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

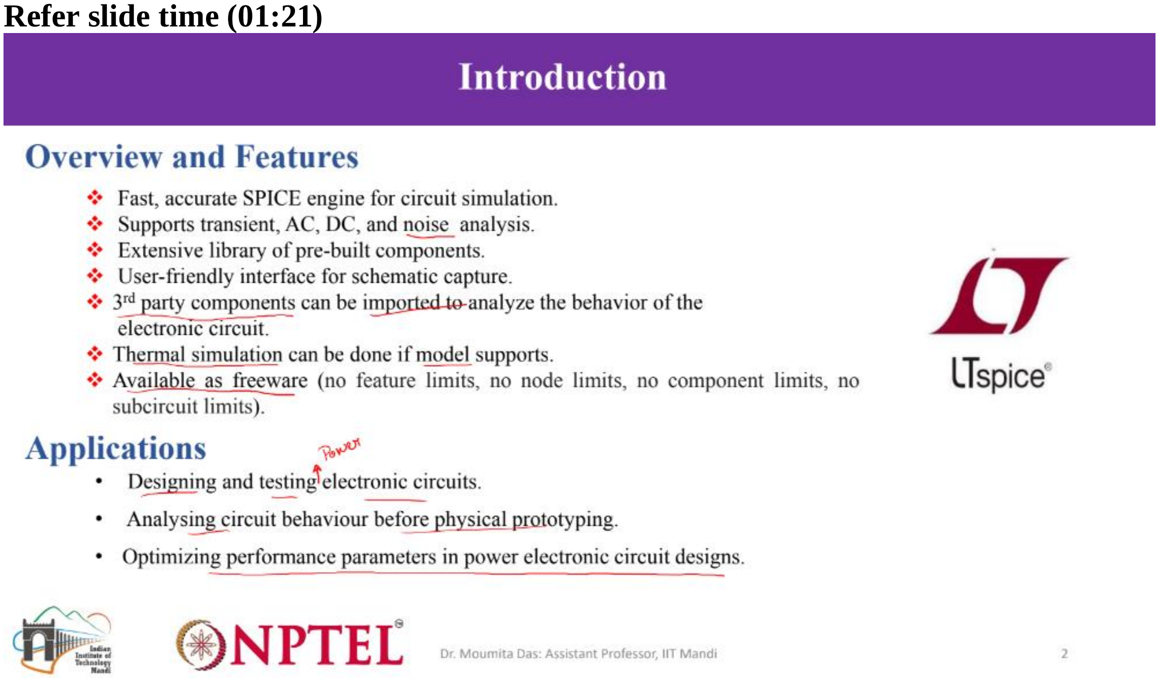
Lecture-25
GETTING STARTED WITH LTSPICE

GETTING STARTED WITH LTSPICE

Welcome to the course on power electronics with wide band gap devices. Today I am going to discuss about LTSpice simulation. So, in the last class I have already discussed about the active filter part. So, design of the active filter part. So next I am going to show you simulation of the converter with and without filter to show the EMI noises. The first step of that is installing the LTSpice software and getting know about this particular software.

Because the simulation which I will be showing it is in LTSpice software. and also I will be showing you how to simulate the wide bandgap devices in this particular software. So let's see how to use this software. So you can see so this is basically logo of the LTSpice software as you can see in this particular screen.

Refer slide time (01:21)



So why we use this particular software is that it is fast accurate spice engine for circuit simulation. It provides actual kind of circuit operation. So, it supports transient, AC, DC and noise analysis which we are interested in for this EMI part, so this noise analysis part. So how these noises are coming? So in MATLAB it is very difficult to see these noises. You can see these noises but like you have to connect many parameters in the circuit simulation in order to see this.

But that can be visible here without much complexity. So that is why I am going to use this particular software. So, this has extensive library and pre-built components and user-friendly interface for schematic capture. And it has third-party components can be imported to analyze the behavior of the electronic circuit. So, this third-party components part is very important.

So, if you see any like GaN device manufacturer, so let us say GaN systems. So, they provide this LT spice model of the GaN device. So, if you want to use that particular device in the simulation, then you have to download that from their website and then you need to import it in the LTSpice because they provide model with respect to the LTSpice software so that is why you need to use this particular software and it has feature of third party components so basically you can download that particular component and you can import it in your simulation means whatever simulation you will be doing so in place of the switch you can import the actual switch provided by the manufacturer and same device you can use in the simulation. So, whatever the device parameter inbuilt to it, it is given in the model. So, it will provide you the simulation results with respect to the actual behavior of the device.

It will be much close to the actual behavior, ok. So, then it will be very easy to understand the system performance by using those devices, right. Otherwise, you can get the system performance, but the practical problems in the devices or the converter, we may lose in that process. But if we can include whatever parameter as consider as the parasitic parameter, let us say capacitor, inductor, all this thing, if we can include, then we can get that kind of behavior, practical behavior, okay. And thermal simulation can be done if model supports.

So, this can be done. This is there in the LTSpice software. But like whatever model you are using. So, if this is supporting then you can do that. Otherwise I mean you can use the model which support the thermal simulation.

And you can check the simulation with respect to different temperature. Right. And the most important thing. It is available as free. So, no feature limits, no mode limits, no component limits, no sub circuit limits.

So, most of the software which we use for the simulation purpose for that we need to pay the license fee. You can use it initially for one month or maybe whatever time is given. Once you download it, after that you have to pay the license fee and then you have to use it. So it becomes very difficult to use any software without getting the license. But in this particular case, this particular software is free.

