dB in that range. But if in case we need higher attenuation then we have to go for higher order filter. So let's see what kind of attenuation we get in second order filter. So when I say second order filter as you can understand from the name there will be two reactive components. So in the first order filter only inductance was present couple inductor and in case of second order filter there will be capacitor component along with the inductor component.

So you can see here. So here we have this inductance similar to first order filter. That is very important. So we will connect these inductors. And before this also there is a capacitor present.

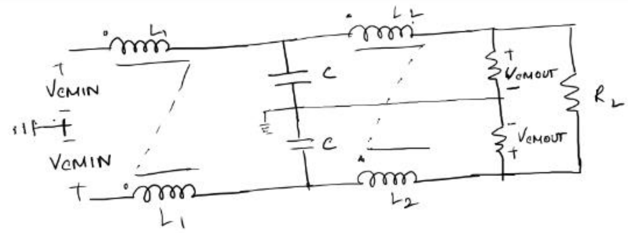
So there will be two capacitors with respect to each line. Because you know like this common mode filter it has to be connected to each line. So since two lines will be there. So that is why we have to consider two capacitor two inductors like this. it will be connected to the resistance RL.

This is VCMin actually. Just write it properly. CM out. So in this case I will just write down attenuation 3 dB. In this case second order filter the important thing is that attenuation will be equal to 12 dB per octave. So you can see 4 times attenuation we can achieve in second order filter. The disadvantage is we have to connect these capacitors in order to reach that particular attenuation level. So, now I will just tell you about third order filter then we can actually look into the design part of any particular filter.

Refer slide time (40:03)

Third order filter :

Attenuation = 18dB



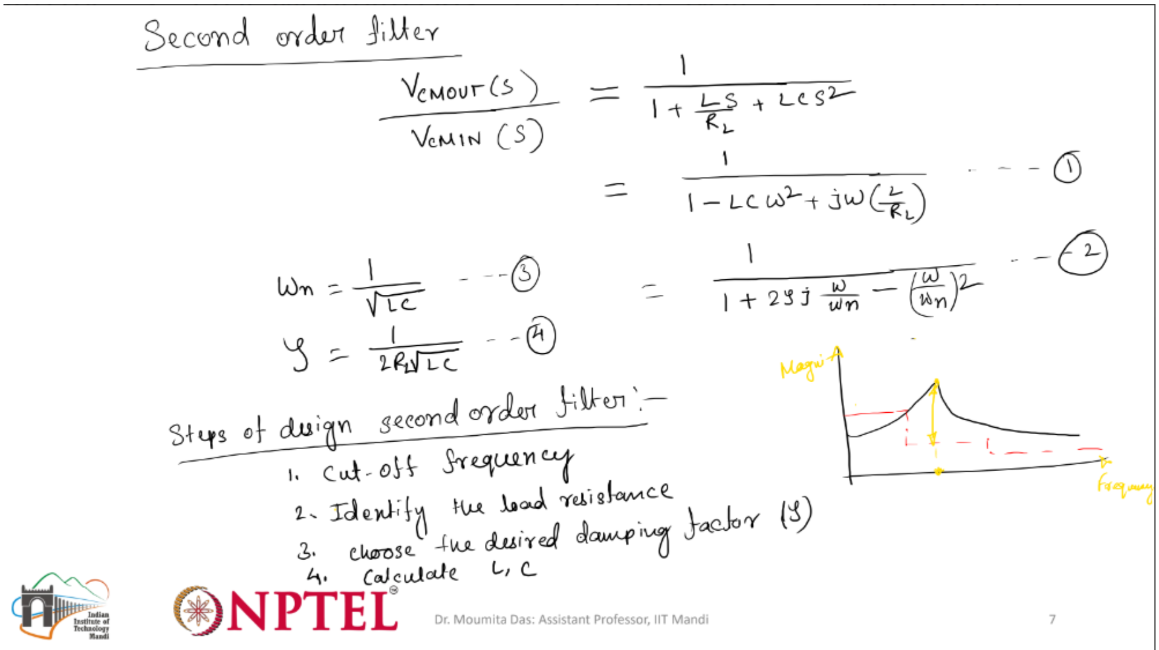
So, now in the third order filter what will be the difference as you can understand from this like again like more number of reactive component will be there three components will be there and due to that the attenuation will be higher as you increase the order of the filter you will get more number of components and then attenuation level will also increase. Your output signal will have much less noise as compared to the first order filter. So, this is how the it works. So, how it looks like? So, attenuation here you will get 18 dB per octave right. So, this will look like so it will have it will basically will have two sets of this inductors and this will be again connected to the capacitor Right.