So you can download it and whatever library and all are given, you can use it without any worry. So generally what happens, for some of the softwares, depending upon the library requirements, they are providing, you can actually pay the fee. So, if you pay lower like license fee, so then probably restriction in the use of library will be there. If you pay the like maximum license fee that is given, so then probably you will be able to use all the libraries but here you don't have those limitations as it is free software right so this is the overview of this particular software so now applications where we are going to consider this kind of simulation so designing and testing of electronic power electronic circuit so designing and testing of electronic circuit and it will also have power electronic circuit or power circuits okay analyzing circuit behavior before physical prototyping so simulation as you understand we need to first analyze any converter device or anything then we can go for the simulation to check Whatever analysis we are doing, so that is matching with the simulation results or not. So, simulation part comes after the analysis part.

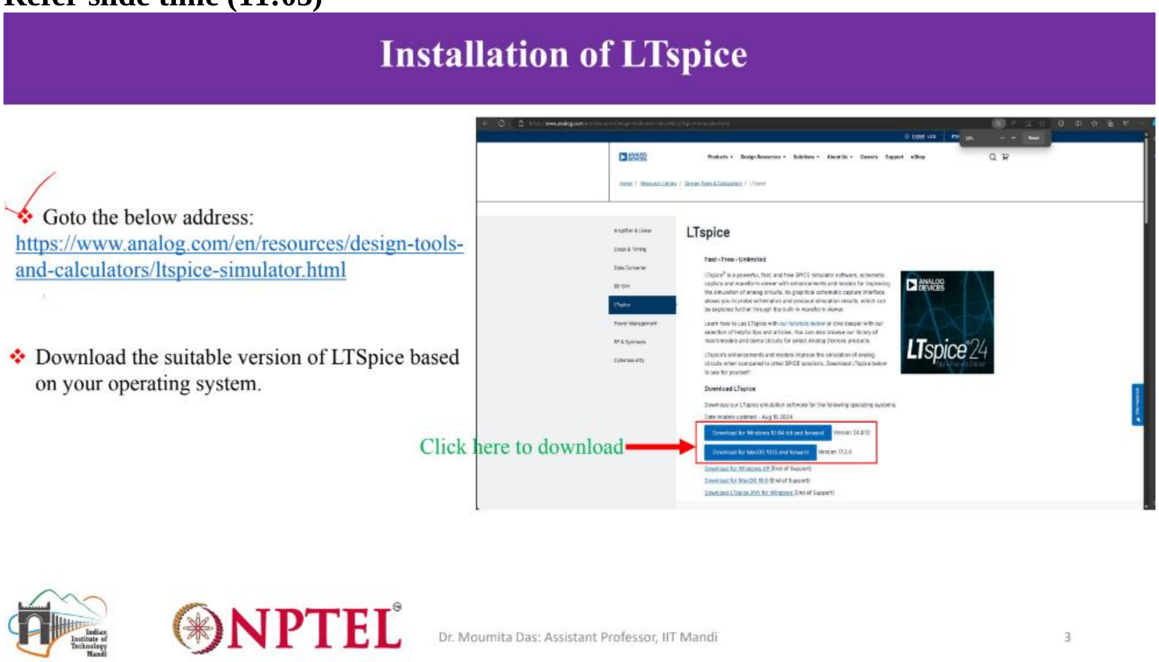
Analysis means basically basic understanding of the converter device or anything. Once the simulation we are getting that is like providing us satisfying kind of result, then we go for actual experimentation or physical testing. So, there are three processes. In these three processes generally what happens this analysis and the simulation it provides ideal kind of behavior. We lose many thing of the experimentation in the simulation or analysis initially if it is the first step.

Eventually after experimenting we have the idea about different parasitics and all the parameters, then we include those in simulation to know how the system is behaving with respect to that. If we can have information of all this thing beforehand, before starting the experiment, then it will be very easy to go for the experimentation. Means, then we will have the idea about what kind of behavior we are expecting and if there is any unsatisfying kind of behavior, say then what to do in order to avoid that kind of situation. So, let's say EMI analysis. so this is something which we don't desire but it is there in the system so now if we have to remove this noise then we have to design the filter so designing the filter comes much after having the knowledge of the noise.

So, in the simulation if we can see the noise beforehand, before going into the experimentation, then what we will do? We will design the filter beforehand and then we go for the experimentation. But if we do not have this kind of feature in the simulation, then we will assume that circuit is behaving like ideal characteristic And in experiment we will not get those kind of result. So, then there will be discrimination and then their problem will come. So, that is why this the simulation software which can providing us ideal kind of behavior that is good for initial understanding. But like before going for hardware prototyping we should choose a software or simulate in a software which provide more practical kind of behavior.

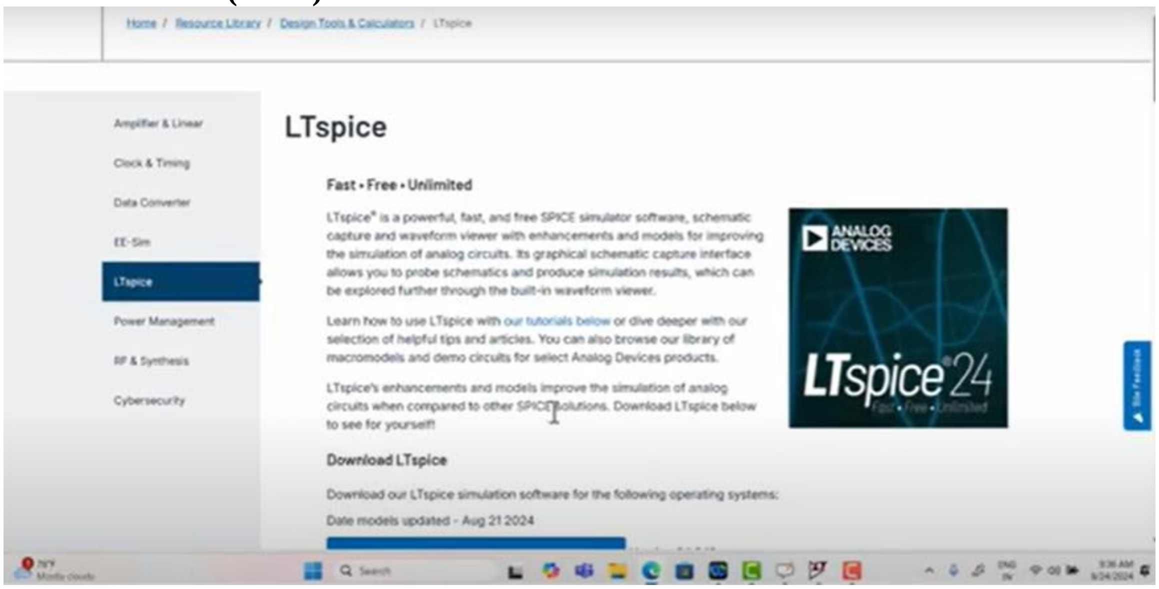
So, that can be done using this particular software. then optimizing the performance parameter in power electronic circuit design so this comes along with the practical characteristics so in that case we can actually optimize the parameter or probably we can use filter or anything in order to provide circuit with respect to the practical scenario okay now Today, I am going to just tell you how to download this software, use it for basic circuit. Then we will go for the complete simulation of the circuit using wide band gap devices and then we will see EMI results. So, let us see how to use this. so in order to download this particular software what you need to do you need to go this particular link so you can see here this link below so it is written here so it is analog www.

Refer slide time (11:05)



analog.com so other things will also coming here so that you can actually type and then you can go to the analog devices website. So, this website will look like this in the right hand side it is given like this the actual like screen how it will be visible to you so that is given here. So, then you have to download Suitable version of the LT spice based on your operating system. So you can see here. So like to Like two versions are given one for Windows and one for Mac So whatever is suitable for your system download that now I will just show you how to download by going into the actual website So, you can see here.

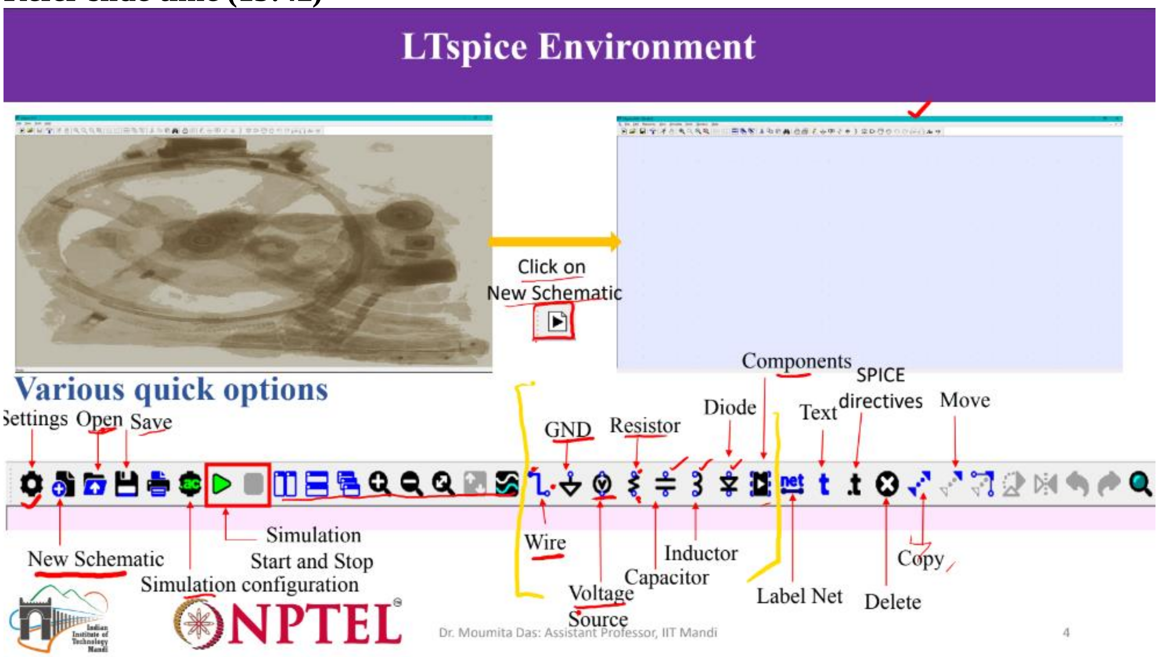
Refer slide time (12:14)



So, this is the website where you can actually download the software. So, you can see here it is written download for Windows 10 64-bit and forward and then the next one is download for Mac OS 10.15 and forward. So, for our system we are using the Windows one. So, Windows one we have already downloaded what you can do you can just press in the windows or mac whatever is suitable to you so download that particular software so you can see here so once i am pressing the downloading so it will download here we already have downloaded this particular software still i am just showing you how the all the complete process will be so that it will be easier for you to use so you can see here once you download then what you need to do you need to use this procedure like once you want to install this so this will be so since we already have this particular software downloaded in our PC so that is why this modify repair or remove is coming right so you can choose so like for your case it will be different so you can choose that and you can go to the next step okay then in that process you can complete the installation of this particular software and now once you install this so I am just exiting from here because you know we already have downloaded version in this so last step will be this finish step so once you complete the downloading process and the installing process you have to reach here the complete downloading and installation and then you have to finish this once you finish this then you have to open the software so this open software before opening the software just so this software window will look like this Okay, so you can open this software window.