This is another inductor. This will be connected to resistor. Right. And this is differencing load. RL. Okay. So, now this will be RL. We can say this is VCMout this is connected to ground like this okay here it will come VCMin it will have the ground connection like this Okay, so you can see the structure of this.

This will also have this coupling. You can consider this as L1, L1, L2, L2. These two inductors will have similar value. Means like L1, L1 as you see with respect to 1 coupled core. So, that will have same value. Similarly, the other inductor will have, that may have different value, but like two inductors will have same value, this L2, L2 and then L1 L1.

This is how it works and then these two capacitors will be also having same magnitude, right. So, this is how the third order filter looks like. Now, we have the knowledge of this different filter now in order to design any filter what we have to do what procedure we need to follow that we will see now so let's consider any filter so first of all let me just tell you about basically the filter designing part So, whenever we have to design the filter,.

Refer slide time (44:24)



So let's consider the second order filter part. So, second order filter what you have seen? You have seen two different components right, inductor and the capacitor. So, when you have two different components, so you can just refer the diagram of the second order filter.

So, I will just tell you the design of the second order filter. So, second order filter is having advantage of like higher level of attenuation compared to the lower value of the component in case of third order filter. The second order filter if I have to consider. So, first what we have to find out? We have to find out basically the transfer function. So, how to find out that? So, you have seen this VCMout. VCMin. So, if I try to write equation in Laplace domain then the simplified equation will be

$$\frac{V_{CMout}(s)}{V_{CMin}(s)} = \frac{1}{1+\frac{L}{R_L}s+LCs^2}$$

$$= \frac{1}{1-LC\omega^2+j\omega\left(\frac{L}{R_L}\right)}$$

$$= \frac{1}{1+j2\zeta\frac{\omega}{\omega_n}-\left(\frac{\omega}{\omega_n}\right)^2}$$

$\omega \equiv$ radian frequency

$R_L \equiv$ the noise load resistance

$$\omega_n \equiv \frac{1}{\sqrt{LC}}$$

$$\zeta \equiv \frac{L}{2R_L\sqrt{LC}}$$

So, previously you can actually see this second order filter it is having let us say this component is L, this component is C and this is your RL right. This is the equation we can get from the second order filter. So, once we have the equation we can actually simplify it in terms of radian frequency. So, then we can write down like above. You can just write down the equation steps, you will find these different equations, which can be again simplified in terms of frequency term omega n equals to 1 under root LC and zeta equals to 1 by 2 RL root under LC. These two terms if we actually replace the equation 1 by using this omega n and zeta then we can write down 1 by 1 plus 2 zeta j omega by omega n minus .

Omega by omega n square. Right. Now we get this equation. We get zeta omega n and then we get the transfer function. Then we can actually plot this to see what will be the system response to check the filter response in this case. Now if we are really designing this filter then what we have to find out? We have to find out the value of L and C. In filter we need only this reactive parameters. How to find out this reactive parameters? That can be find out by using this different equation.

So generally what happens? The attenuation level is given or maybe you can find out from the response itself. So what we have to find out? We have to find out the cutoff frequency. First we have to find out the cutoff frequency in which the attenuation is required. That can we only find out by plotting the noise or measuring the noise from the evaluation board that I told you before.

We have to measure the noise. Once we have the information of the noise, we can actually compare the noise with the standard given. From the standard, we can see where exactly the attenuation is required. What is the frequency level? So that is the cutoff frequency. So let's say, so we have a noise like this. and then our standard is given our standard is given like this so this particular point the attenuation will be required right it is this particular point is beyond the level so in order to find out the cutoff frequency what we have to do we have to find out the noise profile means how this noise looks like.

So that we can get from the measurement. So I already have discussed about the measurement technique. So from there we can get how this signal looks like. So this signal let's say this is the signal and then we have standard given like this. So let's say this is the point the attenuation is required. So we can actually find out what is the magnitude of the attenuation because this is the magnitude and this is the frequency.