Now let me just tell you what to do next. So you can pause here this lecture. So you can complete the downloading process, installing process. Then again you start the lecture in order to understand the next step. but if you don't have the software installed in your system then it will be very difficult to follow the next step because you know next step will be showing you the simulation in this particular software which you can do in your PC the basic circuit simulation basically RL kind of circuit how you can build the circuit how you can actually simulate that particular circuit so if you don't have this software in your system install it first then only you go to the next step in the lecture okay so because you know next three four lectures will be in this particular software only how to simulate this and how to see different types of noises so it will be in this particular software today i am just discussing the basic step So you can see here, so there are different icons are there.

Refer slide time (15:42)



So you can see this is the LTSpice environment. So here first you will be getting this kind of background or the background which i have shown you so then what you have to do i'll show you in the in the software also let me just explain it here first then i'll go to the software so you have to click on the new schematic part so new schematic part will be there which is given in this particular icon so you can see here this particular icon just let me use this this particular icon you can see here so here if you press then it will open new schematic because you know when you will be simulating anything so go to the new schematic part it will open this kind of window and the here you have to make the circuit okay so you can see there are different like symbols are given so these symbols are different things if you actually put your cursor on it so then it will tell you what it is for so just just go through the all the components so you can see this is for the settings so the first one this is for the settings and then this one you can see this is for the new schematic here so there are some places it is given in this way some places it is given in this way so if you like keep your cursor here so it will also show you new schematic so if the new schematic is given in this like symbol use this and open the new window first Then this open means like if you already have any file simulated in your system and which you want to open and see again or probably you want to work on that. So then you can just press open to open that particular simulation which is already there in your system. that you can just import and then use it in the simulation and then obviously save if you want to save the particular simulation that you can play so this is normal for like all kind of like windows so for there this save option will be given this then here comes the simulation configuration so if you press here then simulation you can actually configure and this particular this red highlighted portion here you can actually press start and stop in order to start the simulation or in order to stop the simulation if it is completed then the simulation will stop automatically like any other software but if you want to stop in between let's say simulation is taking long time you want to stop in between so that also possible to do by using this particular icon so this like in between icons this is to zoom in and to see different windows so those things so this particular things.

So this you can just go through. Now the important part comes like what to use in order to build a circuit. So you can see here this is the wire right. So this is the wire which you will be using for connection of electrical parameters. this work as electrical wire as if you are connecting electrical wire between any component so you need to use this particular wire and this is the ground connection if you have to use ground connection so generally like for circuit you can use simple ground so that you have to connect so if you have any other ground other than the circuit ground let's say for EMI generally we use ground other than the circuit common point. So, that ground provide us parasitic capacitance with respect to the circuit positive and negative point and which provide us noises.

So, then you have to connect the ground in order to get those things. Otherwise, you can connect this to the circuit common point for the simulation purpose. This is the voltage source as it is given here. So the voltage source obviously you can just use it for like connected as the input voltage source.

Similarly this is the resistance. So these symbols are similar to the actual circuit symbol. So you can just see it and easily you can understand. So you can see here, so these points are given. So like you can just take and then connect these different points which are actually highlighted in blue color. So these different points using the wires.

Right. now this is the capacitor so this is the capacitor part it is shown here and this is inductor this is diode and then these are the basic components which you need for any circuit right capacitor resistance inductor ground where and input source now the component where probably you have to provide input is the semiconductor devices. So, basically we are using in this particular lecture we are considering wide bandgap devices. But for some application you may need to use silicon device or maybe particular model of the silicon device or maybe particular model of the wide bandgap devices which you can actually find in the components or maybe if you have particular components let us say driver you are designing. So, if you are designing driver you need particular driver IC. So, then what you can do you can actually find different component in this particular library.

So, here different components are available whatever is not available which probably you need for your simulation that you can always download and import if it is available for LTSpice model. If it is not available, then you can actually build your own component by looking into the configuration of the component in the datasheet. That is also possible. So, there are two ways. The simple way is the download and import it.

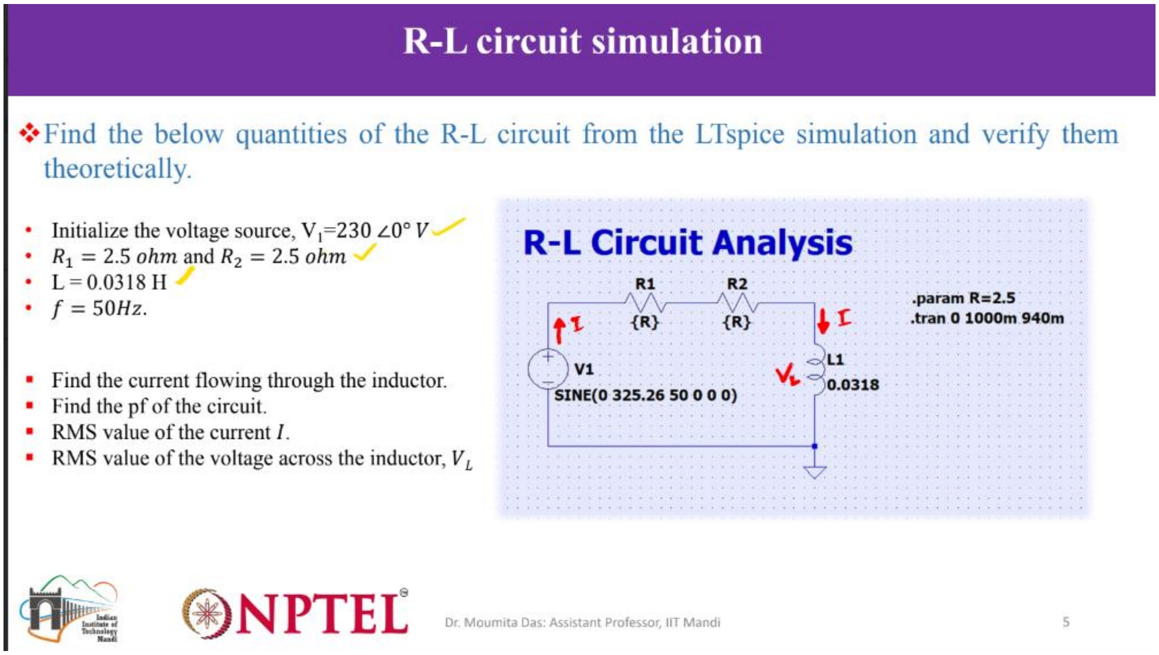
And some components are already there in the library. If it fits you, just use it in the simulation. then there are like these are the different things which you can just use like copy delete label need a spice directive text and all these things so this you can just look into so the part where we are mostly will be focusing once you install this software after that we are actually will be limited in this particular part to build the electrical network right so let us just see how it is in the actual software so I will just open the software part then we can actually see it from there so you can see here so all the windows here you can see in the top it is there so whatever I was just discussing with you say this where ground voltage, resistance, capacitance, inductor, diode and then the library. Then here in this case so you can see here this is settings. If you keep your cursor it will just show that what it is.

New schematic and open. So here we have to open the new schematic. The first step will be new schematic. Now Once you open the new schematic window, so the rest of the things it is now active. So, previously when you just open the software after downloading it, so this were not highlighted.

Means this were not in a condition to use it. So, this will be highlighted once you have the simulation window. So, here you can say this name of the window is written LTSpice draft 1. You can just save it as per your application name or maybe suitable name for you so that it will be easier for you to identify what kind of simulation it is.

Okay. So, now everything is highlighted. So, now we are actually in a position to build an electrical network. Right. So, next what we need to do is that now what we will be doing? We will be building a basic RL circuit. So basic circuit how it is working that is the first step which we will be looking into. So once we have the understanding of the basic circuit then only we can go to the next step.

Refer slide time (25:05)



simulation of the power electronic circuit because you know their switches will come and the things will be much more complicated than the basic circuit so you can see here so this parameter we are selecting for this rl circuit are given here so you can actually choose so this particular window which is shown here in this like dotted grid so this is the circuit window from the LTSpice software so this is after completing the network it will look like this right so you can just select the voltage source and voltage source you will be selecting from the voltage source which is given in the symbol in the LTSpice software and the then you will have the resistances and the inductance so the resistance you can just see r1 r2 it is connected and the values of the resistance will be as per your requirement and whatever you value you want to simulate it for and then L1 it is given some value and then this is the important thing this ground is connected. So, ground you have to connect otherwise the circuit simulation will not be working. So, ground part has to be connected whether it is connected to the common point of the circuit or maybe you have different ground then you can provide different ground name. Right. So the parameter here since this parameter so this r1 and r2 they are having same values.

So that is why this parameter it is defined as the r equals to 2.5 separately here. You can also select like the different value. If it is same value you can just use the common value.

Right. and then you can just you can see this is actually simulated for like this sine wave sinusoidal signal is given here so the signal different like values of the voltage magnitude peak value and the frequency everything is given here so then transient simulation where you want to simulate so that is also given here so you can actually see this is for simulating like RL circuit so different initialization is given here so initialize the voltage source this

V1 it is 230 0 degree voltage right degree you need to give if it is starting from 0 give 0 if it is starting from some other point give that particular degree right then R1 and R2 they are 2.5 ohms each as I told you And then L equals to 0.0318 Henry. So, whenever you are defining any parameter values, so that will be either if it is resistance it will be in ohm, if it is inductance it will be in Henry. If it is in kilo ohm or probably milli Henry, micro Henry, then you have to provide suitable value for that.

Means if it is kilo ohm, so then you have to write 2.5 multiplied by 10 to the power 3. Because you know the unit it will take automatically as ohm. You have to bring the value equivalent to ohm or Henry. You cannot write kilo, milli. mega micro all this kind of thing here right because some of the like simulation softwares they provide this unit so then you can just use that particular unit Here the units are like default units are this.

Now this frequency F is 50 hertz. It is taken for the input voltage. Now what you need to do? Once you build this simulation software you need to actually find the current flowing through the inductor. So the current through the inductor you have to find. Since it is series path the current whatever is flowing through the inductor it will be the same current which is coming from the source itself.

This source will have the same current. So first you have to find current through the inductor and see it in your simulation. That is the first thing. Then find the power factor of the circuit. So power factor of the circuit if you have to see, then what you need to do? You have to see this input voltage along with input current to see what is the power factor again.

And from there you have to calculate the power factor. This is the second thing you have to do. And third thing is the RMS value of the current I. so you may get sinusoidal kind of component then you have to find the rms from there and then rms value of the voltage across the inductor vl then now you got the current you have to find out the voltage here so these are the different parameters which you need to see in order to understand the simulation in this particular software so basic simulation right so just let me i'll come back to here let me go to the rl circuit in the software we have already built this rl circuit already there so we just open it so we already have kept the circuit so this is the circuit you can see here So, these resistances are taken from here. So, you can just press it here and then take it in the window.