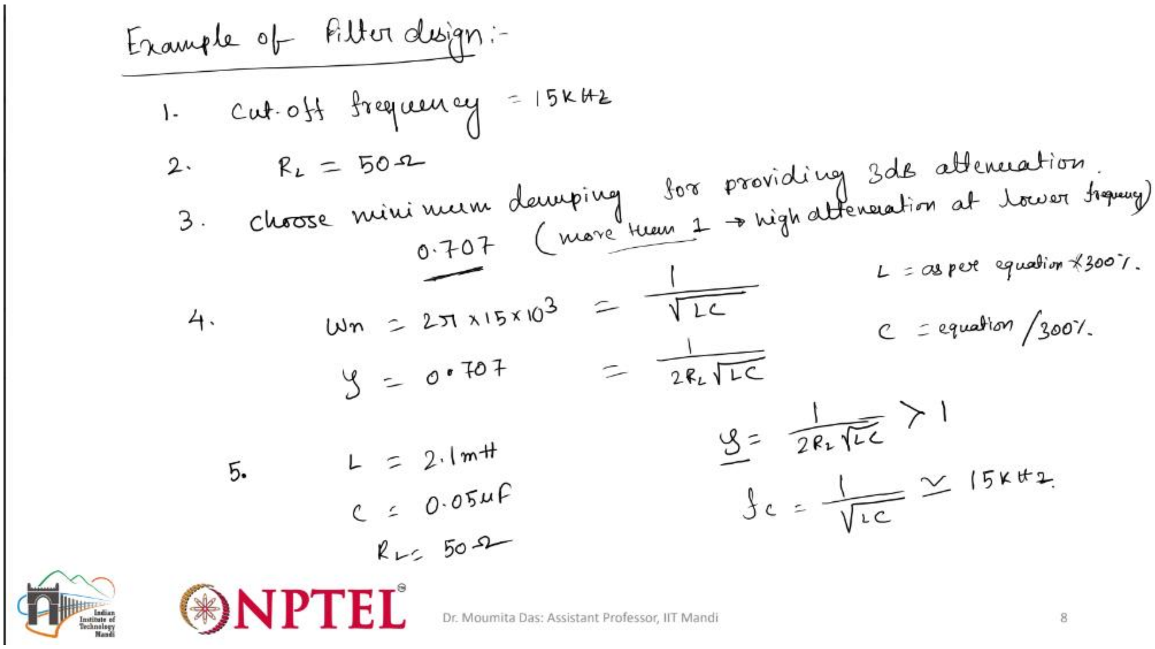
So then this point will be your cutoff frequency point. You have the cutoff frequency then you can actually note it down. So that may be let's say 15 kHz, 15 MHz, anything it can be, right? So, then once you have the cutoff frequency then we have to identify the load resistance because you know that is also required. Identify the load resistance. So, in this equation you can see here this 1, 2 and this term which is there 3, 4. So, here this load resistance part is present identify the load resistance that we can find out from the system itself once we have the system then we can actually find out desired damping factor So, then damping factor is zeta.

So, this zeta we can actually select based on the given like level of selection. So, generally the zeta is specified as 0.707 in that level we can actually find out or we can actually select different value and check whether that is matching with our requirement or not. Then after that what we have to do? Once we have this then we can actually calculate required components. What are the components L and C? So then from 3 and 4 so we have cutoff frequency that is 3 and then damping factor we are selecting that is 4 and we have actually identified the load resistance which is required in 4. So then what we can find out from this we can actually calculate L and C are the required components for the filter.

Now, we have the filter component, right? Then by using this filter component, what we have to see that, so like whatever component we are selecting, it has to also follow some properties. That properties if it is following, then it's okay or maybe we have to actually choose the value near to that. Once we choose the suitable value, then we have to see whether it is matching with the cutoff frequency or not. If it is matching the cutoff frequency, then what we have to do? Then we have to consider those filter for the noise reduction purpose. If it is not matching, then what we have to do? We have to recalculate this filter parameter again and we have to actually use the same procedure again and again until we reaches to the level where this filter is suitable for the system or it is serving the purpose of attenuation of the noise.

So, till then we have to follow the same procedure. So, then we can actually calculate this different filter component. Now, let's consider an example. So, example of like filter design. So, just I have I will just give you the data which I have with me.

Refer slide time (54:54)



Okay, first step what we have to do? We have to find out the cutoff frequency. So, let's take an example of cutoff frequency as 15 kilohertz. We have 15 kilohertz. Then what we have to do? Then we have to either find out the load resistance or select one load resistance.

So let us consider this load resistance is 50 ohm. We have two different parameters. So first and second point we have. Then third what is required? We need damping factor. So damping factor we have to consider as So, this let us select as 0.