Then you will get the resistance. You can see my screen. So, resistance if I am selecting then it is coming. And then similarly capacitance I am selecting it is coming. Then whatever you are connecting.

So, like taking. So, that what you have to do. So, let's say this is the capacitor. So, then you have to connect these two using the wire. Right. So, this is how you have to connect any component in the circuit.

Okay. I will just undo this part. you can just completely cut this using this okay so now this is the thing which i have shown to you so now you need to start the simulation in order to start the simulation where you have to go i have shown you the icon so you can see here so this run or pause so this is like the green highlighted icon it is shown here so you need to press here so if you press here press then it will run okay so now because you know this circuit is very simple so that then it will actually run and stop immediately so if it is like complicated circuit if it has many other like parameters or the switches everything included then it will take some time right now you have to see the current how you can see the current so then you have to use the current here so you can see this selection of the current and voltage probe you have to do so you can actually select here this current you can check and now if you if you want to see the inductor current it is the same current so you can just bring your signal so like across it so if you just move your cursor you can see this red kind of basically symbol so this will give you the voltage right if you just press it here so then it will give you the voltage you know so because you know across it will give you with respect to the ground so wherever you are actually placing so there voltage with respect to ground it will show so you can see here so this voltage is visible now if you want to see current current is the through through current right So through current means you have to bring the cursor in basically on top of the component so then it will show this kind of blue arrow and it is having blue arrow with this particular symbol so this is the current measurement symbol so this if you press just any any of the parameter then it will show current with respect to that particular parameter so here if i press it here it will show current with respect to this particular inductor now because it is series circuit it really doesn't matter whether you are seeing here or this particular resistance or this particular resistance so wherever you will be seeing the magnitude of the current will be same so now one important thing is that here once you are like holding this cursor so their arrow direction this blue arrow direction is from top to bottom right So, then the current whatever you want to see that will be coming from the input voltage source and it will be going to the ground in this direction only. So, the current direction will be similar to the actual current. Now, if you are seeing in the resistance the current direction is opposite. So, then if you want to see the actual direction of the current then you have to change the direction so means like you have to either put minus or you have to see whatever direction you are seeing it will be exactly opposite So that you can do. So now you can see these are the different signal which you can actually.

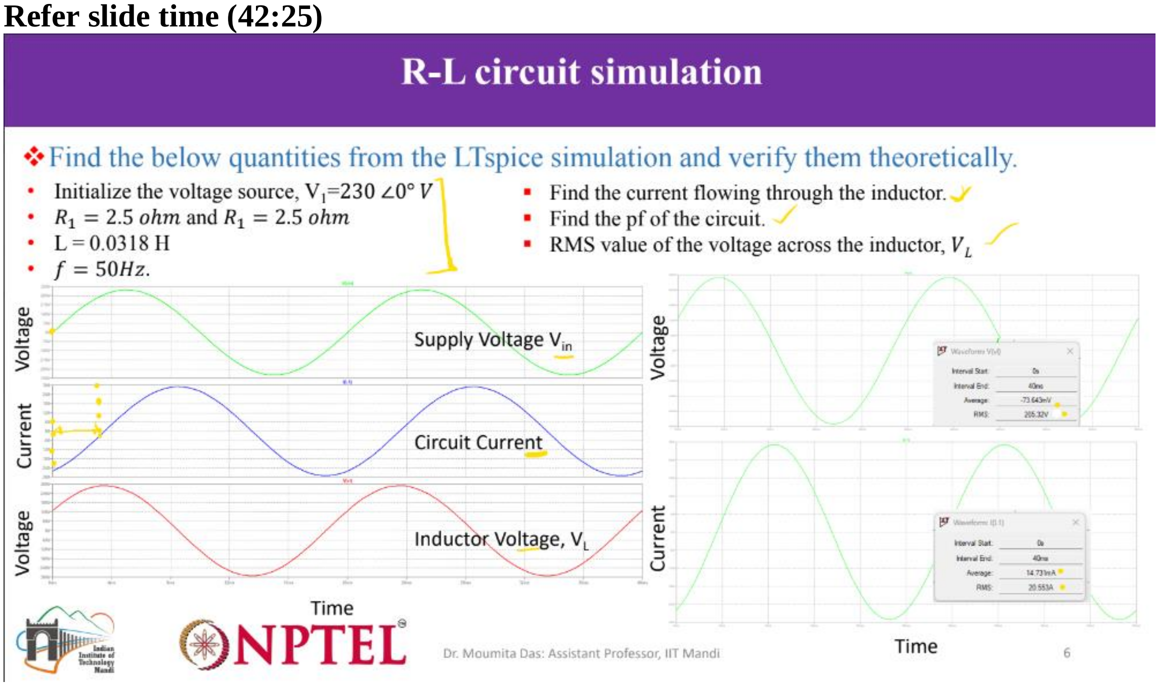
So you can see this cursor you can take and you can actually see different magnitude or the frequency whatever it is you want to see that you can see with respect to that. So you can see this is with respect to the voltage and this is with respect to the current. right so this is actually when the simulation is done the time is given from 0 to 40 millisecond now this time also you can change how to change this time so then what you have to do you have to go to this Simulation window and this is the part where basically you can see this we you have already seen this configured analysis configured analysis part where here you can actually see the stop time how much time you want to simulate it for here you can see like it although the stop time is given thousand millisecond so but the time from which the data is saved that is from 960 milliseconds. So, that is why in the window although the simulation it was running for 1000 millisecond, but in the simulation window it is only showing the simulation with respect to 960 to 1000 millisecond that is only 40 milliseconds. So, that the 0 which was visible in the window that is not the starting point of the converter simulation.

So, starting point there will be some transient part. So, this we have removed in order to see the steady state behavior. If you want to see the transient, you can save the data from 0 or maybe like only initial transient if you want to see like 0 to some 10 second, 10 millisecond or something like that, that you can do. Now there are actually different points are there. Start external DC supply, DC voltage source, stop simulation if steady state is detected. So those things you can just go through and understand and use it if it is required.

Then this is with respect to transient. So we are actually analysing using the transient analysis so there are different analysis also so ac analysis so ac behavior so basically compute the small signal ac behavior of the circuit linearized about its dc operating point so that information if you give so then you can actually analyze the circuit using this ac analysis or maybe you can see the simulation with respect to the ac analysis suppose you have analyzed your circuit for this AC analysis and now you want to just see the simulation then you can go for this part then DC sweep compute the DC operating point of a circuit while stepping independent sources and treating capacitances as open circuits and inductances as short circuit so this DC analysis or DC sweep part is also important for certain application then you can see here now the noise part you can see here perform a stochastic noise analysis of the circuit linearized about its DC operating point so you have to just see give the input output type of sweep number of points per octave you know like the in EMI we have gone through the magnitude with respect to different octaves when we have designed the filter if you remember for second order filter or the first order filter how much noise attenuation is possible per octave so that information you already have so this you can actually use here so noise in order to understand the noises in the circuit or if you want to simulate the circuit for EMI noises then the start frequency stop frequency in where exactly you want to see these noises so those you can do here so dc transfer find this is small signal transfer function DC output point. So, compute the DC operating point. So, this is operating point. Treating capacitances as open circuits and inductances as short circuit and transient frequency response.

So, perform a frequency response analysis transient simulation. This circuit must contain at least one analyzer. The detailed analysis parameters are configured in the for a device right so these are the different analysis generally for simulation we use this transient analysis we just give the stop time start time and it will simulate the circuit for that duration right and then we can see the response or the output with respect to that particular time So, if you want particular type of analysis then you can go for this different analysis part. So, this is possible here in this particular software that is one of the main advantages using this particular software for simulation. this is simulated so let us just see how this voltage like whatever like questions it is given with respect to the rl simulation how we can see this in the result so you can see here so in lt spice so basically so find the below quantities from lt spice simulation and verify the theoretical with the theoretical part so you can see this part this part already we have seen in the simulation how to provide this. so I request to all of you please simulate this in the RL circuit using this parameter and check whether your simulation output is matching with these results or not if it is not matching then probably there is some problem because you know we are just simulating the simple circuit it should match so use these parameters and find the current flowing through the inductor so you can see here since this is the voltage so green one first green one is the supply voltage which is the sinusoidal signal is given with respect to this voltage 230 volts and then frequency is 50 hertz so that is the supply voltage given if it is dc you could have given the degree 0 and the frequency is 0.

Refer slide time (42:25)



So then it will provide the DC voltage. So whatever is like your preference based on that you can simulate. But first simulate using this parameter just to see whether your simulation is working perfectly or not. Now the blue one which is coming.

from the input. So, you can see this green one and blue one. So, it is this blue one is the circuit current which is coming from the source itself. So, from here only you can see these phases are not matching. So, green one and blue one they are not in phase.

So, power factor is definitely not unity. Right. So the inductor voltage you can see. So the inductor current will be similar to the circuit current because it is the series network and the inductor voltage you can see here. So this is you can just this is given in the red color. so this is how you should be getting your voltage current and the supply voltage and magnitudes are given in the left hand side scale which you can just look into and check whether you are getting the same magnitude or not right here you can see the green one the peak magnitude is 325 and then here in the blue one it is actually close to 36 and then the red one which is like the inductor voltage it is close to 300 so that you can check now what we needed to do so find out current flowing through the inductor that you can see in the blue color current from the source find the power factor of the circuit So then if you have to find the power factor you have to actually see the angle between this voltage and the current. So the voltage you can see here so voltage in this case so you can check here so the voltage and the current show the phase difference how much it is coming.

So that you can do so you can see here so in the waveform interval starts from 0 second interval end at 40 millisecond. Similarly it is done with respect to the current interval start at 0 second and interval stop at 40 millisecond. so in this case you can check the voltage average voltage and the rms voltage are given in this particular window similarly current average current and the rms current is given in this particular window so rms value of the voltages you can anyway get from this particular window and rms value of the Current also you can get from this particular window now you have to see so basically So where this inductor current is going to 0 so wherever the inductor current is going to 0 so somewhere it will be There it will be 0 point will be there and the 0 point of the voltage is here So you have to find out the angle between these two These two angle how much it is coming So, this angle will be used to find out the power factor. So, since it is inductive kind of circuit, it is lagging in nature, right. So, like whatever angle you will be getting, so let us say it is around 60 degree or something, cos 60 will be the power factor.

this particular circuit. So, this angle you can very easily check in the window and then you can say this will be the power factor lagging in nature. Anyway, this power factor and all this thing I am assuming you will be knowing. If you do not know, go through the basics of it and try to understand and use it in the simulation itself. This is not where I am going to explain about this. You can just use this in the simulation to check whether the simulation is working properly or not.