707. You can choose different damping factor also, right. Now, what we have to do? We have to find out this L and C from omega n and zeta, right. So, then omega n I can get from 2 pi into this cutoff frequency that is 15 into 10 to the power 3, right. Zeta I have as 0.

707. So, this is equals to root 1 by root under LC and this is equals to 1 by 2 RL root under LC. So, from this I can actually find out the actual value of L and C. So, now whatever value of L and C I will be getting so that can either use that or maybe by using some method I can actually see whether this can be optimized so generally what happens so like this damping factor as I said this you can choose different values but here I am choosing 0.707 minimum damping factor if damping factor is like more than 1 for more than 1 one it will give high attenuation at lower frequency so that is why we are choosing 0.707 that is the minimum Similarly for inductance and capacitance also we have to consider some values.

right. So, we can either choose the calculated values or we can choose slightly bigger values in order to reduce the noises accordingly. So, maybe the inductance and capacitance whatever we are calculating. So, those parameter we can select or maybe one parameter we can actually increase and other parameter we can decrease in order to maintain the required omega n and zeta. So, accordingly we can actually find out the resultant L & C. So, L and C whatever we will be getting here so that we can actually select as per equation But if it is very less, if L is very less, then we can actually try to see higher value of L and that will give us lower value of capacitor in order to maintain these values.

So, accordingly we can get 2.1 millihenry and 0.05 microfarad and RL equals to 54. So, then this we can select and see whether these values are satisfying with the Required noise reduction purpose or not. If it is satisfying then we can use these values. If it is not satisfying then we have to follow the same procedure by using different damping factor to get different other values and then check whether it is satisfying the purpose or not. Then again if it is satisfying then we just stop there otherwise this process will continue until we reach the final value.

So, this is how we can design the filter for the common mode noise considering the second order filter. So, first order filter already I told you that is like very simple RL divided by the required this inductance and then for third order the process will become much more complicated. So, again like we have to write down the equation and from there we have to find out omega n and zeta and then we have to follow the procedure, ok. But this is basically general procedure which you can actually consider for any applications, right. so generally what happens so this zeta like whatever you are selecting as point seven zero seven so this is like more than one all is also acceptable right so zeta generally what happens you can select point seven zero seven or you can select more than one anything is acceptable So, like you have to see that it is more than 0.

707 or whenever you are actually measuring this LC and whenever you have calculated this LC. So, then you can again recheck whatever will be what will be the value of the zeta. So, the zeta value should match either like minimum this value or it will be more than 1 that particular level it should match. So ideally whenever we are selecting the final zeta it should be more than 1. So by using this L and C again you can check 1 by 2 RL root under LC the final value will give some zeta.

So this has to be more than 1. So that check is required. Similarly the cutoff frequency again you have to check. Fc goes to 1 by root under LC. That has to be close to the given cutoff frequency 15 kilohertz. Right. So, generally what happens? This variation comes as we take inductor as per the equation multiplied by 300 percent.

And then for capacitance what we do? We try to reduce it is divided by 300 percent. so that is how we adjust means either we can get whatever values we are getting but then the inductance will be very small it will be very difficult to design that kind of inductance if we increase the inductance then it will be easier to get because you know the common mode choke and everything whatever will be available so that will be of higher value so then we can increase that inductance in order to get it easily or in order to wind it easily Then we have to make sure this capacitance value we are decreasing equally. Means if we are increasing 100% inductance we have to reduce 100% capacitance. If we are increasing 300% we have to reduce 300%. So here the calculation whatever L and C I have written here that is by increasing inductance as 300% and decreasing capacitance as 300%. Once we do that we have all the values we have to recheck zeta and fc because you know like we are not using the actual parameter.

We have now recalculated it. So we have actually modified as per the kind of our specification or maybe whatever will match our requirement. Then we have to recheck these values and we have to see whether this is matching with the design criteria or not. If it is matching means like it has to be greater than 1 zeta and fc nearly 15 kilohertz. Then it is fine or otherwise we have to actually consider different other values of inductor and capacitor. Is it clear to you? So, you can just take any cutoff frequency, you can design the filter, you can see whether you are getting the required attenuation or not.

That can be seen in the simulation. So, this is all about the filter design part considering the passive filter to reduce the common mode noise. So, this noise you can also see in the simulation. Now, the same thing I will be discussing by using the active filter in the next class. So, these are the references.

So, you can actually refer this particular reference common mode filter design guide by Coilcraft. So, this design guidelines you can follow in order to understand this. Thank you. This is all for today.