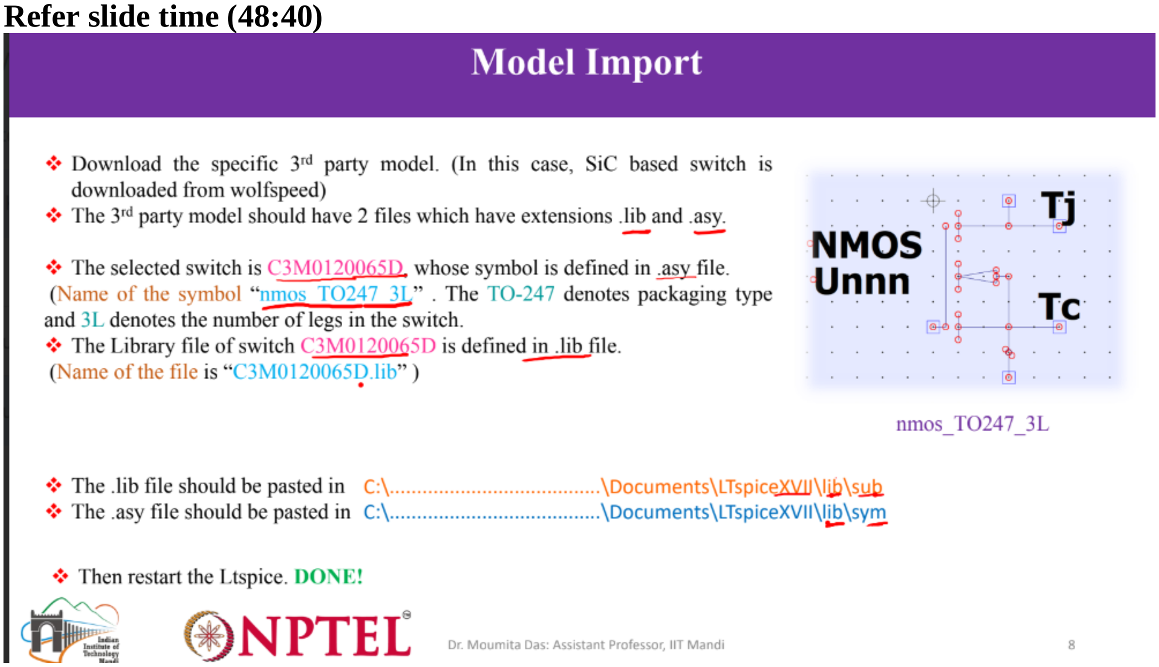
So, already you can see. So, these are the parameters. So, obviously some things are given already here. So, you can check from the simulation waveforms itself. So, these are the things. should be getting so il in the simulation it is coming 20.

553 in theoretical reasons it is around 20.57 and power factor it is 0.45 lagging in the simulation in theoretical 0.447 lagging and the Voltage VL in the simulation, it is the RMS value of the voltage 205.

32. It is coming in theoretical 205.71. So, I request you to simulate the circuit in the LT spice also calculate this parameter in the RL circuit using input voltage as it is given here and check whether you are getting this theoretical and this simulations or not simulations anyway you can understand from here theoretical you just need to write down the basic equations right so once you have the theoretical and the simulation parameters you can just compare you can see it is matching exactly equal so you can see if you if i am taking only one point so 20.5 is the current inductor current voltage 205.

205 so like point but if i remove so 205 around and the power factor 0.45 it is 0.447 means 0.45 So, it is matching exactly equal to the theoretical parameter. So, that you just check.

Refer slide time (48:40)



So, this model import part, so like if I use the actual model, so then what we have to do? We have to actually use this model. download the specific third-party model in this case silicon carbide base switch is downloaded from the speed so anything you can actually download and then you can import it so this part i will be discussing in the next class so model we can just take it so then we can consider it as third party model we can just import it in the library right so you can see this third i'm just telling you the basic part the simulation part i'll be showing in the next class the third party model should have two files which have extensions dot lib and dot asy you can find out any switch model of this extension and you can download that model of the device any switch like silicon carbide or gallium nitride and keep it in your PC so that in the next class when I will be explaining simulation of that then it will be easier for you to connect.

The selection of the switch in this case we have used the C3M0120065D. So, this symbol it is actually downloaded in .asy file. can use this dot lib and dot asy file so name of the symbol we kept it this the TO207 denotes packaging type and 3I denotes the number of legs in the switch so this symbol we have kept like this because this will give the packaging where how the packaging smd through hole what kind of packaging it is and the number of links so that you can just name this symbol and then save it the library file of switch this is defined in dot lib file and then name of the file is this okay so then this library file should be pasted so wherever your like software is like in our case it is in the c folder c folder under documents lt spice library and then Device sub device so and then dot asy file should be pasted in so there are two as I have explained two Extensions dot lib dot asy so dot asy should be pasted here in the blue color and the dot lib in the It is orange color.

So you can check it is a dot asy file is also pasted in the C folder. So, under document LTSpice. So, till this point it is same. So, if it is asy then it will be symbol and if it is .

lib then it will be sub. Okay. So, this is how you can actually import the device. You can either import the same device which we have given here so that it will be easier for you to find out. But if you have different device with you or the files of different devices available then download it and keep it in the folder in this particular way. right once you keep it this particular thing is done so then you need to restart the LT spice once this process is complete you need to restart the LT spice in order to consider this particular imported device in the library.

Okay. So just import this model part. First simulate the basic circuit. Any other circuit if you want to simulate just to check or clear your understanding in this software. And then import any device. And today you just try to do this part. In the next class, I will be discussing about how to use this imported device in the simulation. So, these are the references which you can follow for today's lecture.